68th Annual Meeting of the APS Division of Fluid Dynamics
Boston, Massachusetts
http://www.aps.org/units/dfd/meetings/meeting.cfm?name=DFD15
distribution, parameters which are critically important to better understand long-term hydrocarbon production from shale. We demonstrate that a poro medium composed of granular matter in a thin model fracture becomes heterogeneous and develops channels due to growth of fluid flow coupled with increase in porosity. Erosion is observed to progress through stick-slip events with larger events following lower wait times. Self-clogging is also observed where eroded particles collectively redeposit and jam within the channels, which are then stable to higher fluid fluxes. We model the spatial distribution of the flow within the medium using microscopic and macroscopic descriptions. Here, we study this problem experimentally, starting with the classic experiment of two-phase flow in a ductile porous rock. We solve the model semi-analytically at steady state, and numerically in general. We find that plastic deformation greatly reduces the maximum tensile stress, and that this maximum stress does not always occur at the wellbore. These results imply that hydraulic fracturing may fail in ductile rocks, or that the required injection rate for fracking may be much larger than the rate predicted from purely elastic models.

8:39AM A1.00004 High Speed Strain Measurements Surrounding Hydraulic Fracture in Brittle Hydrogel, WILL STEINHARDT, SHMUEL RUBINSTEIN, Harvard University — Hydraulic fractures of oil and gas shales occur miles underground, below complex, layered rocks, making measurements of their dynamics, extent, or structure difficult to impossible. Rocks are heterogeneous at a wide range of length scales, and investigating how these non-uniformities affect the propagation and extent of fractures is vital to improving both the safety and efficiency of hydraulic fracturing operations. To study these effects we have developed a model system using brittle, heavily cross-linked hydrogels that we can fracture with fluids and observe with a fast camera. By embedding tracer particles within the gel and using laser sheet microscopy, we obtain three dimensional stress and strain maps of the zone surrounding a hydraulic fracture tip. Gels can also be set in layers or interfaces with tunable strengths or with designed heterogeneities, allowing us to understand the fundamental science of hydraulic fractures and investigate the dynamics of controllably complex materials.

8:52AM A1.00005 Visualizing 3D fracture morphology in granular media, MARIE-JULIE DALBE, Civil And Environmental Engineering, Massachusetts Institute of Technology, RUBEN JUANES, Civil And Environmental Engineering, MIT — Multiphase flow in porous media plays a fundamental role in many natural and engineered subsurface processes. The interplay between fluid flow, medium deformation and fracture is essential in geoscience problems as disparate as fracking for unconventional hydrocarbon production, conduit formation and methane venting from lake and ocean sediments, and desiccation cracks in soil. Recent work has pointed to the importance of capillary forces in some relevant regimes of fracturing of granular materials (Sandnes et al., Nat. Comm. 2011), leading to different regimes of the invasion pattern. We present result for the 3D morphology of the invasion, with particular emphasis on the fracturing regime.

9:05AM A1.00006 Wettability and its impact on hydro-capillary fracturing in granular media, MATHIAS TROJER, PIETRO DEANNA, RUBEN JUANES, Massachusetts Inst of Tech-MIT — Two-phase flow in geologic porous media is important in many natural and industrial processes. While it is well known that wetting properties of porous media can vary drastically depending on the type of sandia and the pore fluids, the effect of wettability on capillary-driven fracturing continues to challenge our microscopic and macroscopic descriptions. Here we study this problem experimentally, starting with the classic experiment of two-phase flow in a horizontal Hele-Shaw cell filled with a granular medium. We inject a low-viscosity fluid into a thin bed of glass beads initially saturated with a fluid 350 times more viscous. The control parameters are the injection rate, the confining stress and the contact angle of the liquid-liquid-solid interface; carefully chosen fluid pairs allow us to cover the entire range from drainage to imbibition. We demonstrate that wettability exerts a powerful influence on the invasion/fracturing morphology of unfavorable mobility displacements. High time resolution imaging techniques allow us to quantify matrix displacement and fracture opening dynamics. Our results provide insights on fracture propagation and fracture length distribution, parameters which are critically important to better understand long-term hydrocarbon production from shale.

This material is based upon work supported by DOE Office of Science and Office of Basic Energy Sciences program under DE-FG02-13ER16401

1EPSRC, BP
2PhD Student

8:13AM A1.00002 Impact of ductility on hydraulic fracturing in shales, LUCY AUTON, CHRIS MACMINN, University of Oxford — Hydraulic fracturing is a method for extracting natural gas and oil from low-permeability rocks such as shale via the injection of fluid at high pressure. This creates fractures in the rock, providing hydraulic access deeper into the reservoir and enabling gas to be collected from a large area of the reservoir. Fracturing occurs at points where the perpendicular component of the fluid flux at the interface is the greatest. Adding a stochastic component to the model for the local erosion and deposition thresholds, we find the statistical features of the spatial development of heterogeneity to be consistent with those observed in the experiments.

8:26AM A1.00003 Fluid-driven fractures in brittle hydrogels, NIALL O’KEEFFE, PAUL LINDEN, Department of Applied Mathematics and Theoretical Physics, University of Cambridge — We study the physical mechanisms of fluid-driven fracture in low permeability reservoirs. This is done through the use of laboratory scale experiments on brittle heavily cross-linked hydrogels. These hydrogels have been shown to fracture similarly to “standard” brittle materials, such as PMMA and glass, which have been widely used to model geological mechanics. Crucially, the hydrogels are transparent, and permit fracturing at lower pressures and slower timescales. Their rheological properties can also be altered easily by varying the overall percentage of monomers and cross-linking molecules. Fracture dynamics are usually extremely hard to capture due to the fact that crack tips can approach material sound speeds. The sound speeds in these brittle hydrogels are 2-3 orders of magnitude less than in standard brittle materials. This allows us observe the complex fracture dynamics through the use of high speed camera techniques.

8:39AM A1.00004 High Speed Strain Measurements Surrounding Hydraulic Fracture in Brittle Hydrogel, WILL STEINHARDT, SHMUEL RUBINSTEIN, Harvard University — Hydraulic fractures of oil and gas shales occur miles underground, below complex, layered rocks, making measurements of their dynamics, extent, or structure difficult to impossible. Rocks are heterogeneous at a wide range of length scales, and investigating how these non-uniformities affect the propagation and extent of fractures is vital to improving both the safety and efficiency of hydraulic fracturing operations. To study these effects we have developed a model system using brittle, heavily cross-linked hydrogels that we can fracture with fluids and observe with a fast camera. By embedding tracer particles within the gel and using laser sheet microscopy, we obtain three dimensional stress and strain maps of the zone surrounding a hydraulic fracture tip. Gels can also be set in layers or interfaces with tunable strengths or with designed heterogeneities, allowing us to understand the fundamental science of hydraulic fractures and investigate the dynamics of controllably complex materials.

8:52AM A1.00005 Visualizing 3D fracture morphology in granular media, MARIE-JULIE DALBE, Civil And Environmental Engineering, Massachusetts Institute of Technology, RUBEN JUANES, Civil And Environmental Engineering, MIT — Multiphase flow in porous media plays a fundamental role in many natural and engineered subsurface processes. The interplay between fluid flow, medium deformation and fracture is essential in geoscience problems as disparate as fracking for unconventional hydrocarbon production, conduit formation and methane venting from lake and ocean sediments, and desiccation cracks in soil. Recent work has pointed to the importance of capillary forces in some relevant regimes of fracturing of granular materials (Sandnes et al., Nat. Comm. 2011), leading to different regimes of the invasion pattern. We present result for the 3D morphology of the invasion, with particular emphasis on the fracturing regime.

9:05AM A1.00006 Wettability and its impact on hydro-capillary fracturing in granular media, MATHIAS TROJER, PIETRO DEANNA, RUBEN JUANES, Massachusetts Inst of Tech-MIT — Two-phase flow in geologic porous media is important in many natural and industrial processes. While it is well known that wetting properties of porous media can vary drastically depending on the type of sandia and the pore fluids, the effect of wettability on capillary-driven fracturing continues to challenge our microscopic and macroscopic descriptions. Here we study this problem experimentally, starting with the classic experiment of two-phase flow in a horizontal Hele-Shaw cell filled with a granular medium. We inject a low-viscosity fluid into a thin bed of glass beads initially saturated with a fluid 350 times more viscous. The control parameters are the injection rate, the confining stress and the contact angle of the liquid-liquid-solid interface; carefully chosen fluid pairs allow us to cover the entire range from drainage to imbibition. We demonstrate that wettability exerts a powerful influence on the invasion/fracturing morphology of unfavorable mobility displacements. High time resolution imaging techniques allow us to quantify matrix displacement and fracture opening dynamics. Our results provide insights on fracture propagation and fracture length distribution, parameters which are critically important to better understand long-term hydrocarbon production from shale.
9:31AM A1.00008 Modeling the Effect of Fluid Flow on Growing Network of Fractures in a Porous Medium. MOHAMMED ALHASHIM, DONALD KOCH, Cornell Univ — The injection of a viscous fluid at high pressures in a geological formation induces the fracturing of pre-existing joints. Assuming a constant solid-matrix stress field, a weak joint saturated with fluid is fractured when the fluid pressure exceeds a critical value that depends on the joint’s orientation. In this work, the formation of a network of fractures in a porous medium is modeled. When the average length of the fractures is much smaller than the radius of a cluster of fractured joints, the fluid flow within the network can be described as Darcy flow in a permeable medium consisting of the fracturing network. The permeability and porosity of the medium are functions of the number density of activated joints and consequently depend on the fluid pressure. We demonstrate conditions under which these relationships can be derived from percolation theory. Fluid may also be lost from the fracture network by flow into the permeable rock matrix. The solution of the model shows that the cluster radius grows as a power law with time in two regimes: (1) an intermediate time regime when the network contains many fractures but fluid loss is negligible; and (2) a long time regime when fluid loss dominates. In both regimes, the power law exponent depends on the Euclidean dimension and the injection rate dependence on time.

9:44AM A1.00009 On the Study of Lifting Mechanism of a Soft Porous Media under Fast Compression1. QIANHONG WU, S. SANTHANAM, Villanova University, R. NATHAN, Penn State Berks, VUCBMSS TEAM — Fluid flow in a soft porous media under fast compressions is widely observed in biological systems and industrial applications. Despite much turbulence in the flow, the effects of pressure are not always well understood. In this study, we report on a series of experiments in which the behavior of a soft porous media is analyzed using a high pressure, high-speed state-of-the-art system that enables the visualization of the fluid flow and the determination of the forces generated by the lifting. We report herein a novel approach to treat the problem. The permeability of a soft porous media as a function of its compression was first measured. The material was then employed in a dynamic compression test using a porous-walled cylinder piston apparatus. The obtained transient compression of the porous media and the aforementioned permeability data were applied in different theoretical models for the pore pressure generation, which conclusively proved the validity of the consolidation theory developed by Wu et al. (JFM, 542, 281, 2005). Furthermore, the solid phase lifting force was separated from the total reaction force and was characterized by a new viscoelastic model, containing a nonlinear spring in conjunction with a linear viscoelastic Generalized Maxwell mechanical module. Excellent agreement was obtained between the experiment and the theory. Thus, the lifting forces from both the fluid and the solid were determined.

1This project is supported by NSF Grant 1511096.

Sunday, November 22, 2015 8:00AM - 9:57AM

Session A2 Suspensions: Rheology I 101 - Elisabeth Guazzelli, Aix-Marseille University

8:00AM A2.00001 Rheology of dense suspensions of non colloidal spheres in yield-stress fluids1. ELISABETH GUAZZELLI, SIMON DAGOIS-BOHY, Aix-Marseille Université, CNRS, IUSTI UMR 7343, 13453 Marseille, France, SARAH HORMOZO, Department of Mechanical Engineering, Ohio University, Athens, Ohio 45701-2979, USA, OLIVIER POULIQUEN, Aix-Marseille Université, CNRS, IUSTI UMR 7343, 13453 Marseille, France, AIX-MARSEILLE UNIVERSITE, CNRS, IUSTI UMR 7343 TEAM, DEPARTMENT OF MECHANICAL ENGINEERING, OHIO UNIVERSITY TEAM — Pressure-imposed rheometry is used to study the rheological properties of suspensions of non colloidal spheres in yield stress fluids. Accurate measurements for both the shear stress and particle normal stress are obtained in the dense regime. The rheological measurements are favourably compared to a model based on scaling arguments and homogenisation methods. The detailed account of this study can be found in Dagois-Bohy, S., Hormozi, S., Guazzelli, E., and Pouliquen, O. (2015). Rheology of dense suspensions of non-colloidal spheres in yield-stress fluids. Journal of Fluid Mechanics, 776, R2.


8:13AM A2.00002 Shear thickening regimes of non-Brownian suspensions. CHRISTOPHER NESS, JIN SUN, University of Edinburgh — We propose a unifying regime map for shear flows of dense suspensions of non-Brownian spheres that captures the onset of particle friction and particle inertia as distinct shear thickening mechanisms, while predicting quasistatic and soft particle rheology at high volume fractions and shear rates respectively. Discrete element method simulations reveal both mechanisms of shear thickening, and we show that they can be made to occur concurrently by careful manipulation of simulation parameters. Microstructural transitions associated with frictional shear thickening are precisely predicted, and we find very distinctive divergences of both the static and dynamic microstructure with respect to volume fraction in the thickened and non-thickened states.

8:26AM A2.00003 Hydrodynamic diffusion in non-colloidal suspensions: the role of interparticle forces. NICHOLAS HOH, ROSEANNA ZIA, Cornell University — Hydrodynamic diffusion in the absence of Brownian motion is studied via active microrheology in the purely hydrodynamic limit, to elucidate the transition from colloidal microrheology to the non-colloidal limit, falling-ball rheometry (FBR). Non-Brownian force-induced diffusion in FBR is strictly hydrodynamic in nature; in contrast, force-induced diffusion in colloids is deeply connected to a diffuse boundary layer even when Brownian motion is weak. To connect these two limits, we derive an expression for the hydrodynamic force-induced diffusion via the Smoluchowski equation, where thermal fluctuations play no role. The diffusion is anisotropic, along and transverse to the line of external force. The latter is zero owing to the fore-aft symmetry of Stokes flow. We find that interparticle forces play a crucial role in the hydrodynamic limit; accounting for this force results in longitudinal force-induced diffusion and an effective hydrodynamic diffusion coefficient of the form \(\alpha N^{1/2}\), where \(\alpha\) is the friction coefficient, \(N\) is the particle size, \(v_0\) is the characteristic velocity, and \(k\) the volume fraction, in excellent agreement with experiments and theory for macroscopic FBR. This model connects micro- and macro-scale rheology, and provides insight into the role of interparticle forces for diffusion and rheology even in the limit of pure hydrodynamics.
8:39AM A2.00004 A new state transition in the rheology of dense suspensions. RUJAN MAHARJAN, ERIC BROWN, Yale University — Dense suspensions of hard particles are known to have an effective viscosity that diverges as the packing fraction approaches the liquid-solid transition, $\phi_j$. This is typically measured based on energy dissipation in a steady state shear flow. In a Newtonian fluid, the same viscosity value also determines how long it takes for a flow to relax to a steady state after a change in control. By performing transient flow measurements in a rheometer, we find the transient viscosity of suspensions start to deviate from the steady state viscosity as the packing fraction increases above $\phi_j < \phi_j$. Further, we find the ratio of the normal stress to shear stress reaches a plateau $\sim 1$ for $\phi > \phi_j$. This identifies a new state transition.

8:52AM A2.00005 Rheology and particle dynamics near the flow-arrest transition: a constant stress and pressure approach. MU WANG, JOHN BRADY, California Institute of Technology — We use Brownian dynamics to investigate the relation between the rheology and the microscopic particle dynamics in dense colloidal dispersions at constant stress and pressure. For each imposed stress/pressure pair, the suspension exhibits distinct strain rate distributions depending on the observation time. We measure the long-time self-diffusivity (LTSD) corresponding to the strain rate (inverse shear viscosity) and find that the LTSD results at different imposed stresses collapse to master curves that depends only on the imposed pressure. For low-pressure suspensions, the stress-scaled LTSD diverges at a finite scaled strain rate due to its liquid-like behavior, while at high pressures the scaled LTSD emerges from zero due to the flow-arrest transition. On the other hand, we discover that the particle friction coefficient—the ratio of the particle shear stress to the particle (osmotic) pressure—is proportional to the strain rate scaled by the LTSD for all flowing suspensions. Our results demonstrate the effectiveness of the constant stress and pressure approach for dense suspension rheology, and show that, although the flow of amorphous materials is inherently far-from-equilibrium without a linear response regime, a mean-field description should remain valid.

9:05AM A2.00006 Concentration and velocity measurements in non-Brownian suspensions using ultrasonic imaging. BRICE SAINT-MICHEL, Ecole Normale Supérieure, Lyon, HUGUES BODIGUEL, STEVEN MEEKER, Laboratoire du Futur, Bordeaux, SEBASTIEN MANNEVILLE, Ecole Normale Supérieure, Lyon — We investigate the sedimentation and the re-suspension of dense, non-Brownian particles using ultrasonic imaging in a Couette cell, coupled to rheological measurements. Our setup records the characteristic speckle signal originating from the multitude of particles (PS, PMMA, glass) in the Couette cell scattering an initial ultrasound pulse on a transducer array. This speckle map can be used to estimate the velocity field and the particle concentration in our cell. We will discuss the advantages and limitations of our device using canonical examples when the suspending fluid is Newtonian: viscous re-suspension, sedimentation, particle migration by centrifugal forces and in Taylor vortices. Finally, some results involving non-Newtonian suspending fluids will be presented.

9:18AM A2.00007 Rheological measurements of liquid-solid flows with inertia. ESPERANZA LINARES, MELANY HUNT, California Institute of Technology, ROBERTO ZENIT, Universidad Nacional Autonoma de Mexico — This talk presents experimental measurements of effective viscosity for neutrally-buoyant suspensions in which the Reynolds numbers based on particle diameter varies from 1 to 1000 and for solid fractions from 10% to 50%. The measurements are conducted in a rough-walled, coaxial-cylinder rheometer. For Reynolds numbers from 1 to 100 and solid fractions less than 30%, the effective viscosities increase with Reynolds number and are comparable with recent numerical simulations found in the literature. For higher solid fractions, the effective viscosity shows shear thinning at the lowest shear rates, followed by thickening at higher shear rates. Over this range of Reynolds numbers for a pure fluid, the flow is laminar. At higher Reynolds numbers for a pure fluid, the flow transitions to turbulence. When particles are added under these flow conditions (particle Reynolds number greater than 100), the effective viscosity continues to increase with Reynolds number but with a greater magnitude. At the highest solid fractions, the effective viscosity is independent of shear rate.

9:31AM A2.00008 Characterizing dense suspensions: two case studies from the pharmaceutical industry. DAVID J. GOLDFARB, NAZIA KHAWAJA, IRINA KAZAKEVICH, HIMANSHU BHATTACHARJEE, MICHAEL HESLINGA, CHAD DALTON, Merck Research Laboratories — Liquid suspensions of Active Pharmaceutical Ingredient powders are present as pharmaceutical dosage forms in the form of oral suspensions and injectables. We present two case studies, both dense ($\sim 30-40\%)$ suspensions, in which the physical characterization of the product, specifically, particle size & shape and rheology were key to understanding the key product attributes as pertaining to the manufacturing process and to patient administration. For the one case study, an oral suspension, identifying variations in particle morphology during the wet milling of the product was key to the product understanding and development. Rheological measurements were applied as well. For the second case study, an injectable, results from different particle size measurement techniques and rheological measurements indicated the possibility of flocculation in a formulation. Additionally, measurements were obtained to assess the “injectability” of the product via rheometer and texture analyzer measurements and Poiseuille flow modeling. As a result, the relevant shear rate regime for this drug product administration was identified.

9:44AM A2.00009 Wall slip in suspensions of thermo-responsive particles. THIBAUT DIVOIX, CRPP, CNRS UPR6641 - Université de Bordeaux, VERONIQUE LAPEYRE, VALÉRIE RAVAIANE, ISM - UMR 5255 CNRS, Université de Bordeaux - ENSCBP, SEBASTIEN MANNEVILLE, Laboratoire de Physique de l'ENS de Lyon, Université de Lyon — Flows of suspensions are affected by wall slip, i.e. the fluid velocity $v_f$ in the vicinity of a boundary differs from the velocity $v_p$ of the latter due to the presence of a lubrication layer. Wall slip is quantified by the slip velocity $v_{sw}$, which is defined as $v_{sw} = v_f - v_p$ and displays a power-law scaling with the stress $\sigma$ at the wall. If the slip velocity of dilute suspensions robustly follows $v_{sw} \propto \sigma^p$ with $p \approx 1$, there is no consensus regarding denser suspensions that are sheared in bulk, for which $v_{sw}$ is reported to scale as a power-law of the stress with exponents inconsistently ranging between $p = 0$ and 2. By means of extensive rheometry coupled to velocimetry on a suspension of thermo-responsive particles, we show that such discrepancy is only apparent, and demonstrate that $v_{sw}$ scales as a power law of the viscous stress $\sigma - \sigma_c$, where $\sigma_c$ denotes the yield stress. Tuning the temperature reveals that such scaling holds true over a large range of packing fractions $\phi$ on both sides of the jamming point, and that the exponent $p$ increases continuously with $\phi$, from $p = 1$ (dilute suspensions) to $p = 2$ (dense assemblies). Our results pave the way for a unified description of wall slip.

1 Initial Support for Exploratory Projects (PEPS) scheme (ANR-10-IDEX-03-02)
8:00AM A3.00001 Particle size distribution effects in an irradiated turbulent gas-particle mixture , MONA RAHMANI, GIANLUCA GERACI, GIANLUCA IACCARINO, ALI MANI, Center for Turbulence Research, Stanford University — The effects of particle size distribution on thermodynamic and hydrodynamic behavior of solid particle solar receivers, that involve a turbulent mixture of gas and particles in a radiation environment, are investigated, using DNS with point particles. The turbulent flow is seeded with monodisperse and polydisperse particles, where the mass loading and total frontal area of particles are matched between the two systems. The results show that the variability of the Stokes number for polydisperse particles can significantly influence the particle clustering, and consequently the thermal behavior of the system. Polydisperse particles can influence the Taylor scale of the flow in the turbulent gas phase. Polydisperisty also implies variable thermodynamic and hydrodynamic properties of the particles. Our results show that the thermal behavior of the system with polydisperse particles is set by the integral measures for particle and gas momentum and thermal relaxation times.

8:13AM A3.00002 Motions of particles falling under gravity in a weakly turbulent Rayleigh-Bénard convection , SANGRO PARK, CHANGHOON LEE, Yonsei University — Motions of particles falling under gravity in a weakly turbulent convective flow within two parallel walls is studied numerically. Despite the importance of particle-laden convective flows, the vast majority of studies on Rayleigh-Bénard convection in the last decade have focused on single-phase fluids. Therefore detailed analysis on the behaviors of particles in Rayleigh-Bénard convection is required for fundamental understanding and practical purposes, i.e. prediction of precipitation, design of industrial cooling systems. In this study we use a direct numerical simulation using a pseudo-spectral method in a horizontally periodic channel. The particle motion is tracked by using four point-Hermite and fifth order-Lagrangian interpolation scheme. The flow condition is Rayleigh number $10^6$, Prandtl number 0.7 with a large aspect ratio 6. Particles are influenced by drag force by fluid and gravity for the range of Stokes number 0.01 - 10 and Froude number 0.45. Collisions of particles or force on fluid by particle are not considered. We found that weak particle clustering near the bottom wall is observed at large Stokes number, a similar behavior of particle alignment along gravitational direction in isotropic turbulence, whereas small Stokes number particles quickly follow the motion of thermal structures. The mechanism is discussed using probability density functions of particle locations and average distances between closest particles, etc.

8:26AM A3.00003 Mass transfer in a flow past a non-porous catalyst sphere , BO SUN, SUDHEER TENNETI, SHANKAR SUBRAMANIAM, Iowa State University — Mass transfer in a flow past a particle with a surface chemical reaction occurs in applications involving catalytic reaction. This type of the mass transfer problem has been analyzed by solving the convection-diffusion equation for Stokes flow (Acrivos et al. 1962) or flow at low Reynolds number (Taylor 1963, Gupalo et al. 1972). The objective of this study is to extend our understanding of this mass transfer problem to higher Reynolds number (up to 100) and assemblies of several particles by using particle-resolved direct numerical simulation (PR-DNS) of gas-solid flow. A uniform flow past a non-porous spherical particle with a first-order surface reaction is simulated. The non-dimensional reaction rate constant is the important parameter in the single particle case. The PR-DNS results at low Reynolds number for a single particle are first compared with 2D analytical solutions for concentration fields and the Sherwood number. Finally, the dependence of the concentration field on the non-dimensional reaction rate constant, and comparison of PR-DNS results with other Sherwood number correlations that use the Reynolds analogy to adapt Nusselt number correlations (which do not explicitly account for surface reactions) are explored at high Reynolds number.

8:39AM A3.00004 On the effect of the particle size distribution tails in irradiated turbulent gas-particle mixture , GIANLUCA GERACI, MONA RAHMANI, ALI MANI, GIANLUCA IACCARINO, Center for Turbulence Research - Stanford University — Previous investigations of irradiated particle-laden turbulent flows have shown that the heat transfer is sensitive to the particle size distribution when compared to its thermodynamical equivalent feed with homogeneous (monodisperse) particles. In this work we focus on the shape of the particles size distribution and, in particular, we show how the presence of long tails, at the same nominal mass loading, affects the heat transfer between the particles and gas. We use DNS of turbulence with point particles to show that a particle size distribution is able to mitigate the particle clustering and thus increases the efficiency of the energy transfer to the fluid. The addition of a right tail to the same particle size distribution, while redistributing the mass loading towards large particles, lowers the efficiency of the heat transfer to the gas. Furthermore, we show that the addition of a left tail has the opposite effect. The small particles increase the heat transfer, hence the average temperature of the gas at the outlet section. The simultaneous presence of both tails has the same impact on the behavior of the system as the inclusion of the right tail only, indicating the dominance of the large particles.

8:52AM A3.00005 Numerical Simulation of Shock Interaction with Deformable Particles Using a Constrained Interface Reinitialization Scheme , THOMAS L. JACKSON, PRASHANTH SRIDHARAN, University of Florida, JU ZHANG, Florida Institute of Technology, S. BALACHANDAR, University of Florida — In this work we present axisymmetric numerical simulations of shock propagating in nitromethane over an aluminum particle for post-shock pressures up to 10 GPa. The numerical method is a finite-volume based solver on a Cartesian grid, which allows for multi-material interfaces and shocks. To preserve particle mass and volume, a novel constraint reinitialization scheme is introduced. We compute the unsteady drag coefficient as a function of post-shock pressure, and show that when normalized by post-shock conditions, the maximum drag coefficient decreases with increasing post-shock pressure. Using this information, we also present a simplified point-particle force model that can be used for mesoscale simulations.

9:05AM A3.00006 Heat Transfer and Drag of a Sphere: Variable Density and Buoyancy Effects , SWETAVA GANGULI, SANJIVA LELE, Stanford University — How do forces acting on a particle change in the presence of significant heat transfer from the particle, a variable density fluid or gravity? We define unit problems isolating subsets of these phenomena and solve them via particle resolved simulations. Our investigations are agnostic to the Boussinesq regime and encompass both, the short time (acoustic) behavior and the subsequent nearly-incompressible flow field that is established. Defining $\lambda$ as the ratio of the initial particle-fluid temperature difference to the far-field fluid temperature, we observe that the particle size affects the acoustic response whereas $\lambda$ and $Re$ affects the low-Mach response. The heating of the fluid near the particle affects the drag significantly which is studied in a parameter space where $Re$, $\lambda$ and the Grashof number are varied. In the isothermal case, the drag computed numerically matches the drag correlation of Clift-Grace-Weber. For heated particles, using the density of the fluid at the particle surface in the correlation under-estimates the drag (e.g. by 30% when $\lambda = 1$), using the dynamic viscosity of the fluid at the particle surface over-estimates the drag (e.g. by 17 % when $\lambda = 1$) and using both still over-estimates the drag (e.g. by 13% when $\lambda = 1$). The deviations increase as $\lambda$ increases.

1This research is funded by the PSAAP II program at Stanford University
9:18AM A3.00007 Convergence of Beer’s Law for Radiation Transmission in Particle-Laden Turbulent Flows1. ARI FRANKEL, Stanford University, RICK RAUENZAHN, Los Alamos National Laboratory, GIANNI MELLO, ALI MANI, Stanford University — Discrete random particulate media have been shown to produce significant deviations from Beer’s law for radiation transmission. Though particle-resolved ray tracing models can exactly resolve the transmission, the computational expense of such approaches can be prohibitive in settings involving many particles where the radiative transfer equation must be solved at every time step. In this work we investigate the validity of projecting Lagrangian particles onto an Eulerian concentration field and using Beer’s law on a local basis. We take particle distributions produced from clustering in turbulent flows and perform both particle-resolved Monte Carlo ray tracing and Beer’s law computations. We show that the error in the calculated transmission decreases as the grid is refined, but that the homogenization error increases rapidly as the grid size approaches the particle diameter.

1This work was supported by the PSAAP2 program at Stanford University.

9:31AM A3.00008 Investigation of unsteadiness in Shock-particle cloud interaction: Fully resolved two-dimensional simulation and one-dimensional modeling. ZAHRA HOSSEINZADEH-NIK, JONATHAN D. REGELE, Iowa State University — Dense compressible particle-laden flow, which has a complex nature, exists in various engineering applications.Shock waves impacting a particle cloud is a canonical problem to investigate this type of flow. It has been demonstrated that large flow unsteadiness is generated inside the particle cloud from the flow induced by the shock passage. It is desirable to develop models for the Reynolds stress to capture the energy contained in vortical structures so that volume-averaged models with point particles can be simulated accurately. However, the previous work used Euler equations, which makes the prediction of vorticity generation and propagation inaccurate. In this work, a fully resolved two dimensional (2D) simulation using the compressible Navier-Stokes equations with a volume penalization method to model the particles has been performed with the parallel adaptive wavelet-collocation method. The results still show large unsteadiness inside and downstream of the particle cloud. A 1D model is created for the unclosed terms based upon these 2D results. The 1D model uses a two-phase simple low dissipation AUSM scheme (TSLAU) developed by [Hosseinzahe-Nik et al, 53rd AIAA Aerospace Sciences Meeting (2015)] coupled with the compressible two phase kinetic energy equation.

9:44AM A3.00009 Transient Simulation of Accumulating Particle Deposition in Pipe Flow. JAMES HEWETT, MATHIEU SELLIER, Department of Mechanical Engineering, University of Canterbury — Colloidal particles that deposit in pipe systems can lead to fouling which is an expensive problem in both the geothermal and oil & gas industries. We investigate the gradual accumulation of deposited colloids in pipe flow using numerical simulations. An Euler-Lagrangian approach is employed for modelling the fluid and particle phases. Particle transport to the pipe wall is modelled with Brownian motion and turbulent diffusion. A two-way coupling exists between the fouled material and the pipe flow; the local mass flux of depositing particles is affected by the surrounding fluid in the near-wall region. This coupling is modelled by changing the cells from fluid to solid as the deposited particles exceed each local cell volume. A similar method has been used to model fouling in engine exhaust systems (Paz et al., Heat Transfer Eng., 34(8-9):674-682, 2013). We compare our deposition velocities and deposition profiles with an experiment on silica scaling in turbulent pipe flow (Kokhanenko et al., 19th AFMC, 2014).

Sunday, November 22, 2015 8:00AM - 9:57AM – Session A4 Particle-Laden Flows: Turbulence Modulation

8:00AM A4.00001 Investigation of turbulence modulation in particle-laden flows using the lattice Boltzmann method. CHENG PENG, NICHOLAS GENEVA, HAODA MIN, LIAN-PING WANG, University of Delaware — Turbulent modulation by finite-size solid particles has been studied experimentally and numerically in the past several decades. Previous studies have revealed that resolving the interfaces between particle surfaces and fluid is crucial to properly include finite-size effects on local fluid turbulence. Finite-size particles also produce pseudo-turbulence that may not decay locally, leading to a stronger nonlinear dependence of the level of turbulence modulation on the particle volume fraction. In this study we apply the lattice Boltzmann method (LBM) to perform interface-resolved simulations of turbulent particle-laden flow, focusing on local turbulence dynamics at the scale of particle size. We will discuss the accuracy of this mesoscopic approach when compared to other macroscopic methods. We consider both fully developed homogeneous isotropic (HI) turbulent flows and turbulent channel flows laden with finite-size particles. The particle volume fraction is around 10% and the particle-to-fluid density ratio is of the order of one. Conditional statistics as a function of distance from the moving particle surfaces are studied in detail, and are used to help interpret global turbulence modulation by particles. Grid convergence of these conditional statistics will be discussed.

8:13AM A4.00002 Modulation to the compressible homogenous turbulence by heavy point particles: Effect of particles’ density1. ZHENHUA XIA, YIPENG SHI, SHIYI CHEN, Peking University — In this paper, two-way interactions between heavy point particles and forced compressible homogenous turbulence are simulated by using a localized artificial diffusivity scheme and an Eulerian-Lagrangian approach. The initial turbulent Mach number is around 1.0 and the Taylor Reynolds number is around 110. Seven different simulations of 10^6 particles with different particle densities (or Stokes number) are considered. The statistics of the compressible turbulence, such as the Mach number, kinetic energy, dilatation, and the kinetic energy spectra, from different simulations are compared with each other, and with the one way undisturbed case. Our results show that the turbulence is suppressed if the two-way coupling backward interactions are considered, and the effect is more obvious if the density of particles is higher. The kinetic energy spectrum at larger Stokes number (higher density) exhibits a reduction at low wave numbers and an augmentation at high wave numbers, which is similar to those obtained in incompressible cases. The probability density functions of dilatation, and normal upstream Mach number of shocklets also show that the modulation to the shocklet statistics is more apparent for particles with higher density.

1We acknowledge the financial support provided by National Natural Science Foundation of China (Grants Nos. 11302006, and U1530107)
production across all spatial scales. A direct energy source/sink, which is highly wavenumber and Stokes number dependent and (2) through a potentially severe reduction in TKE on turbulence. It will be shown in this talk that particles modify the surrounding turbulence in two distinct but related ways: (1) through a Stokes (St = [1, 10, 100] based on the centerline Kolmogorov scale) numbers in order to understand the spectral extent of the particle influence with Lagrangian point particles, are used to compute spectral energy budgets at several Reynolds (friction Reynolds numbers up to 900) and in subgrid or Eulerian-based models. In this work, data from direct numerical simulations of turbulent planar Couette flow, two-way coupled turbulence, how this is a function of the particle inertia relative to flow timescales, and how this potentially upscale influence should be modeled dynamics and turbulent motions. An important question which remains unresolved, however, is the scale over which small particles modify the turbulence has been studied extensively in both numerical and experimental contexts, and results from a complex interaction between particle bounded turbulence term to account for the interaction between the boundary layer flow and the droplets. Our presentation will focus on several comparisons approach with the raindrops being considered as non-deformable, non-spinning and non-interacting droplets. We employ an inter-phase coupling acceleration leads to boundary layer momentum loss. We model the airflow within a flat plate boundary layer using a Lagrangian-Eulerian work focuses primarily on the numerical quantification of boundary layer momentum loss caused due to raindrops. The collision of raindrops momentum and the loss of vehicle momentum due to its collision with the raindrops are the primary reasons causing the drag to increase. Our challenge especially for slow moving vehicles such as airships. Effective roughening of airfoil surface caused by an uneven water film, loss of flow that aircrafts operating under heavy rainfall conditions face deterioration of lift and increase in drag. This scenario can be a critical design observation by Li et al. (2012). The root-mean-squared vorticity, turbulence production and viscous dissipation are modified in a similar manner bottom wall and reduced in the outer flow region due to the presence of particles, consistent qualitatively with the previous experimental investigation by Mahesh et al. [J. Comput. Phys. (2004) 197:215-240]. Several cases are presented illustrating the method’s efficacy for studying particle-laden flow. Included are cases featuring freely moving particles within turbulent fluid flow.

1This work is supported by the DOE.

Effects of finite-size particles on the turbulent flows in a square duct1. ZHAOSHENG YU, ZHAOWU LIN, XUEMING SHAO, State Key Laboratory of Fluid Power Transmission and Control, Department of Mechanics, Zhejiang University, Hangzhou 310027, China; LIAN-PING WANG, Department of Mechanical Engineering, University of Delaware, Newark, Delaware 19716, USA — Fully resolved numerical simulations of the particle-laden turbulent flows in a square duct are performed with a direct-forcing fictitious domain method. The effects of the finite-size particles on the mean and root-mean-square (RMS) velocities are investigated at the friction Reynolds number of 150 (based on the friction velocity and half duct width) and the particle volume fractions ranging from 0.78% to 7.07%. For the neutrally buoyant case, our results show that the mean secondary flow is enhanced and its circulation center shifts closer to the center of the duct cross-section when the particles are added. The reason for the particle effect on the mean secondary flow is analyzed by examining the terms in the mean streamwise vorticity equation. The particles enhance the wall-tangential component of the RMS velocity (i.e. Reynolds normal stress) more than its wall-normal component in the near-wall region near the corners, resulting in the enhancement in the gradients of the normal stress difference, which we think is mainly responsible for the enhancement in the mean secondary flow. The particles accumulate preferentially in the near-corner region in the neutrally buoyant case. In addition, the effects of particle sedimentation are examined at different Shields numbers.

1The work was supported by the National Natural Science Foundation of China (11372275) and Research Fund for the Doctoral Program of Higher Education of China (2013010110035).

Near-wall turbulence modification by small, heavy particles in a horizontal channel flow. JUNGHOOON LEE, CHANGHOON LEE, Yonsei University — Near-wall turbulence modification by particles in a horizontal channel flow is investigated via direct numerical simulation coupled with a point-force approximation for small, heavy particles with a diameter smaller than the Kolmogorov length scale of the flow. The Stokes numbers considered are 0.72, 0.81 and 5.3 in wall units and the Froude number is much smaller than 1, indicating that the influence of gravity on particle motion is strong. Particle-particle collisions are not taken into account to focus on the interactions of particles with turbulence. Water droplets in air turbulence are considered. When a particle touches the wall, it is removed, and a new particle is introduced at a random location, with a velocity identical to the fluid velocity at the new position, maintaining a constant particle mass loading. It is shown that the turbulence intensities are enhanced near the bottom wall and reduced in the outer flow region due to the presence of particles, consistent qualitatively with the previous experimental observation by Li et al. (2012). The root-mean-squared vorticity, turbulence production and viscous dissipation are modified in a similar manner to the turbulence intensities. The physical mechanisms responsible for this turbulence modification behavior are discussed by examining the modification of coherent turbulent structures.

Experimental evidence shows that aircrafts operating under heavy rainfall conditions face deterioration of lift and increase in drag. This scenario can be a critical design challenge especially for slow moving vehicles such as airships. Effective roughening of airfoil surface caused by an uneven water film, loss of flow momentum and the loss of vehicle momentum due to its collision with the raindrops are the primary reasons causing the drag to increase. Our work focuses primarily on the numerical quantification of boundary layer momentum loss caused due to raindrops. The collision of raindrops with a solid surface leads to formation of an ejecta fog of splashed back droplets with their sizes being of the order of micrometers and their acceleration leads to boundary layer momentum loss. We model the airflow within a flat plate boundary layer using a Lagrangian-Eulerian approach with the raindrops being considered as non-deformable, non-spinning and non-interacting droplets. We employ an inter-phase coupling term to account for the interaction between the boundary layer flow and the droplets. Our presentation will focus on several comparisons (velocity field, lift and drag at various angles of attack) with the results of the standard (rain-free) Prandtl boundary layer flow.

1Indian Institute of Technology, Delhi

The phenomenon of turbulence modification in wall-bounded turbulence has been studied extensively in both numerical and experimental contexts, and results from a complex interaction between particle dynamics and turbulent motions. An important question which remains unresolved, however, is the scale over which small particles modify the turbulence. How this is a function of the particle inertia relative to flow timescales, and how this potentially upscale influence should be modeled in subgrid or Eulerian-based models. In this work, data from direct numerical simulations of turbulent planar Couette flow, two-way coupled with Lagrangian point particles, are used to compute spectral energy budgets at several Reynolds (friction Reynolds numbers up to 900) and Stokes (St = [1, 10, 100] based on the centerline Kolmogorov scale) numbers in order to understand the spectral extent of the particle influence on turbulence. It will be shown in this talk that particles modify the surrounding turbulence in two distinct but related ways: (1) through a direct energy source/sink, which is highly wavenumber and Stokes number dependent and (2) through a potentially severe reduction in TKE production across all spatial scales.
8:00AM A5.00001 Inside out: Speed-dependent barriers to reactive mixing, DOUGLAS KELLEY, THOMAS NEVINS, University of Rochester — Reactive mixing occurs wherever fluid flow and chemical or biological growth interact over time and space. Those interactions often lead to steep gradients in reactant and product concentration, arranged in complex spatial structures that can cause wide variation in the global reaction rate and concentrations. By simultaneously measuring fluid velocity and reaction front locations in laboratory experiments with the Belousov-Zhabotinsky reaction, we find that the barriers defining those structures vary dramatically with speed. In particular, we find that increasing flow speed causes reacting regions to move from vortex edges to vortex cores, thus turning the barriers “inside out”. This observation has implications for reactive mixing of phytoplankton in global oceans.

8:13AM A5.00002 A Gibbs Formulation for Reactive Materials with Phase Change, D. SCOTT STEWART, Mechanical Science and Engineering, University of Illinois, Urbana, IL — A large class of applications have pure, condensed phase constituents that come into contact, chemically react and simultaneously undergo phase change. Phase change in a given molecular material has often been considered to be separate from chemical reaction. Continuum models of phase change often use a phase field model whereby an indicator function is allowed to change from one value to another in regions of phase change, governed by evolutionary (Ginzburg-Landau) equations, whereas classic chemical kinetics literally count species concentrations and derive kinetics evolution equations based on species mass transport. We argue the latter is fundamental and is the same as the former, if all species, phase or chemical are treated as distinct chemical species. We pose a self-consistent continuum, thermo-mechanical model to account for significant energetic quantities with correct molecular and continuum limits in the mixture. A single stress tensor, and a single temperature is assumed for the mixture with specified Gibbs potentials for all relevant species, and interaction energies. We discuss recent examples of complex reactive material modeling, drawn from thermic and propellant combustion that use this new model.

8:26AM A5.00003 Propagation of symmetric and non-symmetric lean hydrogen flames in narrow channels: influence of heat losses, CARMEN JIMENEZ, VADIM KURDYUMOV, CIEMAT — Direct numerical simulations, including detailed chemistry and transport, are used to investigate the structure and stability of freely propagating lean hydrogen flames in planar narrow channels. Depending on the flame burning rate and the wall properties, the flame-wall heat exchange can result in flame extinction. For large heat losses only the fastest burning flames, corresponding to fast reactant flowing rates can propagate. We study how the resulting flame solutions, symmetric and non-symmetric, can coexist for the same set of parameters. The symmetric solutions are calculated imposing symmetric boundary conditions in the channel mid-plane and when this restriction is relaxed non-symmetric solutions develop. This indicates that the symmetric flames are unstable to non-symmetric perturbations, as predicted before within the context of a constant density model. Moreover, the burning rates of the non-symmetric flames are found to be significantly larger than those of the corresponding symmetric solution and therefore the range of conditions for flame extinction and flashback also differ. This shows that assuming in CFD that the flame should reproduce the symmetry of the cold flow can have important safety implications in micro scale combustion devices burning lean hydrogen mixture.

8:39AM A5.00004 Fluid-Plasma Coupling in Hydrogen Flames, LUCA MASSA, JONATHAN RETTER, NICK GLUMAC, GREGG ELLIOT, JONATHAN FREUND, University of Illinois — Recent experiments show that hydrogen diffusion flames at low Reynolds number can be markedly affected by a dielectric barrier discharge (DBD) plasma. The flame surface deforms and flattens, and light emissions increase. We develop a simulation model to analyze the mechanisms that causes these changes, and apply it to numerical calculations of axisymmetric flames with co-annular DBD, matching the corresponding experiments. Body forces due to charge sheaths are found to be the main mechanism, with radicals produced by plasma excitation playing a secondary role for the present conditions. The non-actuated flame flickers at approximately 10 Hz, in good agreement with the experiments. As the DBD voltage is increased, the flame flattens and oscillations decrease, eventually ceasing above a threshold value. The fully flattened case has a stoichiometric surface lying flat across the fuel orifice, with flame temperature exceeding significantly the adiabatic flame value. A force based on a linearized plasma sheath flattens the flame and acts as a boundary condition, and presents a good estimate for the threshold flattening potential. In faster flowing regimes, radical production by the plasma becomes more important.

8:52AM A5.00005 Supersonic Particle Impacts: Cold Spray Deposition of Polymeric Material, TRENTON BUSH, DAVID SCHMIDT, JONATHAN P. ROTHSTEIN, UMass Amherst — When a solid, ductile particle impacts a substrate at sufficient velocity, the resulting heat, pressure, and plastic deformation at the interface can produce bonding. The use of a supersonic gas flow to accelerate such particles is known as Cold Spray deposition. The Cold Spray process has been commercialized for some metallic materials, but further research is required to unlock the exciting material properties possible with polymeric compounds. In this work, we present a combined computational and experimental study whose aim is to define the necessary flow conditions for a convergent-divergent de Laval nozzle to produce successful bonding in a range of polymers. From our initial exploration of temperature-pressure space, we will reveal a material dependent ‘window of deposition’ where successful deposition is possible. Furthermore, we will present our computational work on the development of an optimized nozzle profile that maximizes particle total energy (kinetic plus thermal) upon impact and thus maximizes the likelihood of successful deposition. These predictions will be confirmed by the experimental results presented.

9:05AM A5.00006 SPIV study of two interactive fire whirls, KATHERINE HARTL, Princeton University, ALEXANDER SMITS, Princeton University, Monash University — Fire whirls are buoyancy-driven standing vortex structures that often form in forest fires. Capable of lifting and ejecting flaming debris, fire whirls can hasten the spread of fire lines and start fires in new places. Here we study the interaction of two jets in an externally applied circulation as an introduction to the study of two interacting fire whirls. To study this interaction we use two burner flames supplied with DME and induce swirl by entraining air through a split cylinder that surrounds both burners. Three components of velocity are measured using Stereo Particle Image Velocimetry both inside and outside the fire whirl core, at the base, midsection, and above the top of the fire whirls. The effects on the height and circulation on the distance between the burners, the rate of fuel supplied to the burners, and the gap size, are examined.

9:18AM A5.00007 System Modeling for Ammonia Synthesis Energy Recovery System, GABRIELA BRAN ANLEU, PIRUV KAVEHPOUR, ADRIENNE LAVINE, University of California, Los Angeles, AMMONIA THERMOCHEMICAL ENERGY STORAGE TEAM — An ammonia thermochemical energy storage system is an alternative solution to the state-of-the-art molten salt TES system for concentrating solar power. Some of the advantages of this emerging technology include its high energy density, no heat losses during the storage duration, and the possibility of long storage periods. Solar energy powers an endothermic reaction to disassociate ammonia into hydrogen and nitrogen, which can be stored for future use. The reverse reaction is carried out in the energy recovery process; a liquid ammonia mixture flowing through a porous catalytic ammonia synthesis reaction. The goal is to use the ammonia synthesis reaction to heat supercritical steam to temperatures on the order of 650°C as required for a supercritical steam Rankine cycle. The steam will flow through channels in a combined reactor-heat exchanger. A numerical model has been developed to determine the optimal design to heat supercritical steam while maintaining a stable exothermic reaction. The model consists of a transient one dimensional concentric tube counter-flow reactor-heat exchanger. The numerical model determines the inlet mixture conditions needed to achieve various steam outlet conditions.
9:31AM A5.00008 Uphill diffusion and phase separation in partially miscible multicomponent mixtures¹

Debby R. Chakravorty, Sabina G. Lauffenburger, University of Illinois, CHICKASAW COLE, University of Michigan, ROBERT BEHRINGER, Duke University — The partially miscible multicomponent mixtures, which are frequently encountered in preclinical processes, often exhibit complicated behaviors, and are critical to the production rate, energy efficiency, and pollution controls. Recent studies have been mainly focused on phase behaviors. However, the coupled phase equilibrium and transport process, which may be the answer to phase separations observed in experiments, is not well researched. Here, we present a numerical and theoretical study on coupled mixing of heavy oil and supercritical water, and the results of our state-of-art modeling agree with experimental measurements. We find that due to the non-ideal diffusion driving force, (1) strong uphill diffusion of heavy oil fractions occurs, (2) a new heavy oil phase is separated starting from the plait point, and heavy fractions become highly concentrated, and (3) water diffusion initially overshoots in oil, and is expelled lately. Finally, we conclude our analysis applicable to different molecules and conditions.

¹The authors thank Saudi Aramco for supporting this work (contract number 6600023444).

9:44AM A5.00009 Wind Tunnel Testing of a Hydrogen Jet in a Turbulent Crossflow Altered by a Dielectric Barrier Discharge¹

Ping He, Ashwin Raghavan, Ahmed Ghoniem, Massachusetts Inst of Tech-MIT — It has been demonstrated that plasmas can fundamentally alter the combustion process. The radical production can decrease combustion timescales and the body force produced by the driving electric currents can improve fuel/oxygen mixing and alter the shape of the steady state flame. We study these mechanisms for a fuel jet exhausting into a well-characterized turbulent cross-flow of air upon a Dielectric Barrier Discharge (DBD) plasma produced at the jet exit. The fuel is hydrogen diluted in cases with N₂ and Ar. Laser breakdown provides the energy deposition for ignition above the jet. The likelihood of sustained ignition for various fuel compositions and cross-flow conditions is considered along with flame properties once ignited both under the influence of the DBD plasma and without. Additionally, the effect of the DBD on flame blow-off is investigated. The jet is varied from low-momentum ratios (∼ 10⁻²) to high (∼ 1) to alter the relative contributions of the body forces and radical production on the combustion process. This system is studied to quantify the effect of the DBD plasma and discover opportunities for control.

¹This material is supported by the DOE, NNSA, Award DE-NA0002374.

Sunday, November 22, 2015 8:00AM - 9:57AM — Session A6 Granular Flows: Force Transmission 105 - Joshua Dijksman, Wageningen University

8:00AM A6.00001 Quantitative Comparison of Experiments and Numerics in Granular Materials

Joshua Dijksman, Wageningen University, Jie Ren, Merck & Co., Robert Behringer, Duke University, Lenka Kovalcinova, Lou Kondic, New Jersey Institute of Technology, MIRO KRAMAR, Konstantin MischaiKow, Rutgers University — It is challenging to experimentally probe the microstructure of sheared granular media. Comparisons of numerical results to experiments are thus rare. We present a direct match of experimental and discrete element method results on a sheared two-dimensional granular system. We compare the micro and mesostructural properties of the packing using several different metrics, among which measures that quantify the role of the force network topology. We find a quantitative match in the experimental and numerical approaches, and our results surprisingly indicate that the number of rattlers in the packing is a robust indicator of the mechanical and topological composition of the packing.

8:13AM A6.00002 Local to global avalanches in sheared granular materials

Dengming Weng, Lanzhou University, Dong Wang, Duke University, Thibault Bertrand, Yale University, Jonathan Bares, Duke University, Bob Behringer, Duke University — Commonly, granular materials yield or flow if sufficiently large shear stress is applied, leading to avalanche-like behavior. Rearrangement phenomenon can produce dramatic events like snow avalanches, landslides or earthquakes. For experimentally sheared media, we seek to understand the dynamics of the grain rearrangements from the local to the global scale. In this work, force networks and displacement fields are measured on two-dimensional sheared material for cyclically sheared photoelastic circular particles. Avalanches, their size, location and duration are extracted at the global scale from the rapid variation of the macroscopic energy stored in the system whereas at the local scale they are measured from the energy drop, displacement and rotation of each particle. Statistics of those different quantities are computed and correlated to test their intrinsic entanglement and analyze their universal dynamics. These results are quantitatively different from what has been observed for different analytic coarse-grained approaches and permit a clear measurement of the effect of the packing fraction and inter-particle friction coefficient on the statistical behavior.

8:26AM A6.00003 Avalanches and local force evolution in a granular stick-slip experiment¹

Aghil Abed Zadeh, Jonathan Bares, Robert Behringer, Duke University — We carry out experiments to characterize stick-slip for granular materials. In our experiment, a constant speed stage pulls a slider which rests on a vertical bed of circular photoelastic particles in a 2D system. The stage is connected to the slider by a spring. We measure the force on the spring by a force sensor attached to the stage. The distribution of avalanches during the experiment is presented, and the frequency of occurrence of avalanches is estimated. The power spectrum of the force signal to understand the effect of the loading speed and of the spring stiffness on the statistical behavior of the system. From a more local point of view and by using a high speed camera and the photoelastic properties of our particles, we characterize the internal granular structure during avalanches. Image processing and analyzing the skeleton of force network inside the media, we try to understand the flow of particles and evolution of force chains inside the media and during avalanches.

¹NSF DMR1206351, NASA NNX15AD38G, and the W. M. Keck Foundation

8:39AM A6.00004 Force network in a three-dimensional sheared material

Nicolas Brodo, Inria Bordeaux, Jonathan Bares, Duke University, Joshua Dijksman, Wageningen University, Bob Behringer, Duke University — Force chains in 2D granular media have been widely studied over the past decade. However the force network evolution when a 3D granular medium is sheared remains poorly understood due to the complexity of carrying out experimental observations. We present experiments using a novel apparatus to measure particle motion and inter-particle forces in the case of the quasi-static deformation of a 3D sphere packing subject to shear and compression. We perform these experiments on slightly polydisperse and very low-friction soft hydrogel spheres. We resolve the microscopic force network in a three dimensional packing by imaging the entire packing at each loading steps using a laser scanning technique. By resolving particle deformations via custom image analysis software, we extract all particle contacts and contact forces with a very good accuracy. We observe an increase of pressure, P, the Reynolds pressure during shear. This rise in pressure is associated with the evolution of the microscopic force network. The flow of particles is also investigated and correlated to the evolution of the network.
Forces and flows during high speed impacts on a non-Newtonian suspension. MELODY LIM, JONATHAN BARES, ROBERT BEHRINGER, Duke University — Above a certain mass fraction of particle suspensions of dense granular particles in water exhibit non-Newtonian behavior, including impact-activated solidification. Although it has been suggested that the solidification of the suspension depends on interactions with the suspension boundary, quantitative experiments on the forces experienced by the boundaries of the suspension have not been reported. In the present experiments, we determine the magnitude and timings of impactor-driven events in both the boundaries and body of the suspension using high-speed video, tracer particles, and photoelastic contact imaging. We observe a shock front propagation during impacts. The dynamics of this shockfront are strongly correlated to the dynamics of the intruder. Additionally, we observe a second extremely fast shockfront, associated with the propagation of forces to the boundaries of the suspension. The dynamics of this shockfront do not depend on the intruder dynamics, but are correlated to the volume fraction of cornstarch particles in the suspension.

Fluid mechanical scaling of impact craters in unconsolidated granular materials. COLIN S. MIRANDA, DAVID R. DOWLING, Dept. of Mech. Eng., Univ. of Mich., Ann Arbor MI 48109 — A single scaling law is proposed for the diameter of simple low- and high-speed impact craters in unconsolidated granular materials where spall is not apparent. The scaling law is based on the assumption that gravity- and shock-wave effects set crater size, and is formulated in terms of a dimensionless crater diameter, and an empirical combination of Froude and Mach numbers. The scaling law involves the kinetic energy and speed of the impactor, the acceleration of gravity, and the density and speed of sound in the target material. The size of the impactor enters the formulation but divides out of the final empirical result. The scaling law achieves a 98% correlation with available measurements from drop tests, ballistic tests, missile impacts, and centrifugally-enhanced gravity impacts for a variety of target materials (sand, alluvium, granulated sugar, and expanded perlite). The available measurements cover more than 10 orders of magnitude in impact energy. For subsonic and supersonic impacts, the crater diameter is found to scale with the 1/4- and 1/6-power, respectively, of the impactor kinetic energy with the exponent of the exponent crossover occurring near a Mach number of unity. The final empirical formula provides insight into how impact energy partitioning depends on Mach number.

A theoretical study of burrowing in dry soil using razor clam-inspired kinematics. AMOS WINTER, MONICA ISAVA, MIT — This work investigates whether the digging kinematics of Ensis directus, the Atlantic razor clam, could be utilized in dry soil. In wet soil, E. directus uses contractions of its valves to relieve stress on the surrounding soil, and then draw water towards its body to create a pocket of fluidized substrate. This locally fluidized zone requires much less force to move through than static soil, resulting in burrowing energy that scales linearly with depth, rather than depth squared. In dry soil, if the valves of a clam-like device are contracted fast enough, the horizontal stress in the soil could be brought to a zero-stress state. This would correspondingly reduce the local vertical stresses to zero, which could dramatically lower the force required to burrow compared to moving through static dry soil. Using analytical models of soil failure mechanics, we investigated the critical timescales for inducing a zero-stress state in soil surrounding an E. directus-like device with contracting valves. This device was modeled as a similar size to a real razor clam (15 mm wide). It was found that for most dry soils, the device would have to contract its valves in 0.02 seconds, a speed within the realm of possibility for a mechanical system. These results suggest that the burrowing method used by E. directus could feasibly be adapted for digging in dry soil.

Resolving Two Dimensional Angular Velocity within a Rotary Tumbler1. NATHANIEL HELMINIAK, DAVID HELMINIAK, VIKRAM CARIAPA, JOHN BORG, Marquette University — In this study, a horizontally oriented cylindrical tumbler, filled at variable depth with cylindrical media, was rotated at various constant speeds. A monoplane layer of media was photographed with a high-speed camera and images were post processed with Particle Tracking Velocimetry (PTV) algorithms in order to resolve both the translational and rotational flow fields. Although the translational velocity fields have been well characterized, contemporary resources enable the ability to expand upon and refine data regarding rotational characteristics of particles within a rotary tumbler. The results indicate that particles rotate according to intermittent no-slip interactions between the particles and solid body rotation. Particles within the bed, not confined to solid body rotation, exhibited behavior indicative of2 gearing between particles; each reacting to the tangential component of contact forming rotation chains. Furthermore, it was observed that solid body interactions corresponded to areas of confined motion, as areas of high interaction dissuaded no-slip rotation, while areas of developing flow tended towards no-slip rotation.

1Special thanks to: NASA Wisconsin Space Grant Consortium Program as well as Marquette University OPUS College of Engineering
8:13AM A7.00002 Modeling and analysis of thermoacoustic instabilities in an annular combustor, SANDEEP MURTHY, TARAHEH SAYADI, VINCENT LE CHENADEC, University of Illinois, Urbana-Champaign, PETER SCHMID, Department of Mathematics, Imperial College London — A simplified model is introduced to study thermo-acoustic instabilities in axisymmetric combustion chambers. Such instabilities can be triggered when correlations between heat-release and pressure oscillations exist, leading to undesirable effects. Gas turbine designs typically consist of a periodic assembly of \( N \) identical units; as evidenced by documented studies, the coupling across sectors may give rise to unstable modes, which are the highlight of this study. In the proposed model, the governing equations are linearized in the acoustic limit, with each burner modeled as a one-dimensional system, featuring acoustic damping and a compact heat source. The coupling between the burners is accounted for by solving the two-dimensional wave equation over an annular region, perpendicular to the burners, representing the chamber’s geometry. The discretization of these equations results in a set of coupled delay-differential equations, that depends on a finite set of parameters. The system’s periodicity is leveraged using a recently developed root-of-unity formalism (Schmid et al, 2015). This results in a linear system, which is then subjected to modal and non-modal analysis to explore the influence of the coupled behavior of the burners on the system’s stability and receptivity.

8:26AM A7.00003 Azimuthally forced flames in an annular combustor, NICHOLAS WORTH, JAMES DAWSON, NTNU, EPAMINONDAS MASTORAKOS, University of Cambridge — Thermoacoustic instabilities are more likely to occur in lean burn combustion systems, making their adoption both difficult and costly. At present, our knowledge of such phenomena is insufficient to produce an inherently stable combustor by design, and therefore an improved understanding of these instabilities has become the focus of a significant research effort. Recent experimental and numerical studies have demonstrated that the symmetry of annular chambers permit a range of self-excited azimuthal modes to be generated in annular geometry, which can make the study of isolated modes difficult. While acoustic forcing is common in single flame experiments, no equivalent for forced azimuthal modes in an annular chamber have been demonstrated. The present investigation focuses on the novel application of acoustic forcing to a laboratory scale annular combustor, in order to generate azimuthal standing wave modes at a prescribed frequency and amplitude. The results focus on the ability of the method to isolate the mode of oscillation using experimental pressure and high speed OH\(^*\) measurements. The successful excitation of azimuthal modes demonstrated represents an important step towards improving our fundamental understanding of this phenomena in practically relevant geometry.

9:05AM A7.00006 ABSTRACT WITHDRAWN

9:18AM A7.00007 Analysis of Premixed Flame Response and Rayleigh Criterion through a Novel Flame Transfer Function, VIJAYA KRISHNA RANI, SARMA RANI, Univ of Alabama - Huntsville — Linear modal analysis of combustion instabilities requires a flame transfer function which describes the flame heat-release response to acoustic perturbations. In this study, a novel flame transfer function (FTF) is developed that provides an explicit relationship between heat-release and pressure fluctuations for laminar premixed flames. While the FTF is generalized for any mean flame shape, a triangular mean flame stabilized at the cross-sectional interface of a dump combustor is analyzed. For this flame, the effects on the FTF magnitude and phase of the acoustic frequency, location (on the mean flame), modal index, and the mean Mach number are investigated. To illustrate and analyze the Rayleigh’s criterion, the spatio-temporal integral of the correlation of pressure and heat-release fluctuations is calculated. It is found that the magnitude of the FTF shows harmonic-like oscillations whose amplitude decreases with frequency, suggesting that the flame shows preferential response to certain frequencies than others. The oscillatory behavior becomes increasingly prominent as one moves away from the flame anchoring point(s). Finally, evaluation of the Rayleigh integral clearly demonstrates the flame-acoustic phase shifts at which combustion instability may arise.

9:31AM A7.00008 Modeling of piezoelectric energy extraction in a thermoacoustic engine with multi-pole time-domain impedance, JEFFREY LIN, Stanford University, CARLO SCALO, Purdue University, LAMBERTUS HESSELINK, Stanford University — We have carried out the first high-fidelity Navier-Stokes simulation of a complete thermoacoustic engine with piezoelectric energy extraction. The standing-wave thermoacoustic piezoelectric (TAP) engine model comprises a 51 cm long cylindrical resonator, containing a thermoacoustic stack on one end and capped by a PZT-5A piezoelectric diaphragm on the other end, tuned to the frequency of the thermoacoustically-amplified mode (388 Hz). A multi-pole broadband time-domain impedance model has been adopted to accurately simulate the measured electromechanical properties of the piezoelectric diaphragm. Simulations are first carried out from quasi-quiescent conditions to a limit cycle, with varying temperature gradients and stack configurations. Stack geometry and boundary layers are fully resolved. Acoustic energy extraction is then activated, achieving a new limit cycle at lower pressure amplitudes. The scaling of the modeled electrical power output and attainable thermal-to-electric energy conversion efficiencies are discussed. Limitations of extending a quasi-one-dimensional linear approximation based on Rott’s theory to a (low amplitude) limit cycle are discussed, as well as nonlinear effects such as thermoacoustic energy transport and viscous dissipation.
9:44AM A7.00009 Suppression of Leidenfrost effect via low frequency vibrations.
BOON THIAM NG, YEW MUN HUNG, MING KWANG TAN. School of Engineering, Monash University, 47500 Bandar Sunway, Malaysia — Leidenfrost effect occurs when vapor layer forms in between the coolant and the hot surface above Leidenfrost point, which dramatically reduces the cooling efficiency due to low thermal conductivity of the vapor layer. To prevent surface overheating, there have been number of reported methods to suppress the Leidenfrost effect that were mainly based on functionalization of the substrate surface and application of electric field across the droplet and substrate. In this work, we induce low frequency vibrations (f ~ 100 Hz) to the heated substrate to suppress the Leidenfrost effect. Three distinct impact dynamics are observed based on different magnitudes of surface acceleration and surface temperature. In gentle film boiling regime, formation of thin spreading lamella around the periphery of the impinged droplet is observed; in film boiling regime, due to thicker vapor cushion, rebound of the impinged droplet is observed; in contact boiling regime, due to the direct contact between the impinged droplet and heated substrate, ejection of the tiny droplet is observed. Also, estimated cooling enhancement ratio for contact boiling regime shows an improvement from 95% to 105%.

Sunday, November 22, 2015 8:00AM - 9:57AM –
Session A8 Magnetohydrodynamics 108 - Oleg Zikanov, University of Michigan - Dearborn

8:00AM A8.00001 Elevator mode convection in liquid metal blankets for fusion reactors1, OLEG ZIKANOV, LI LIU. University of Michigan - Dearborn — The work is motivated by the design of liquid-metal blankets for nuclear fusion reactors. Mixed convection in a downward flow in a vertical duct with strong constant-rate heating of one wall (the Grashof number up to 10^12) and strong magnetic field (the Hartmann number up to 10^3) is considered. It is found that in an infinitely long duct the flow is dominated by exponentially growing elevator modes having the form of a combination of ascending and descending jets. An analytical solution approximating the growth rate of the modes is derived. Analogous flows in finite-length pipes and ducts are analyzed using the high-resolution numerical simulations. The results of the recent experiments are reproduced and explained. It is found that the flow evolves in cycles consisting of periods of exponential growth and breakdowns of the jets. The resulting high-amplitude fluctuations of temperature is a feature potentially dangerous for operation of a reactor blanket.

1Financial support was provided by the US NSF (Grant CBET 1232851)

8:13AM A8.00002 Thermal convection in a horizontal duct with strong axial magnetic field1, XUAN ZHANG, OLEG ZIKANOV. University of Michigan - Dearborn — The work is motivated by design of liquid metal blankets of nuclear fusion reactors. The effect of convection on the flow within a toroidally oriented duct is analyzed. Non-uniform strong heating arising from capture of high-speed neutrons is imposed internally, while the walls are assumed to be isothermal. Very strong heating (the Grashof number up to 10^12) and strong magnetic field (the Hartmann number up to 10^3) corresponding to the realistic fusion reactor conditions are considered. Stability of two-dimensional flow states is analyzed using numerical simulations. The unstable modes at high Hartmann and Grashof number are found to have large wavelengths. The integral properties of developed three-dimensional flows are close to those of two-dimensional flows at the typical parameters of a fusion reactor. We also consider the effect of the weak transverse component of the magnetic field on the flow.

1Financial support was provided by the US NSF (Grant CBET 1232851).

8:26AM A8.00003 Instability of secondary flows in an electromagnetically forced curved duct, JEAN BOISSON, ROMAIN MONCHAUX, IMSIA, UME, SEBASTIEN AUMATRE, CEA/DSM/SPEC/SPHYNX — In this presentation, we investigated the secondary flow forced electromagnetically between two fixed copper cylinders. The gap geometry corresponds to a rectangular curved duct with a large aspect ratio. We have performed low Hartmann number runs (M < 500) and we used ultrasonic probes to access the azimuthal and axial velocity profiles. We have characterized a transition between magnetohydrodynamic and hydrodynamic regimes. These regimes are controlled by the Elsasser number. The hydrodynamic regime corresponds to a classic flow in rectangular curved duct presenting a single pair of stable contra-rotative vortices (2-cell mode) and shows a transition to double pair of vortices (4-cell mode) for large enough magnetic Dean number Κ_M (Dean number adjusted by the magnetic field). We have explored the experimental stability diagram for 100 < Κ_M < 600 and we found a 2-cell/4-cell transition involving the coexistence of the two modes. Therefore we suggest that this MHD device is a good alternative to study flows in curved duct compare to helicoidal or U-turn devices. First, the range of accessible geometry is large. Then as this system is closed, the control parameter can be globally controlled allowing us to observe the flow for at a given forcing on long time scales.

8:39AM A8.00004 Longtime persistence of linear dynamic in magnetococonvective1, SEBASTIEN RENAUDIERE DE VAUX, RÉMI ZAMANSKY, WLADIMIR BERGEZ, PHILIPPE TORDJEMAN. Institut de Mécanique des Fluides de Toulouse — We extend the Chandrasekhar’s marginal stability of an infinite conducting fluid layer heated from below and subjected to a vertical magnetic field to non-zero growth rate. We show that this regime is relevant by comparing the linear stability results with direct numerical simulations (DNS). The growth-rate and wavelengths are accurately predicted by the linear eigenvalue problem, even at largely overcritical Rayleigh numbers. Moreover, it is found that the linear dynamics break down when the amplitude of the velocity perturbations is of the order of the characteristic buoyancy velocity \( W_{buoy} = \sqrt{g\beta \Delta T h} \), with \( g \) the gravitational acceleration, \( \beta \) the thermal expansion coefficient, \( \Delta T > 0 \) the bottom-top temperature difference and \( h \) the height of the layer. At large timescales, there is no memory phenomenon from the onset of convection: the height of the enclosure is only responsible for the size of the structures.

1We acknowledge support from the french Commissariat a l’Energie Atomique from Cadarache
8:52AM A8.00005 Non-linear interactions of magneto-Poincare and magnetostrophic waves in rotating shallow water magnetohydrodynamic. ARAKEL PETROSYAN, DMITRY KLIMACHKOV, Space Research Institute of the Russian Academy of Sciences — We have investigated the interaction of wave packets in the magnetohydrodynamic shallow water flows in external vertical magnetic field. Using the asymptotic multiscale methods we received that three magneto-Poincare waves interact, three magnetostrophic waves also interact, We obtained intermode interactions: two magnetostrophic waves and magneto-Poincare wave, two magneto-Poincare waves and magneto-strophic wave. In all cases we derived nonlinear equations of three waves interactions and showed the existence of two types of instability mechanisms: decay instabilities and parametric growth. It has been found that there are four types of decay instabilities: magneto-Poincare wave decays into two magneto-Poincare waves, magnetostrophic wave decays into two magnetostrophic waves, magneto-Poincare decays into one magnetostrophic and one magneto-Poincare wave, and magnetostrophic wave decays into one magneto-Poincare and one magnetostrophic wave. The growth rates of decay instabilities were received. Also four types of parametric growth were investigated: magneto-Poincare waves amplification, magnetostrophic waves amplification, magneto-Poincare wave growth in field of magnetostrophic wave, and magnetostrophic wave growth in field of magneto-Poincare wave.

9:05AM A8.00006 Stochastic Non-Resistive Magnetohydrodynamic System with Lévy Noise1, MANIL T. MOHAN, SIVAGURU S. SRITHARAN, Air Force Inst of Tech - WPAFB, UTPAL MANNA, Indian Institute of Science Education and Reserach, Thiruvananthapuram — The incompressible, viscous and resistive magnetohydrodynamic (MHD) system consists of the Navier-Stokes equations coupled with the Maxwell equations. MHD has many applications in various fields ranging from Astrophysics to nuclear fusion devices. When the magnetic diffusivity is taken to be zero, we get the incompressible, viscous and non-resistive MHD equations. In this work, we consider the incompressible, viscous and non-resistive MHD equations with Lévy noise in two and three dimensions. We prove the local in time existence and uniqueness (path wise) strong solution to the stochastic non-resistive MHD system up to a maximal stopping time. For proving this, we first consider a class of bounded solutions with finite higher order energy in space variable and prove the solutions of the smoothed version of the stochastic MHD system exist. We find a collection of positive stopping times on which the norms of the smoothed version solution are uniformly bounded. For any stopping time from this collection, the smoothed version solution is a Cauchy sequence and hence is convergent. An application of Sobolev interpolation results and Banach-Alaoglu theorem yield the existence of local in time strong solution. We finally show that this local strong solution is path wise unique.

9:18AM A8.00007 Irregular magnetohydrodynamic shock refraction in the presence of a normal magnetic field1, VINCENT WHEATLEY, The University of Queensland, PAVAMAN BILGI, California Institute of Technology, RAVI SAMTANEY, King Abdullah University of Science and Technology, DALE PULLIN, California Institute of Technology — Shock refraction occurs when an incident shock encounters a density interface, which is important in a number of applications. When all waves resulting from the interaction meet at a point, this is termed regular shock refraction. In magnetohydrodynamics, analytical solutions for regular refraction cases show that magnetohydrodynamic waves transport vorticity from the shocked density interface so that it is not a shear layer. This is the mechanism that underpins the suppression of shock driven instabilities in the presence of a magnetic field. Here, we examine the case of irregular shock refraction where the initial magnetic field is normal to the incident shock. Regular analytical solutions are used to map the boundary of the irregular refraction region in parameter space. Beyond this boundary, the structure of irregular solutions is investigated via numerical simulations. Particular attention is given to whether all fluid interface emanating from wave intersection points are free of vorticity.

9:31AM A8.00008 Vorticity transport in shock driven plasma flows: A comparison of MHD and two-fluid models1, DARYL BOND, VINCENT WHEATLEY, The University of Queensland, DALE PULLIN, The California Institute of Technology, RAVI SAMTANEY, King Abdullah University of Science and Technology — Suppression of the Richtmyer-Meshkov instability in a plasma, through the application of a seed magnetic field, has been studied in the framework of ideal magnetohydrodynamics. These studies have shown that suppression is achieved through the transport of vorticity by magnetohydrodynamic waves away from a perturbed fluid-fluid interface where it was baroclinically generated by shock impact. The implementation of a more physically accurate, fully electromagnetic, two-fluid plasma representation allows a more realistic investigation of vorticity transport in shock driven plasma flows. Results comparing ideal one-dimensional two-fluid and magnetohydrodynamic flows are presented. Substantial increases in the complexity of the flow field and vorticity transport dynamics are observed with important ramifications for the stabilization of shock driven interfaces.

9:44AM A8.00009 Turbulence in the Heliosheath: spectral analysis from Voyager 1 and 2 data. FEDERICO FRATERNALE, LUCA GALLANA, MICHELE IOVIENO, Politecnico di Torino, Dimeas, SOPHIE FOSSON, None, ENRICO MAGLI, Politecnico di Torino, det. MERAV OPHER, None, JOHN RICHARDSON, MIT, Kavli Institute, RACHEL MORGAN, MIT, Aeronautics and Aerospace, DANIELA TORDELLA, Politecnico di Torino, Dimeas, SOPHIE FOSSON, None, ENRICO MAGLI, Politecnico di Torino, det. MERAV OPHER, None, JOHN RICHARDSON, MIT, Kavli Institute, RACHEL MORGAN, MIT, Aeronautics and Aerospace — The Voyager 2 spacecraft is traveling through the heliosheath, the outermost layer in heliosphere where the solar wind is slowed by the interstellar gas, while Voyager 1 has entered the local interstellar medium. They are providing the fist in-situ measurement of plasma and magnetic fields in that regions. We focus on the differences between the energetic particle intensity variations seen by the Voyager 1 and 2 crafts that are crossing the sectored and the unipolar as well as the sectored heliosheath regions, respectively. We try to provide a spectral analysis of the full heliosheath, characterizing the plasma and magnetic field turbulence through the estimate of the spectral properties in the different frequency ranges. Signal reconstruction techniques are mandatory to reconstruct spectra due to extreme data sparsity (up to 97% missings in high resolution data beyond 80 AU). We use three different methods: correlation computation coupled with the maximum likelihood reconstruction, compress sensing and a genetic algorithm to estimate the gap influence on reconstructed spectra. These methods have been previously validated on 1979 data and synthetic hydrodynamics fluid turbulent fields. Results on power density, energy and helicity spectra will

Sunday, November 22, 2015 8:00AM - 9:57AM – Session A9 Flow Instability: Wakes 109 - Patricia Ern, IMFT
8:00AM A9.00001 Vortex pairing in the wake of an oscillating bubble rising in a thin-gap cell. PATRICIA ERN, AUDREY FILELLA, VERONIQUE ROIG, IMFT, Toulouse University and CNRS, France — We investigate experimentally the oscillatory motion and wake of a bubble rising in a counter flow in a thin gap cell (3 mm) by shadowgraphy and PIV. The equivalent diameter \( d \) of the bubble in the plane of the cell is used to define the Archimedes number \( Ar = \frac{\sqrt{\pi}}{g} (\nu \text{ is the kinematic viscosity and } g \text{ the gravitational acceleration}). \) The counter flow is characterized by the Reynolds number \( Re_{cf} \) based on the mean liquid velocity and the gap thickness. For \( 500 \leq Ar \leq 5500 \) and \( 0 \leq Re_{cf} \leq 200 \), the mean vertical velocity of the bubble relative to the counter flow, \( V_{br} \), corresponds to the mean rising velocity in liquid at rest; and the frequency and the amplitude of the oscillatory motion superpose for all \( Re_{cf} \) when normalized with \( V_{br} \) and the timescale \( d/V_{br} \). For a given size of the bubble \( (d \approx 9.5 \text{ mm and } Ar \approx 2900) \) corresponding to a Reynolds number based on \( V_{br} \) and \( d \) of about 1900, we then investigate in detail the wake associated to the bubble in several counter flows. As \( Re_{cf} \) increases, the number of vortices released increases. Furthermore, the wake of the bubble undergoes vortex pairing for \( 0 \leq Re_{cf} \leq 110 \), whereas no vortex pairing is observed for \( Re_{cf} \geq 140 \).

8:13AM A9.00002 Experimental investigation of freely falling thin disks: Transition of three-dimensional motion from zigzag to spiral. ZHUANG SU, CUNBIAO LEE, Peking Univ — The motion of a freely falling thin disk was investigated experimentally. Different motion modes, including planar zigzag and three dimensional spiral, were identified based on the measurements of the whole six degrees of freedom of the disk. The final motion modes in the fall were found to change with the dimensionless moments of inertial \( (I^*) \), which is determined by the aspect ratio of the disk and the density ratio between the disk and water. The motion mode is zigzag in the range of \( 2.95 \times 10^{-2} \) to \( 1.17 \times 10^{-2} \) and spiral in the range of \( 7.36 \times 10^{-3} \) to \( 1.47 \times 10^{-3} \) in our experiments. A zigzag to spiral transition process was found in the lower \( I^* \) range. Two different types of transition were identified, which are zigzag-spiral monotonous transition in the lower and higher Reynolds number range \( (600 \text{ to } 1000 \text{ and } \geq 2900 \text{ in our experiments}) \) and zigzag-spiral-zigzag intermittence transition in the middle range. The forces acted on the disk were also investigated. Different force behaviors corresponding to different types of wake structures were identified and analyzed.

8:26AM A9.00003 Experimental investigation of the elastic flag spontaneous flapping in water flow. YONGXIA JIA, LICHAO JIA, ZHUANG SU, YIDING ZHU, HUJING YUAN, CUNBIAO LEE, Peking Univ — The flapping stability and the response of a thin two-dimensional flag of low bending rigidity to the Reynolds number was investigated. The three relevant non-dimensional parameters governing fluid-structure problems that concern the interaction of elastic flags with high-speed fluid flows are the structure-to-fluid mass ratio, the non-dimensional bending rigidity and the Reynolds number. To study the mechanisms of the transition from the periodic flapping to chaotic flapping, we use PIV and flow visualization techniques to obtain the whole flow field around the midspan of the immersed elastic flag interacting with fluid in both periodic and chaotic states. A moving interface detection technique is used to determine the flag position and velocity. Virtual particle images are imposed in the flag region in the PIV algorithm, of which the displacements are evaluated by the flag movement. We find that the value of \( St \) is constrained in the narrow range of \( 0.2 < St < 0.31 \) based on the flapping amplitude. We find that the transition to chaos occurs at a critical Reynolds number \( Re = 60800 \). For the larger Reynolds number, the high-strength vortices are distributed in a detached region away from the free end of the flag during the intermittent snapping events in the chaotic regime.

8:39AM A9.00004 The effect of Reynolds number on the drag of a rectangular cylinder. ROBERT BREIDENTHAL, JONATHAN WAI, University of Washington — Direct numerical simulations of the flow past a rectangular cylinder at low Reynolds number reveal that the aspect ratio for maximum drag is much less than that measured at high Reynolds number. Nakaguchi et al. (1967) discovered a remarkably sharp peak in the drag coefficient at a cylinder aspect ratio of 0.62 for \( Re = 20000 \). In contrast, our numerical simulations at \( Re = 500 \) indicate a maximum-drag aspect ratio of 0.2. This dramatic difference is attributed to the rollup station of the laminar vortex sheet from the separating boundary layer. Essentially inviscid, the rollup process scales with the thickness of the vortex sheet at the separation point, which in turn varies inversely with the square root of the Reynolds number. Consequently, at low Reynolds number, the sheet remains thin and laminar, curving tightly toward the cylinder. On the other hand, at high Reynolds number, the vortex sheet promptly rolls up into a rapidly growing, turbulent shear layer. The thick, turbulent layer has a large displacement thickness, deflecting the outer streamlines and altering its own trajectory so that it curves relatively gradually toward the cylinder. Bearman and Trueman (1972) showed that the peak drag corresponds to the shear layer nearly reattaching to the bluff body and rolling up into vortices very close to the base of the cylinder. The low pressure of the vortex cores is reflected in a low base pressure and thus high drag. The critical aspect ratio is much smaller for the laminar vortex sheet because of its more tightly curved trajectory.

8:52AM A9.00005 ABSTRACT WITHDRAWN —

9:05AM A9.00006 ABSTRACT WITHDRAWN —

9:18AM A9.00007 Stability Analysis of Flow Past a Wingtip. ADAM EDSTRAND, Florida State University, PETER SCHMID, Imperial College in London, KUNIHiko TAIRA, LOUIS CATTAFIESTA, Florida State University — Trailing vortices are commonly associated with diminished aircraft performance by increasing induced drag and producing a wake hazard on following aircraft. Previous stability analyses have been performed on the Batchelor vortex (Heaton et al., 2009), which models a far field axisymmetric vortex, and airfoil wakes (Woodley & Peake, 1997). Both analyses have shown various instabilities present in these far field vortex-wake flows. This complicates the design of control devices by excluding consideration of near field interactions between the wake and vortex shed from the wing. In this study, we perform temporal and spatial bi-global stability analyses on the near field wake of the flow field behind a NACA0012 wing computed from direct numerical simulation at a chord Reynolds number of 1000. The results identify multiple instabilities including a vortex instability, wake instability, and mixed instability that includes interaction between the wake and vortex. As these modes exhibit wave packets, we perform a wave packet analysis (Obrist & Schmid, 2010), which enables the prediction of spatial mode structures at low computational cost. Furthermore, a bi-global parabolized stability analysis is performed, highlighting disparities between the parallel and parabolized analysis.

1ONR Grant N00014010824 and NSF PIRE Grant OISE-0968313
9:31AM A9.00008 Bifurcations beneath the bluff body instability modes. AMALENDU SAU, Gyeongsang National University — A new family of Hopf bifurcations is detected in a cylinder wake. Besides widely known streamwise bifurcations, our study reveals a new route to wake transition via cross-stream flow undulations and a class of previously unknown spanwise bifurcations along von Karman vortex cores; leading to an improved understanding of wake transformation and the transitional flow physics. It shows, alternate vortex shedding generates significant cross-stream momentum transfer, which facilitates self-sustained spanwise wake oscillation and growth of sequence of bifurcations along Karman corelines. The study shows how spanwise oscillation of pressure/velocity/KE coupling growing with Re and influence onset of “Mode A” and “Mode B” instabilities. It reports two distinct stages of wake undulation for 125≤Re≤240. While weakly subcritical periodic-oscillation of pressure/velocity/vorticity along Karman corelines and the uniform/wider length-scale bifurcations dominate during “Mode A” instability, the transition to “Mode B” is prompted following eruption/swapping of significantly smaller variable and growth of sequence of bifurcations along Karman corelines. The study shows how spanwise oscillation of pressure/velocity/KE coupling growing with Re and influence onset of “Mode A” and “Mode B” instabilities. It reports two distinct stages of wake undulation for 125≤Re≤240. While weakly subcritical periodic-oscillation of pressure/velocity/vorticity along Karman corelines and the uniform/wider length-scale bifurcations dominate during “Mode A” instability, the transition to “Mode B” is prompted following eruption/swapping of significantly smaller variable and growth of sequence of bifurcations along Karman corelines. The study shows how spanwise oscillation of pressure/velocity/KE coupling growing with Re and influence onset of “Mode A” and “Mode B” instabilities. It reports two distinct stages of wake undulation for 125≤Re≤240.

9:44AM A9.00009 Global stability analysis of turbulent 3D wakes. GEORGIOS RIGAS, University of Cambridge, DENIS SIPP, ONERA LA FAE DAEF, MATTHEW JUNIPER, University of Cambridge — At low Reynolds numbers, corresponding to laminar and transitional regimes, hydrodynamic stability theory has aided the understanding of the dynamics of bluff body wake-flow and the application of effective control strategies. However, flows of fundamental importance to many industries, in particular the transport industry, involve high Reynolds numbers and turbulent wakes. Despite their turbulence, such wake flows exhibit organisation which is manifested as coherent structures. Recent work has shown that the turbulent coherent structures retain the shape of the symmetry-breaking laminar instabilities and only those manifest as large-scale structures in the near wake (Rigas et al., JFM vol. 750:R5 2014, JFM vol. 778:R2 2015). Based on the findings of the persistence of the laminar instabilities at high Reynolds numbers, we investigate the global stability characteristics of a turbulent wake generated behind a bluff three-dimensional axisymmetric body. We perform a linear global stability analysis on the experimentally obtained mean flow and we recover the dynamic characteristics and spatial structure of the coherent structures, which are linked to the transitional instabilities. A detailed comparison of the predictions with the experimental measurements will be provided.

Sunday, November 22, 2015 8:00AM - 9:57AM – Session A10 Microscale Flows: Electrokinesis

8:00AM A10.00001 Energy Conversion over Super-hydrophobic Surfaces. HUI ZHAO, SHENGJIE ZHAI, University of Nevada Las Vegas — The streaming potential generated by a pressure-driven flow over a charged slip-stick surface with an arbitrary double layer thickness is both theoretically and experimentally studied. To understand the impact of the slip, the streaming potential is compared against that over a homogeneously charged smooth surface. Our results indicate that the streaming potential over a super-hydrophobic surface only can be enhanced under certain conditions. In addition, the Onsager relation which directly relates the magnitude of electro-osmotic effect to that of the streaming current effect has been explicitly proved to be valid for thin and thick double layers and homogeneously charged super-hydrophobic surfaces. Comparisons between the streaming current and electro-osmotic mobility for an arbitrary electric double layer thickness under various conditions indicate that the Onsager relation seems applicable for arbitrary weakly charged super-hydrophobic surfaces though there is no general proof. Knowledge of the streaming potential over a slip-stick surface can provide guidance for designing novel and efficient microfluidic energy-conversion devices using super-hydrophobic surfaces.

8:13AM A10.00002 Numerical Optimization Strategy for Determining 3D Flow Fields in Microfluidics. ALEX EDEN, MARIN SIGURDSON, IGOR MEZIC, CARL MEINHART, Univ of California - Santa Barbara — We present a hybrid experimental-numerical method for generating 3D flow fields from 2D PIV experimental data. An optimization algorithm is applied to a theory-based simulation of an alternating current electrothermal (ACET) microchannel in conjunction with 2D PIV data to generate an improved representation of 3D steady state flow conditions. These results can be used to investigate mixing phenomena. Experimental conditions were simulated using COMSOL Multiphysics to solve the temperature and velocity fields, as well as the quasi-static electric fields. The governing equations were based on a theoretical model for ac electrothermal flows. A Nelder-Mead optimization algorithm was used to achieve a better fit by minimizing the error between 2D PIV experimental velocity data and numerical simulation results at the measurement plane. By applying this hybrid method, the normalized RMS velocity error between the simulation and experimental results was reduced by more than an order of magnitude. The optimization algorithm altered 3D fluid circulation patterns considerably, providing a more accurate representation of the 3D experimental flow field. This method can be generalized to a wide variety of flow problems.

8:26AM A10.00003 Electroconvection near the interface of ion-selective membranes and a microchannel. KAREN WANG, ALI MANI, Stanford Univ — The transport dynamics of electroconvective flows near ion-selective membranes subject to charged sidewalls is simulated using a direct numerical simulation of the coupled Poisson-Nernst-Planck and Navier-Stokes equations. Previous studies have investigated electroconvective instability near infinitely large flat membranes and have demonstrated their role in significant enhancement of transport via advection effects. This study demonstrates how the presence of sidewalls from a connecting microchannel can affect the onset of electroconvective flows and also impact the net ion transport rate. We demonstrate that sidewalls without charge stabilize the flow and delay the onset of electroconvection while walls with properly signed charges can induce flow and lead to enhancement of transport. Impact of the sidewalls in energy and throughput efficiency will also be discussed.

8:39AM A10.00004 Simultaneous Aggregation and Height Bifurcation of Colloidal Particles near Electrodes in Oscillatory Electric Fields. SCOTT BUKOSKY, WILLIAM RISTENPART, Dept. Chemical Engineering, University of California Davis — The behavior of micron scale colloidal particles near electrodes in oscillatory electric fields is known to be sensitive to the identity of the surrounding electrolyte. For example, micron-scale particles suspended in 1 mM NaCl aggregate laterally near the electrode upon application of a low-frequency (~ 100 Hz) field, but the same particles suspended in NaOH are instead observed to laterally separate. Recent work has revealed that, contrary to previous reports, particles suspended in NaOH indeed aggregate under some conditions while simultaneously exhibiting a distinct bifurcation in average height above the electrode. Here, we elaborate on this observation by demonstrating the existence of a critical frequency (~ 25 Hz) below which particles in NaOH aggregate laterally and above which the same particles separate. At sufficiently low frequencies, particles still exhibit a distinct height bifurcation, but those particles immediately adjacent to the electrode surface also move laterally to form aggregates. These results indicate that the current decoupling of electrolytes as either aggregating or separating is misleading, and that the key role of the electrolyte instead is to set the magnitude of a critical frequency at which particles transition between behaviors.
channels and Turns, which are basic building blocks in microfluidic devices. Furthermore, we study the inverse problem in which we define the desired flow pattern and solve for the zeta potential distribution required in even in the immediate vicinity of the disk. We then illustrate the ability to generate complex flow fields using superposition of such disks. We obtain a solution for the case of a disk with uniform zeta potential and show that the flow field created is an exact dipole, the inhomogeneous parts in these equations are governed by gradients in zeta potential parallel and perpendicular to the applied electric field, respectivley. We present an analytical study, validated by numerical simulations, of electroosmotic flow in a Hele-Shaw configuration with non-uniform zeta potential distribution. Applying the lubrication approximation and assuming thin electric double layer, we derive a pair of uncoupled Poisson equations for the pressure and the stream function, and show that the inhomogeneous parts in these equations are governed by gradients in zeta potential parallel and perpendicular to the applied electric field, respectively. We obtain a solution for the case of a disk with uniform zeta potential and show that the flow field created is an exact dipole, even in the immediate vicinity of the disk. We then illustrate the ability to generate complex flow fields using superposition of such disks. Furthermore, we study the inverse problem in which we define the desired flow pattern and solve for the zeta potential distribution required in order to establish it. We demonstrate that such inverse problem solutions can be used to create directional flows confined within narrow regions, without physical walls. We show that these solutions can be assembled to create complex microfluidic networks, composed of intersecting channels and turns, which are basic building blocks in microfluidic devices.

Continuous-flow Electrophoretic Separation of Particles with Dissimilar Charge-to-Mass Ratios via the Wall-induced Non-inertial Lift, Cory Thomas, Andrew Todd, Xinyu Lu, Xiangchun Xuan, Clemson University — Traditional electrophoresis separates particles with dissimilar charge-to-mass ratios along the channel length direction in a batchwise mode. We present in this talk a continuous-flow electrophoretic separation of particles in the transverse direction of a straight microchannel. This separation stems from the particle property-dependent lateral migration due to the wall-induced non-inertial electrical lift force. It is demonstrated through both a binary and a ternary separation of polymer particles based on surface charge and size. A numerical model is also developed to understand this separation and to study the parametric effects.

Electrothermal Flow Enhanced Sample mixing in a Ratchet Microchannel, Christian Brumme, Ryan Shaw, Yilong Zhou, Xinyu Lu, Xiangchun Xuan, Clemson University — We present in this talk an electrokinetic method for sample mixing in a ratchet microchannel. Due to Joule heating effects in the background electrolyte, temperature gradients are created around the ratchets causing non-uniform fluid properties. The action of electric field on these thermally induced property gradients yields an electric force that can manifest itself in the flow field in the form of circulations. We demonstrate the use of electrothermal flow circulations to enhance sample mixing through both experiment and numerical modeling.

Control of colloid transport via solute gradients in dead-end channels, Sangwoo Shin, Eujin Um, Princeton University, Patrick Warren, Unilever R&D Port Sunlight, Howard Stone, Princeton University — Transport of colloids in dead-end channels is involved in widespread applications ranging from drug delivery to geophysical flows. In such geometries, Brownian motion may be considered as the sole mechanism that enables transport of colloidal particles into or out of the channels, which is, unfortunately, an extremely inefficient transport mechanism for microscale particles. Here, we explore the possibility of diffusiophoresis as a means to control the colloid transport by introducing a solute gradient along the dead-end channels. We demonstrate that the transport of colloidal particles into the dead-end channels can be either enhanced or completely prevented via diffusiophoresis. We also observe a size-dependent focusing of the particles where, as the particle size increases, the particles tend to concentrate more, and they tend to reside deeper in the channel. Our findings have implications for all manners of controlled release processes, especially for site-specific drug delivery systems where localized targeting of drugs with minimal dispersion to the non-target is essential.

The Formation of Ion Concentration Polarization Layer Induced by Bifurcated Current Path, Junsuk Kim, Hyomin Lee, Inhee Cho, Snu, Ece, Hoyoung Kim, Snu, Mae, Sungen Jae Kim, Snu, Ece, Ees Team, Mfm Team — Ion Concentration Polarization (ICP) is a fundamental electrokinetic phenomenon that occurs near a porous membrane boundary layer. In this work, a new ICP device that bifurcated the current path was fabricated using micro/micro-nano/nano/micro hybrid channel connection, while a conventional ICP device has employed micro/micro/nano/micro channel connection. The propagation of ICP layer was initiated from the nano-channel at high concentration regime and from micro-nano/nano connection at low concentration regime. Interestingly, the reverse propagation was observed at low concentration regime as well. These combined effects conveyed a competition between two distinguishable propagations at intermediate concentration regime, caused by singularity of the bifurcated current path. Experiments and an equivalent circuit analysis were conducted for this bifurcation. As a result, the conductance ratio of electrolyte to Nafion governed the bifurcation. Conclusively, the bifurcation-induced ICP layer formation was able to be characterized by analyzing current-time characteristic which have two distinct RC delay times.

Plumes from vertical buoyancy sources detrain - where does it matter?, Rachaell Bonnebaigt, DAMTP, University of Cambridge, C. P. Caulfield, BP Institute and DAMTP, University of Cambridge, Paul Linden, DAMTP, University of Cambridge — Buildings often have heated vertical surfaces, such as patches of wall heated by sunlight. How do these heated surfaces affect the temperature stratification in a room? Using analogue laboratory experiments and a theoretical model, we investigate the stratification in a sealed, insulated space containing a vertically distributed buoyancy source. In the experiments, the source drives a turbulent plume, as in the theoretical model. However, the plume then detains (enters into the ambient) at intermediate heights. This detrainment is not accounted for in current theoretical models. We compare theoretical and experimental ambient density profiles to see whether detrainment is significant for various boundary conditions on the heated wall.

Sunday, November 22, 2015 8:00AM - 9:57AM – Session A11 Convection and Buoyancy-Driven Flows: Plumes

8:00AM A11.00001 Plumes from vertical buoyancy sources detrain - where does it matter?, Rachaell Bonnebaigt, DAMTP, University of Cambridge, C. P. Caulfield, BP Institute and DAMTP, University of Cambridge, Paul Linden, DAMTP, University of Cambridge — Buildings often have heated vertical surfaces, such as patches of wall heated by sunlight. How do these heated surfaces affect the temperature stratification in a room? Using analogue laboratory experiments and a theoretical model, we investigate the stratification in a sealed, insulated space containing a vertically distributed buoyancy source. In the experiments, the source drives a turbulent plume, as in the theoretical model. However, the plume then detains (enters into the ambient) at intermediate heights. This detrainment is not accounted for in current theoretical models. We compare theoretical and experimental ambient density profiles to see whether detrainment is significant for various boundary conditions on the heated wall.
The plume’s trajectory is flatter with the plume angle scaling on $s$. This result is particularly crucial for modeling wildfire spread and also taken into account. The physical mechanisms leading to formation of this triangular shape and the entrainment properties of such a structure will be presented.

1Work supported by the CNES-France

8:26AM A11.00003 Critical modes due to Archimedean buoyancy and dielectrophoretic force in a dielectric liquid in cylindrical annulus\(^1\), ANTOINE MEYER, LOMC, UMR6294, CNRS-Université du Havre, HARUNORI YOSHIKAWA, LMJAD, UMR 7351, CNRS- Université de Nice Sophia Antipolis., OLIVIER CRUMÉY-ROLLE, INNOCENT MUTABAZI, LOMC, UM 6294, CNRS-Université du Havre — An incompressible dielectric fluid is confined in a cylindrical annulus maintained at two different temperatures and an electric tension in Earth gravity. The coupling between the electric field and the thermal variation of the permittivity leads to a dielectrophoretic force that acts as a buoyancy force to induce convective flows. We have performed the linear stability analysis to determine the critical parameters and the nature of critical modes for different values of the control parameters. Four types of modes were found. For weak values of the electric tension, the critical modes are either hydrodynamic or thermal modes depending on the Prandtl number and for large values of electric tension lead to electric modes [1]. For its intermediate values, critical modes are columnar vortices, similar to those observed in simulations of the convection in a cylindrical annulus with a radial gravity [2].


8:39AM A11.00004 Bifurcation and Stability Analyses in Horizontal Convection, PIERRE-YVES PASSAGIA, ALBERTO SCOTTI, BRIAN WHITE, Department of Marine Sciences, University of North Carolina, Chapel Hill, NC 27599, USA — Horizontal Convection is a flow driven by differential buoyancy forcing across a horizontal surface. It has been considered as a simple model to study the influence of heating and cooling at the ocean surface on the Meridional Overturning Circulation. The mechanisms responsible for transition to turbulence are presented using a bifurcation and global stability analyses of the two-dimensional baseflows. The forcing imposed at the surface creates a circulation characterized by a sinking plume near the pole and an upwelling at the equator. Increasing the magnitude of the forcing, the steady states are shown to undergo a sub-critical bifurcation, leading to a transition in the behaviour of the descending plume. These steady states are shown to become unstable to both two and three-dimensional perturbations. The three-dimensional instability modes are characterized by counter-rotating vortices located in the plume and are associated with the Rayleigh-Taylor instability. The two-dimensional instability modes are associated with the vortex shedding of the plume, spreading into the abyss. Using the available potential energy analysis framework, we identify the instability mode responsible for the best mixing efficiency.

8:52AM A11.00005 Sedimentation from Particle-Laden Plumes in Stratified Fluids, BRUCE SUTHERLAND, University of Alberta, YOUN SUB HONG, University of Toronto — Laboratory experiments are performed in which a mixture of particles, water and a small amount of dye is continuously injected upwards from a localized source into a uniformly stratified ambient. The particle-fluid mixture initially rises as a forced plume (which in most cases is buoyant, though in some cases due to high particle concentration is negative-buoyant at the source), reaches a maximum height, collapses upon itself and then spreads as a radial intrusion. The particles are observed to rain out of the descending intrusion and settle upon the floor of the tank. Using light attenuation, the depth of the particle mound is measured after the experiment has run for a fixed amount of time. In most experiments the distribution of particles is found to be approximately axisymmetric about the source with a near Gaussian structure for height as a function of radius. The results are compared with a code that combines classical plume theory with an adaptation to stratified fluids of the theory of Carey, Sigurdsson and Sparks (JGR, 1988) for the spread and fall of particles from a particle-laden plume impacting a rigid ceiling. Re-entrainment of particles into the plume is also taken into account.

9:05AM A11.00006 Computational study of the formation and evolution of a three-dimensional gravity current, ANDREW OOI, SHUANG ZHU, The University of Melbourne, NADIM ZGHEIB, BALACHAN-DAR SIVARAMAKRISHNAN, University of Florida — Gravity currents occur when fluids of different density are brought together. They are relevant in many engineering applications such as the dispersion of hazardous gas cloud or the spillage heavy chemicals from marine vehicles. Thus far, most of the studies have assumed that the gravity current is two-dimensional (or “planar”) as it travels down the slope, i.e. the gravity current is homogeneous in the spanwise direction. In this study, we utilise data from direct numerical simulation to investigate the formation and evolution of a fully three-dimensional gravity current propagating down a uniform slope. Previous theoretical studies have predicted that three-dimensional gravity current will evolve towards a “self-similar” circular wedge shape. Flow visualization from experiments showed that, contrary to the theoretical prediction, the gravity current takes on a shape that is more akin to a triangular wedge. Data from our direct numerical simulation agrees with the experimental observation. Furthermore, it has been found that the shape of this triangular wedge is relatively insensitive to the initial shape of the gravity current. The physical mechanisms leading to formation of this triangular shape and the entrainment properties of such a structure will be presented.

9:18AM A11.00007 Trajectory of a plume in a power-law velocity profile, ALI TOHIDI, NIGEL KAYE, Clemson University — Highly buoyant plumes, bent-over by a cross flow, occur in many situations ranging from waste-water discharges into rivers up to wildfire plumes in the atmosphere. Highly buoyant plumes have a steeper initial trajectory and, therefore, rise to regions of higher velocity. Hence, their trajectory will be more greatly affected by vertical variations in horizontal velocity. It is shown that, for a power-law boundary layer, the volume and momentum fluxes scale on the square of the plume’s path ($s^2$) compared to $s^{4/3}$ for a uniform velocity. The plume’s trajectory is flattered with the plume angle scaling on $s^{-1}$ compared to $s^{-1/3}$ in the uniform case. However, experiments two regimes in the literature indicates that, under certain conditions, the boundary layer velocity profile makes little difference to the plume trajectory and algebraic equations developed for plumes in a uniform cross flow are adequate. Source length scale analysis is used to establish criteria for when to include boundary layer velocity variations. Such variations are only important when either the momentum length scale or buoyancy length scale is considerably greater than the release height of the plume. This result is particularly crucial for modeling wildfire plumes.
location of gas concentration in
leak source, also exhibits interesting plume behavior. In the simulated field tests, compressed gas tanks are used to mimic leaks and generate
diagnostics, including thermodynamic data and velocity from hot-wire and multi-hole probes. To characterize the system behavior and verify its
effectiveness, field tests have been conducted over controlled rangeland burns and over simulated leaks. In the former case, since fire produces
carbon dioxide over a large area, this was an opportunity to test in an environment that while only vaguely similar to a carbon sequestration
leak source, also exhibits interesting plume behavior. In the simulated field tests, compressed gas tanks are used to mimic leaks and generate
gaseous plumes. Since the sensor response time is a function of vehicle airspeed, dynamic calibration models are required to determine accurate
location of gas concentration in \((x,y,z,t)\). Results are compared with simulations using combined flight and atmospheric dynamic models.

1Supported by Department of Energy Award DE-FE00012173
8:52AM A12.00005 Compressible Viscoelastic Flows Generated by Vibrating Nanoscale Structures in Simple Liquids. JOHN SADER, DEBADI CHAKRABORTY, The University of Melbourne, MATTHEW PELTON, University of Maryland, Baltimore County, EDWARD MALACHOSKY, PHILIPPE GUYOT-SIONNET, University of Chicago, KUAI YU, TODD MAJOR, MARY SAJINI DEVADAS, GREGORY HARTLAND, University of Notre Dame — Recent measurements show that the natural viscoelastic response of simple nominally Newtonian liquids, like water and glycerol, can be interrogated directly using the high frequency (20 GHz) vibration of nanomaterials. The extensional mode vibrations of bipyrimal gold nanoparticles were used, generating a predominantly incompressible shear flow. Here, we study the complementary and general case of compressible viscoelastic nanoscale flows. We show that all available constitutive models for these flows, with the exception of a very recent proposal, do not reproduce the required response at high frequency. We demonstrate the utility of this recent model through measurements of the breathing mode vibrations of single gold nanowires immersed in glycerol, over the 40-70 GHz frequency range.

9:05AM A12.00006 Shear alignment of lamellar mesophase systems1, JAJU S.J., KUMARAN V., Indian Institute of Science, Bangalore 560 012, India. — Mixtures of oil, water and surfactants form different microphases. Some of these phases, e.g. lamellar, hexagonal phases, lead to complex rheological behaviour at macroscale due to inherent anisotropy and irregularities in the microstructures. We present a comprehensive simulation study to examine the structure-rheology relationship in lamellar phase flow. At mesoscale, Reynolds number (Re), Schmidt number (Sc), Ericksen number (Er), extent of segregation between hydrophilic and hydrophobic components (r), ratio of viscosity of the two components (µ/µo), and system size to layer width ratio (L/λ) complete the lamellar phase description. We have used lattice Boltzmann simulations to study a two dimensional lamellar phase system of moderate size. The domains and grain boundaries seen at low Sc are replaced by isolated edge dislocations at high Sc. The alignment mechanism does not change with changes in layer bending moduli (Er), viscosity contrast or r. Increasing segregation, increases disorder; this however does not lead to higher resistance to flow. At high Er, the shear tries homogenise the concentration field and disrupt layer formation. We see significantly higher peak viscosity at low Er at high viscosity contrast and due to defect pinning.

9:18AM A12.00007 An immersed boundary method for fluid-structure interactions in a nematic liquid crystal. SAVERIO SPAGNOLIE, University of Wisconsin-Madison — The nematic phase of a liquid crystal is characterized by a spontaneous local molecular alignment leading to an anisotropic (direction-dependent) response to deformations. A body moving through such a phase can induce complex viscous and elastic structures in the flow, and the fluid’s anisotropic response can generate surprising forces on the immersed body. Bacteria swimming in a liquid crystal, for instance, have been observed to align with the orientation of the underlying director field. The complexity of such problems generally makes mathematical analysis intractable, and the computation of solutions can still be very challenging. In this talk an immersed boundary method for computing fluid-structure interactions in a nematic liquid crystal will be discussed. The Ericksen-Leslie equations, or a more general Landau-de Gennes model, are solved on a fixed, regular grid. Immersed boundaries communicate forces onto the fluid as in Peskin’s original method, but now also torques on the nematic director field through molecular anchoring boundary conditions. Sample applications will also be discussed, including the locomotion of undulatory bodies in anisotropic fluids.

9:31AM A12.00008 Viscoelastic Flow Modelling for Polymer Flooding. SHAUVIK DE, JOHAN PADDING, FRANK PETERS, HANS KUIPERS, Eindhoven University of Technology, MULTI-SCALE MODELLING OF MULTI-PHASE FLOWS TEAM — Polymer liquids are used in the oil industry to improve the volumetric sweep and displacement efficiency of oil from a reservoir. Surprisingly, it is not only the viscosity but also the elasticity of the displacing fluid that determine the displacement efficiency. The main aim of our work is to obtain a fundamental understanding of the effect of fluid elasticity, by developing an advanced computer simulation methodology for the flow of non-Newtonian fluids through porous media. We simulate a 3D unsteady viscoelastic flow through a converging diverging geometry of realistic pore dimension using computational fluid dynamics (CFD). The primitive variables velocity, pressure and extra stresses are used in the formulation of models. The viscoelastic stress part is formulated using a FENE-P type of constitutive equation, which can predict both shear and elongational stress properties during this flow. A Direct Numerical Simulation (DNS) approach using Finite volume method (FVM) with staggered grid has been applied. A novel second order Immersed boundary method (IBM) has been incorporated to mimic porous media. The effect of rheological parameters on flow characteristics has also been studied. The simulations provide an insight into 3D flow asymmetry at higher Deborah numbers. Micro-Particle Image Velocimetry experiments are carried out to obtain further insights. These simulations present, for the first time, a detailed computational study of the effects of fluid elasticity on the imbibition of an oil phase.

9:44AM A12.00009 A computational study of two-phase viscoelastic systems in a capillary tube with a sudden contraction/expansion1, METIN MURADOGLU, DAULET IZBASSAROV, Koc University — Two-phase viscoelastic systems are computationally studied in a pressure-driven tube with a sudden contraction and expansion using a finite-difference/front-tracking method. The effects of viscoelasticity in drop and bulk fluids are investigated including high Weissenberg and Reynolds number cases up to W i = 100 and Re = 100. The FENE-CR model is used to account for the fluid viscoelasticity. Extensive computations are performed to examine drop dynamics over a wide range of parameters. It is found that viscoelasticity interacts with drop interface in a non-monotonic and complicated way, and the two-phase viscoelastic systems exhibit very rich dynamics especially in the expansion region. At high Re, the drop undergoes large deformation in the contraction region followed by shape oscillations in the downstream of the expansion. For a highly viscous drop, a re-entrant cavity develops in the contraction region at the trailing edge which, in certain cases, grows and eventually causes encapsulation of ambient fluid. The re-entrant cavity formation is initiated at the entrance of the contraction and is highly influenced by the viscoelasticity. The effects of viscoelasticity are reversed in the constricted channel: Viscoelasticity in drop/continuous phase hinders/enhances format

1The authors would like to thank the Department of Science and Technology, Government of India for financial support, and Supercomputer Education and Research Centre at Indian Institute of Science for the computational resources.

Sunday, November 22, 2015 8:00AM - 9:44AM — Session A13 Free Surface Flows I: Wakes and Turbulence 201 - Laura Pauley, Penn State University
developing turbulent boundary layer in order to study the modification of the boundary layer flow field due to the effects of the water free surface. began. Cinematic Stereo PIV measurements are performed in planes parallel to the free surface by imaging the flow from underneath the tank ship moving at the same velocity, with a length equivalent to the length of belt that has passed the measurement region since the belt motion Re 

is a critical combination of Fr and We numbers -based on the local momentum thickness- that plays a critical role to the onset of entrainment. consider a turbulent boundary layer developing over an infinitely long moving plate. Our primary objective is to test the hypothesis that there the side of surface vessels, is characterized by highly turbulent bubbly region close to the surface and two-phase mixing. In this study, we will

The entrainment of air at the free-surface of a turbulent boundary layer remains a poorly understood problem. This flow, typically found at 

— Turbulent fluctuations in the vicinity of the water free surface along a flat, vertically oriented surface-piercing plate are studied experimentally using a laboratory-scale experiment. In this experiment, a meter-wide stainless steel belt travels horizontally in a loop around two rollers with vertically oriented axes, which are separated by 7.5 meters. This belt device is mounted inside a large water tank with the water level set just below the top edge of the belt. The belt, rollers, and supporting frame are contained within a sheet metal box to keep the device dry except for one 6-meter-long straight test section between rollers. The belt is launched from rest with a 3-g acceleration in order to quickly reach steady state velocity. This creates a temporally evolving boundary layer analogous to the spatially evolving boundary layer created along a flat-sided ship moving at the same velocity, with a length equivalent to the length of belt that has passed the measurement region since the belt motion began. Turbulent surface PIV measurements are performed in planes parallel to the free surface by imaging the flow from underneath the tank in order to study the modification of the boundary layer flow field due to the effects of the water free surface.

The 2D+t computation approximates the development of the wake at a fixed location as an object moves past but applies cyclical boundary conditions in the streamwise direction. A modified (Parabolized) Navier-Stokes (PNS) method has the same numerical efficiency as the classical 2D+t method but includes additional streamwise gradient terms derived in the transformation from a moving reference frame to a fixed reference frame. The present paper aims to assess the capability of the 2D+t and PNS methods for a laminar/turbulent wake interacting with a surface wave described by the Stokes velocity distribution. Results from 3D simulations will be used for validation to find criteria where 2D+t and PNS methods deliver results with an acceptable level of accuracy.

8:13AM A13.00002 Modeling variable density turbulence in the wake of an air-entraining transom stern1 . KELLI HENDRICKSON, DICK YUE, Massachusetts Institute of Technology — This work presents a priori testing of closure models for the incompressible highly-variable density turbulent (IHVDT) flows in the near wake region of a transom stern. This three-dimensional flow is comprised of convergent corner waves that originate from the body and collide on the ship center plane forming the “rooster tail” that then widens to form the divergent wave train. These violent free-surface flows and breaking waves are characterized by significant turbulent mass flux (TMF) at Atwood number At = (ρ₂ - ρ₁)/(ρ₂ + ρ₁) ≈ 1 for which there is little guidance in turbulence closure modeling for the momentum and scalar transport along the wake. To wht, this work utilizes high-resolution simulations of the near wake of a canonical three-dimensional transom stern using conservative Volume-of-Fluid (cVOF), implicit Large Eddy Simulation (ILES), and Boundary Data Immersion Method (BDIM) to capture the turbulence and large scale air entrainment. Analysis of the simulation results across and along the wake for the TMF budget and turbulent anisotropy provide the physical basis of the development of multiphase turbulence closure models. Performance of isotropic and anisotropic turbulent mass flux closure models will be presented.

8:26AM A13.00003 A Study of Water Wave Wakes of Washington State Ferries1 . BRADLEY PERFECT, JAMES RILEY, JIM THOMSON, University of Washington, ENDICOTT FAY, Washington State Ferries — Washington State Ferries (WSF) operates a ferry route that travels through a 600m-wide channel called Rich Passage. Concerns of shoreline erosion in Rich Passage have prompted this study of the generation and propagation of surface wave wakes caused by WSF vessels. The problem was addressed in three ways: analytically, using an extension of the Kelvin wake model by Darmon et al. (J. Fluid Mech., 738, 2014); computationally, employing a RANS Navier-Stokes model in the CFD code OpenFOAM which uses the Volume of Fluid method to treat the free surface; and with field data taken in Sept-Nov, 2014, using a suite of surface wave measuring buoys. This study represents one of the first times that model predictions of ferry boat-generated wakes can be tested against measurements in open waters. The results of the models and the field data are evaluated using a root mean square comparison of predicted and measured surface wave height as well as other metrics. Furthermore, the model predictions and field measurements suggest differences in wave amplitudes for different class vessels. Finally, the relative strengths and weaknesses of each prediction method as well as of the field measurements will be discussed.

8:39AM A13.00004 Footprints of turbulence over a viscous liquid . MARC RABAUD, ANNA PAQUIER, FREDERIC MOISY, Laboratoire FAST, Orsay — We observe the dynamics of tiny deformations at the surface of a viscous liquid sheared by a turbulent airflow using Free-Surface Synthetic Schlieren, which allows for time-resolved measurements of the topography with a micrometric accuracy. We are interested here in the low-velocity regime, before the onset of quasi-monochromatic wind waves. In this regime, the observed small and disorganized surface deformations directly result from the applied turbulent pressure field filtered by viscous and capillary effects. The amplitude of the footprints is found to increase linearly with air velocity, and the spatio-temporal dynamics is compatible with the known dynamics of the streaks of the turbulent boundary layer over a flat rigid wall.

8:52AM A13.00005 The Turbulent Boundary Layer Near the Air-Water Interface on a Surface-Piercing Flat Plate1 . NATHAN WASHUTA, NAEEM MASNADI, JAMES H. DUNCAN, University of Maryland — Turbulent fluctuations in the vicinity of the water free surface along a flat, vertically oriented surface-piercing plate are studied experimentally using a laboratory-scale experiment. In this experiment, a meter-wide stainless steel belt travels horizontally in a loop around two rollers with vertically oriented axes, which are separated by 7.5 meters. This belt device is mounted inside a large water tank with the water level set just below the top edge of the belt. The belt, rollers, and supporting frame are contained within a sheet metal box to keep the device dry except for one 6-meter-long straight test section between rollers. The belt is launched from rest with a 3-g acceleration in order to quickly reach steady state velocity. This creates a temporally evolving boundary layer analogous to the spatially evolving boundary layer created along a flat-sided ship moving at the same velocity, with a length equivalent to the length of belt that has passed the measurement region since the belt motion began. The present study of surface PIV measurements are performed in planes parallel to the free surface by imaging the flow from underneath the tank in order to study the modification of the boundary layer flow field due to the effects of the water free surface.

The entrainment of air at the free-surface of a turbulent boundary layer remains a poorly understood problem. This flow, typically found at the side of surface vessels, is characterized by highly turbulent bubbly region close to the surface and two-phase mixing. In this study, we will consider a turbulent boundary layer developing over an infinitely long moving plate. Our primary objective is to test the hypothesis that there is a critical combination of Fr and We numbers -based on the local momentum thickness- that plays a critical role to the onset of entrainment. In particular, we will report two-phase. DNS of a temporally developing turbulent boundary layer with waterside Reynolds number ranging from Re₉₀₀ = 900 to 1200. The computational domain is large enough to accommodate the range of eddies found in such flow. We utilize a conservative solver, where the air-water interface is sharply defined using a level-set formulation. Turbulent statistics away from the surface are presented. Rate of entrainment and the range of scales of entrained droplets are considered. Conditional averages of the flow field (i.e. vorticity, curvature, etc.) are reported to identify the mechanisms that trigger air-entrainment.

Supported by ONR N000141110588 monitored by Dr. Thomas Fu
9:18AM A13.00007 Micro-swimmer dynamics in free-surface turbulence subject to wind stress, CRISTIAN MARCHIOLI, SALVATORE LOVECHIO, ALFREDO SOLDATI, University of Udine — We examine the effect of wind-induced shear on the orientation and distribution of motile micro-swimmers in free-surface turbulence. Winds blowing above the air-water interface can influence the distribution and productivity of motile organisms via the shear generated just below the surface. Swimmer dynamics depend not only by the advection of the fluid but also by external stimuli like nutrient concentration, light, gravity. Here we focus on gyrotaxis, resulting from the gravitational torque generated by an asymmetric mass distribution within the organism. The combination of such torque with the viscous torque due to shear can reorient swimmers, reducing their vertical migration and causing entrainment in horizontal fluid layers. Through DNS-based Euler-Lagrangian simulations we investigate the effect of wind-induced shear on the motion of gyrotactic swimmers in turbulent open channel flow. We consider different wind directions and swimmers with different reorientation time (reflecting the ability to react to turbulent fluctuations). We show that only stable (high-gyrotaxis) swimmers may reach the surface and form densely concentrated filaments, the topology of which depends on the wind direction. Otherwise swimmers exhibit weaker vertical fluxes and segregation at the surface.

9:31AM A13.00008 Understanding capillary wave turbulence using discrete quasi-resonant kinetic equation, YULIN PAN, DICK YUE, Massachusetts Institute of Technology — Weak turbulence power-law spectrum can be physically understood from the kinetic equation (KE), which governs the evolution of wave spectrum due to nonlinear resonant interactions. For capillary waves, KE yields a stationary solution of a power-law spectrum, with energy flux from large to small scales due to triad resonant interactions. The condition of triad resonance, however, may not be satisfied if wavenumber can only take discrete values. This happens physically when the wavenumber is finite, or numerically when discrete wavenumber grid is used. Under this situation, energy flux is governed by quasi-resonant interactions; KE is not directly applicable and the underlying physics is not fully understood. We conduct a numerical study of KE on a discrete grid, where the frequency mismatch \( \Delta \omega \) of a triad is restrained from being nonzero. The energy transfer within such triads is accounted for by a generalized delta function \( \delta_\omega(\Delta \omega) \), which obtains its maximum at \( \Delta \omega = 0 \) and rapidly decreases as \( \Delta \omega \) increases. The width \( \epsilon \) of \( \delta_\omega(\Delta \omega) \) thus characterizes the nonlinear broadening. The simulation results elucidate the physics for different levels of nonlinear broadening relative to a given discrete grid.

Sunday, November 22, 2015 8:00AM - 9:57AM — Session A14 Industrial Applications I 202 - Ruo-Qian Wang, MIT

8:00AM A14.00001 A Respiratory Airway-Inspired Low-Pressure, Self-Regulating Valve for Drip Irrigation, RUO-QIAN WANG, AMOS G. WINTER, Dept. of Mechanical Engineering, MIT, GEAR LAB TEAM — One of the most significant barriers to achieving large-scale dissemination of drip irrigation is the cost of the pump and power system. An effective means of reducing power consumption is by reducing pumping pressure. The principle source of pressure drop in a drip system is the high flow resistance in the self-regulating flow resisters installed at the outlets of the pipes, which evenly distribute water over a field. Traditional architectures require a minimum pressure of \( \sim 1 \) bar to maintain a constant flow rate; our aim is to reduce this pressure by 90% and correspondingly lower pumping power to facilitate the creation of low-cost, off-grid drip irrigation systems. This study presents a new Starling resistor architecture that enables the adjustment of flow rate with a fixed minimum pressure demand of \( \sim 0.1 \) bar. A Starling resistor is a flexible tube subjected to a transmural pressure, which collapses the tube to restrict flow. Our design uses a single pressure source to drive flow through the flexible tube and apply a transmural pressure. Flow into the flexible tube is restricted with a needle valve, to increase the transmural pressure. Using this device, a series of experiments were conducted with different flexible tube diameters, lengths and wall thickness. We found that the resistance of the needle valve changes flow rate but not the minimum transmural pressure required to collapse the tube. A lumped-parameter model was developed to capture the relationships between valve openings, pressure, and flow rates.

8:13AM A14.00002 A Phase-Field Method for Simulating Fluid-Structure Interactions in Multi-Phase Flow, XIAONING ZHENG, Massachusetts Institute of Technology, GEORGE KARNIADAKIS, Brown University — We investigate two-phase flow instabilities by numerical simulations of fluid structure interactions in two-phase flow. The first case is a flexible pipe conveying two fluids, which exhibits self-sustained oscillations at high Reynolds number and tension related parameter. Well-defined two-phase flow patterns, i.e., slug flow and bubbly flow, are observed. The second case is external two-phase cross flow past a circular cylinder, which induces a Kelvin-Helmholtz instability due to density stratification. We solve the Navier-Stokes equation coupled with the Cahn-Hilliard equation and the structure equation in an arbitrary Lagrangian Eulerian (ALE) framework. For the fluid solver, a spectral/hp element method is employed for spatial discretization and backward differentiation for time discretization. For the structure solver, a Galerkin method is used in Lagrangian coordinates for spatial discretization and the Newmark-\( \beta \) scheme for time discretization.

8:26AM A14.00003 Shape optimisation of an underwater Bernoulli gripper, TIM FLINT1, MATHIEU SELLIER2, University of Canterbury — In this work, we are interested in maximising the suction produced by an underwater Bernoulli gripper. Bernoulli grippers work by exploiting low pressure regions caused by the acceleration of a working fluid through a narrow channel, between the gripper and a surface. This mechanism is used for non-contact adhesion to various surfaces and may be used to hold a robot to the hull of a ship while it inspects welds for example. A Bernoulli type pressure analysis was used to model the system with a Darcy friction factor approximation to include the effects of frictional losses. The analysis involved a constrained optimisation in order to avoid cavitation within the mechanism which would result in decreased performance and damage to surfaces. A sensitivity based method and gradient descent approach was used to find the optimum shape of a discretised surface. The model’s accuracy has been quantified against finite volume computational fluid dynamics simulation (ANSYS CFX) using the \( k-\omega \) SST turbulence model. Preliminary results indicate significant improvement in suction force when compared to a simple geometry by retaining a pressure just above that at which cavitation would occur over as much surface area as possible.

1Doctoral candidate in the Mechanical Engineering Department of the University of Canterbury, New Zealand.
2Associate Professor in the Mechanical Engineering Department of the University of Canterbury, New Zealand.
8:39AM A14.00004 Flow control of a centrifugal fan in a commercial air conditioner. 

JiYu Kim, KyEongTae Bang, HaeCheon Choi, Seoul National University, EungRyeol Seo, YongHun Kang, Samsung Electronics

— Air-conditioning fans require a low noise level to provide user comfort and quietness. The aerodynamic noise sources are generated by highly unsteady, turbulent structures near the fan blade. In this study, we investigate the flow characteristics of a centrifugal fan in an air-conditioner indoor unit and suggest control ideas to develop a low noise fan. The experiment is conducted at the operation condition where the Reynolds number is 163000 based on the blade tip velocity and chord length. Intermittent separation occurs at the blade leading edge and thus flow significantly fluctuates there, whereas vortex shedding occurs at the blade trailing edge. Furthermore, the discharge flow observed in the axial plane near the shroud shows low-frequency intermittent behaviors, resulting in high Reynolds stresses. To control these flow structures, we modify the shapes of the blade leading edge and shroud of the centrifugal fan and obtain noise reduction. The flow characteristics of the base and modified fans will be discussed.

1Supported by 0420-20130051

9:05AM A14.00006 The Other Source of Inducer Backflow. 

Tate Fanning, Ryan Lundgreen, Daniel Maynes, Stephen Correll, Brigham Young Univ - Provo, Kerry Oliphant, Concepts NREC — High suction performance inducers are used as a first stage in turbopumps to hinder cavitation and promote stable flow. Despite the distinct advantages of inducer use, an undesirable region of backflow and resulting cavitation can form near the tips of the inducer blades. This flow phenomenon has long been attributed to tip leakage flow, or the flow induced by the pressure differential between pressure and suction sides of an inducer blade at the tip. We examine backflow of a single inducer geometry at varying flow coefficients with a tip clearance of 0.4 mm and a tip clearance of 0.95 mm. Removing the tip clearance removes any potential tip leakage flow. Despite the removal of the tip leakage flow, backflow persists, and is essentially unaffected. We have observed backflow penetrating 1.1 tip diameters upstream of the leading edge in the inducer with tip clearance, and 0.95 tip diameters in the inducer without tip clearance under the same flow coefficient. A comprehensive analysis of these simulations suggests that blade inlet diffusion, not tip leakage flow, is the driving force for the formation of tip backflow.

9:18AM A14.00007 Design tools for microstructured optical fiber fabrication. 

Peter Buchak, Darren Crowdy, Imperial College London, Yvonne Stokes, Michael Chen, University of Adelaide — The advent of microstructured optical fibers (MOFs) has opened up possibilities for controlling light not available with conventional optical fiber. A MOF, which differs from a conventional fiber by having an array of air channels running along its length, is fabricated by heating and drawing a glass preform at low Reynolds number. However, because surface tension causes the cross section to deform, the geometry of the channels in the fiber differs from the preform. As a result, fabricating a desired fiber configuration may necessitate extensive trial and error. In this talk, we describe our work on fiber drawing, which has led to methods for predicting the fiber geometries that result at given draw conditions. More importantly, our methods can be used to obtain the preform configuration required to produce a fiber with a desired arrangement of channels. We have implemented our methods in software tools to facilitate preform design.

9:31AM A14.00008 Addressing heat transfer and uniformity of flow in the Muon g-2 tracker design.

Nicholas A. Pohlman, Guanrong Luo, Andrew Behnke, Northern Illinois University — Recent experiments in high energy physics found a possible deviation in the predicted value in the magnetic dipole moment of the muon particle within the standard model. To explore this term with higher precision, the Muon g-2 experiment (E989) is being built with an integrated tracker system inside the vacuum chamber where typical conductive and convective heat transfer methods are not available. The placement and packing of mylar straws and electronics filled with an argon gas mixture are designed to maximize the resolution for tracking decaying orbits. Using the space and magnetic field constraints, simulations in heat transfer and gas flow are presented to demonstrate the feasibility of the design to maintain temperature of electronic circuits and uniformity of gas replenishment in the tracker straw tubes. Results will show that initial estimates of using argon gas for electronics cooling is insufficient therefore requiring concentric-tube liquid cooling. Additionally, the impedance paths of the gas through the straw end pieces is dependent on features of both the radial and axial orientation. Preliminary data of prototype performance during a Summer 2015 beam test experiment will also be reported.

1Funding provided by FermiLab and the Department of Energy

9:44AM A14.00009 Massive separation around bluff bodies: comparisons among different cfd solvers and turbulence models.

Vincenzo Armenio, Ahmad Fakhari, Univ of Trieste - Trieste, Andrea Petronio, IFLUIDS s.r.l., Roberta Padovan, Chiara Pittaluga, Giovanni Caprino, Cetena, Italy — Massive flow separation is ubiquitous in industrial applications, ruling drag and hydrodynamic noise. In spite of considerable efforts, its numerical prediction still represents a challenge for CFD models in use in engineering. Aside commercial software, over the latter years the opensourse software OpenFOAM® (OF) has emerged as a valid tool for prediction of complex industrial flows. In the present work, we simulate two flows representative of a class of situations occurring in industrial problems: the flow around sphere and that around a wall-mounted square cylinder at $Re = 10000$. We compare the performance two different tools, namely OF and ANSYS CFX 15.0 (CFX) using different unstructured grids and turbulence models. The grids have been generated using SNAPPYHEX-MESH and ANSYS ICEM CFD 15.0 with different near wall resolutions. The codes have been run in a RANS mode using $k-\epsilon$ model (OF) and SST $k-\omega$ (CFX) with and without wall-layer models. OF has been also used in LES, WMLES and DES mode. Regarding the sphere, RANS models were not able to catch separation, while good prediction of separation and distribution of stresses over the surface were obtained using LES, WMLES and DES. Results for the second test case are currently under analysis.

1Financial support from COSMO “cfd open source per opera morta” PAR FSC 2007-2013, Friuli Venezia Giulia
Appendage shapes in the context of the IPL instability. On a sphere increase the side force significantly compared to trivial shapes (such as an elliptic sheet). We also find that appendages may be

Characteristics past various bluff-body configurations, SUNAKRANENI SOUMYA, Graduate student, Indian Institute of technology Madras, K. ARUL PRAKASH, Associate Professor, Indian Institute of technology Madras — Numerical simulation of five different bluff body configurations with splitter plate is carried out to analyse the fluid flow and heat transfer characteristics for Reynolds number (Re) ranging from 50-200. The governing equations are discretized using SUPG - finite element method. The bluff body configurations considered are elliptic cylinder of axis ratios (AR=0.5-1.0), square cylinder, rhombus of axis ratios (AR=0.5-1.0), equilateral triangle, and semi-circular cylinder. The splitter plate length varied from L=0.0Dc-6.0Dc, (Dc = Bluff body hydraulic diameter). It is observed that interaction of separated shear layers from top and bottom surfaces of the body is inhibited and vortex shedding is suppressed for certain combinations of bluff body configuration, Re and splitter plate length and wake region is modified significantly. Reduction in drag approximately of the order 2% to 50% is attained and overall heat transfer (Q) is increased due to splitter plate.

The effect of splitter plate on fluid flow and heat transfer characteristics past various bluff-body configurations, SUNAKRANENI SOUMYA, Graduate student, Indian Institute of technology Madras, K. ARUL PRAKASH, Associate Professor, Indian Institute of technology Madras — Numerical simulation of five different bluff body configurations with splitter plate is carried out to analyse the fluid flow and heat transfer characteristics for Reynolds number (Re) ranging from 50-200. The governing equations are discretized using SUPG - finite element method. The bluff body configurations considered are elliptic cylinder of axis ratios (AR=0.5-1.0), square cylinder, rhombus of axis ratios (AR=0.5-1.0), equilateral triangle, and semi-circular cylinder. The splitter plate length varied from L=0.0Dc-6.0Dc, (Dc = Bluff body hydraulic diameter). It is observed that interaction of separated shear layers from top and bottom surfaces of the body is inhibited and vortex shedding is suppressed for certain combinations of bluff body configuration, Re and splitter plate length and wake region is modified significantly. Reduction in drag approximately of the order 2% to 50% is attained and overall heat transfer (Q) is increased due to splitter plate.

Investigation and control of dynamic stall of an aerofoil ramp up motion, MARCO EDOARDO ROSTI, MOHAMMAD OMIDYEGANEH, ALFREDO PINELLI, City University London — Direct Numerical Simulations of the flow around a NACA0020 aerofoil at Re = 20 x 10^3 undergoing a ramp up motion has been undertaken (α ∈ [0°, 20°], α˙c/|U|∞ = 0.12). New insights on the vorticity dynamics in the baseline case are discussed using a number of post-processing techniques. We will present and discuss the effects of a passive control technique based on the use of a thin flap hinged via a torsional spring to the suction side of the aerofoil. The interaction between the flap dynamics (modelled as an infinitely thin plate) and the fluid have been carried out using an original Immersed Boundary Method applied to a finite volume solver. When the spring constant is chosen to lock the flap oscillations into the main shedding frequency, the back flow induced by the primary vortex is strongly reduced by the presence of the flap. Moreover, the flap is capable to enhance and protract the lift overshoot typical of the dynamic stall also alleviating the subsequent lift-breakdown. These beneficial behaviour is mainly due to the establishment of a fluid structure interaction cycle that continuously regenerate the primary vortex which is ultimately responsible for the enhanced lift.

Control of Vortex Sheddin on an Airfoil using Mini Flaps at Low Reynolds Number, DAISUKE OSHIYAMA, DAIJU NUMATA, KEISUKE ASAI, Tohoku University — In this study, the effects of mini flaps (MFs) on a NACA0012 airfoil were investigated experimentally at low Reynolds number. MFs are small flat plates attached to the trailing edge of an airfoil perpendicularly. All the tests were conducted at the Tohoku-University Basic Aerodynamic Research Tunnel at the chord Reynolds number of 25,000. Aerodynamic forces were measured using a 3-component balance and the surface flow was visualized by luminescent oil film technique. The results of force measurement show that attachment of MFs enhances lift and the enhanced lift increases with MF height. On the other hand, the results of oil flow visualization show that attached height of MFs enlarges the separated region on the airfoil rather than diminishes it. To understand the physical mechanism of MFs for lift enhancement, the flow around the airfoil was visualized by the smoke-wire method and the wake profile behind the airfoil was measured using a hot wire anemometer. It was found that vortexes shed periodically from the tip of the MFs and interact with the separated shear layer from the upper surface. This unsteady vortex shedding forms a low-pressure region on the upper surface, generating higher lift. These results suggest that the height of MFs controls the frequency of vortex shedding behind the MF, forcing the separated shear layer on the upper surface flow in unsteady manner.

Experimental investigation of flow past a sphere with trip, RAHUL DESHPANDE, ADITYA DESAI, Indian Institute of Technology Kanpur, Kanpur, U.P, India, VIVEK KANTTI, National Wind Tunnel Facility, Indian Institute of Technology Kanpur, Kanpur, U.P, India, SANJAY MITTAL, Indian Institute of Technology Kanpur, Kanpur, U.P, India — The flow over a smooth sphere and a sphere with a trip was experimentally investigated in the Reynolds number range 1 x 10^3 to 5 x 10^5 through unsteady force measurements. The size of the trip is 1.5 percent of the diameter D of the sphere and measurements are made for its motion from the stagnation point for 10, 20 and 30 degrees. The statistics of the drag and lateral forces were studied for a range of subcritical to supercritical Reynolds numbers to understand the effect of a trip on the critical flow regime of a sphere. Two different flow characteristics are observed over the sphere surface depending on the streamwise location of the trip. For subcritical Reynolds numbers, a significant mean side force is observed in the direction of the trip side of the sphere. On gradually increasing the Reynolds number, the flow over the sphere enters the critical regime and the direction of the side force reverses from the trip side to the non - trip side of the sphere which continues to be observed well within the early supercritical regime.

The authors would like to thank the efforts put in by Mr. Sharad Saxena and Mr. Akhilesh Pal from National Wind Tunnel Facility, IIT - Kanpur in conducting the experiments.
Mixing referred to as the mix-norm, which is equivalent to the H-field, which in the case of a zero-mean field is the L-norm. A natural measure of homogeneity of a mixture is the variance of the concentration s of the index Skin. Numerically well-conditioned. It can be readily applied to complex geometry flows. Can be identified and used for controller design. This method avoids manipulating large matrices and is therefore computationally efficient and predictable, alterations to the flow properties.

The work is supported by the Dynamics and Control Program at AFOSR, grant FA9550-12-1-0114.

The importance of being fractional in mixing: optimal choice of the index s in $H^{-s}$ norm. A natural measure of homogeneity of a mixture is the variance of the concentration field, which in the case of a zero-mean field is the $L^2$-norm. Mathew et al. (Physica D 2005) introduced a new multi-scale measure to quantify mixing referred to as the mix-norm, which is equivalent to the $H^{-1/2}$ norm, the Sobolev norm of negative fractional index. Unlike the $L^2$-norm, the mix-norm is not conserved by the advection equation and thus captures mixing even in the non-diffusive systems. Furthermore, the mix-norm is consistent with the ergodic definition of mixing and Lin et al. (JFM 2011) showed that this property extends to any norm from the class $H^{-s}$, $s > 0$. We consider a zero-mean passive scalar field organised into two layers of different concentrations advected by a flow field in a torus. We solve two non-linear optimisation problems. We identify the optimal initial perturbation of the velocity field with given initial energy as well as the optimal forcing with given total action (the time integral of the kinetic energy of the flow) which both yield maximal mixing by a target time horizon. We analyse sensitivity of the results with respect to $s$-variation and thus address the importance of the choice of the fractional index.

This work was supported by the University of Cambridge Centre for Doctoral Training, the Cambridge Centre for Analysis.
8:39AM A16.00004 Contractive control design for Navier-Stokes systems with the incompressibility constraint relaxed. HUAN YU, POORIYA BEYHAGHI, THOMAS BEWLEY, University of California, San Diego, FLOW CONTROL LAB TEAM — One approach to the linear stabilization of near-wall transitional channel flow is via the Orr-Sommerfeld/Square equations. This formulation is delicate, as it reduces the three momentum equations and the divergence-free constraint of the incompressible NSE down to a highly non-normal set of two equations, one for the wall-normal velocity and one for the wall-normal vorticity, and involves inverting a Laplacian with boundary conditions embedded. A simple formulation for the purpose of control design is given by simply dropping the divergence-free constraint from the problem considered altogether, and at the same time dropping the pressure gradient from the momentum equations, which acts to enforce this constraint. What remains is three coupled Burgers equations. In general, there is no relationship between the stability of such constrained and unconstrained systems; however, if the unconstrained system is contractive (a condition stronger than just stability), the constrained system is also contractive. We have investigated this approach to control design for NS systems. We have proved a fundamental result: if an unconstrained, uncontrolled channel flow system is not contractive, there is no boundary control that can make it contractive.

8:52AM A16.00005 Control of laminar wake flows using the Sum-of-Squares approach. DAVIDE LASAGNA, OWEN TUTTY, University of Southampton, DEQING HUANG, SERGEI CHERNYSHENKO, Imperial College London — A novel feedback control design methodology for finite-dimensional, reduced-order models of incompressible turbulent fluid flows, aiming at reduction of long-time averages of key quantities, is presented. The key enabler is a recent advance in control design systems with polynomial dynamics, i.e. the discovery that the Sum-of-Squares decomposition of a non-negative polynomial, or the construction of one of such functions, can be computed via semidefinite programming techniques. Firstly, the theoretical difficulties of treating long-time averages are relaxed by abstracting the analysis to upper bounds of such averages. Then, the problems of estimation and optimisation via control design of these bounds are conveniently reformulated into constructing suitable non-negative polynomial functions, using Sum-of-Squares programming techniques. To showcase the methodology, linear and nonlinear polynomial-type state-feedback controllers are designed to reduce the fluctuations kinetic energy in the wake of a circular cylinder at Re = 100, using rotary oscillations. A compact, reduced-order Galerkin model of the actuated wake is first derived using Proper Orthogonal Decomposition. Controllers are then designed and implemented in direct numerical simulations of the flow.

9:05AM A16.00006 A suboptimal feedback control theory based on a quadratic sensitivity and tabulation approach. YEOJONG KIM, SEONGWON KANG1, Sogang Univ., Korea — The main objective of this study is to develop a new systematic flow control approach based on the suboptimal feedback control (SFC) theory by addressing a few issues from the previous theory, such as the time-dependent linear or nonlinear Schrödinger equation to control the time-dependent quantum channel flow. The objective is to derive a systematic controller based on partial differential equations. In the previous SFC theory, a physical assumption or tuning process is necessary for a user-defined parameter to control flows successfully. In the present study, this issue is addressed by introducing an appropriate optimality condition based on a quadratic control sensitivity. In order to build a practical control framework, the revised theory is reformulated as a tabulation approach using a modified Green’s function method, which achieves both efficiency and accuracy. The effectiveness of the proposed method is tested using laminar and turbulent flows such as a two-dimensional Taylor vortex problem and turbulent channel flow. As a result, the proposed approach shows a similar or better control performance compared to the previous one, without a need to determine a control parameter.

1 Corresponding author

9:18AM A16.00007 Control of Quantum Fluid Dynamics and Adaptive Phase Compensation for Laser Propagation in Turbulence. JONATHAN GUSTASSON, SIVAGURU S. SRITHARAN, Air Force Institute of Technology — Equations of High Energy Laser propagation in a turbulent medium and the equations of quantum fluid dynamics are connected through a mathematical transformation. In this way the problem of adaptive phase compensation can be phrased as an initial velocity control problem for quantum fluid dynamics. The quantum hydrodynamics equation can be derived by applying the Madelung transformation to the time-dependent linear or nonlinear Schrödinger equation. This quantum hydrodynamics equation can then be written in the form of the time-dependent linear or nonlinear Schrödinger equation, which is suitable for control design. The quantum hydrodynamics equation can thus be a good way to understand adaptive optics and laser propagation through the atmosphere. A Riemann solver within the Clawpack framework has been developed. An initial value optimization problem will be solved using adjoint methods. The initial phase can be controlled when the beam leaves the laser apparatus. The control method can also be coupled to a Navier-Stokes solver in order to study thermal blooming where the laser heats the air and changes the index of refraction. The change in refractive index will in turn affect the propagation of the Laser beam. Using optimal control techniques, it is possible to adjust the beam in order to compensate for the heating.

9:31AM A16.00008 Shell model of optimal passive-scalar mixing. CHRISTOPHER MILES, CHARLES DOERING, Univ of Michigan - Ann Arbor — Optimal mixing is significant to process engineering within industries such as food, chemical, pharmaceutical, and petrochemical. An important question in this field is “How should one stir to create a homogeneous mixture while being energetically efficient?” To answer this question, we consider an initially unmixed scalar field representing some concentration within a periodic domain. This passive-scalar field is advected by the velocity field, our control variable, constrained by physical quantity such as energy or enstrophy. We consider two objectives: local-in-time (LIT) optimization (what will maximize the mixing rate now?) and global-in-time (GIT) optimization (what will maximize mixing at the end time?). Throughout this work we use the H−1-norm of the quantity such as energy or enstrophy. To gain a better understanding, we provide a simplified mixing model by using a shell model of passive-scalar advection. LIT optimization in this shell model gives perfect mixing in finite time for the energy-constrained case and exponential decay to the perfect-mixed state for the enstrophy-constrained case. Although we only enforce that the time-average energy (or enstrophy) equals a chosen value in GIT optimization, interestingly, the optimal control keeps this value constant over time.

9:44AM A16.00009 Oscillatory flow past a slip cylindrical inclusion embedded in a Brinkman medium. D. PALANIAPPAN, Department of Mathematics & Statistics, Texas A&M University - Corpus Christi — Transient flow past a circular cylinder embedded in a porous medium is studied based on Brinkman model with Navier slip conditions. Closed form analytic solution for the stream-function describing slow oscillatory flow around a solid cylindrical inclusion is obtained in the limit of low-Reynolds-number. The key parameters such as the frequency of oscillation λ, the permeability constant δ, and the slip coefficient ξ dictate the flow fields and physical quantities in the entire flow domain. Asymptotic steady-state analysis when λ → 0 reveals the paradoxic behavior detected by Stokes. Local streamline patterns of small times demonstrate interesting flow patterns. Rapid transitions including flow separations and eddies are observed far away from the solid inclusion. Analytic expressions for the wall shear stress and the force acting on the cylinder are computed and compared with existing results. It is noted that the slip parameter in the range 0 ≤ ξ ≤ 0.5 has a significant effect in reducing the stress and force. In the limit of large permeability, Darcy (potential) flow is recovered outside a boundary layer. The results are of some interest in predicting maximum wall stress and pressure drop associated with biological models in fibrous media.
actuation programs leads to enhancement of cycle lift and pitch stability, and reduced cycle hysteresis. Evolution, and advection of the dynamic stall vortex, actuation during the downstroke accelerates flow attachment. Superposition of such actuation programs using multiple actuation bursts during the pitch cycle. While actuation during the upstroke primarily affects the formation, velocimetry that is acquired phased-locked to the motion of the airfoil. The aerodynamic loading can be significantly altered by a number of the airfoil are assessed using time-dependent measurements of the lift force and pitching moment coupled with time-resolved particle image velocimetry that are an order of magnitude shorter than the airfoil's convective time scale. The effects of the actuation on the aerodynamic characteristics that are responsible for the lift response are identified. 

To investigate the physical mechanisms that lead to the observed forces, we consider a simple model of two-dimensional flow over a NACA 0009 airfoil at moderate Reynolds number. We model actuation as a momentum source that imposes a specified velocity in a small region near the leading edge. The actuation parameters are varied to determine how the instantaneous and phase-averaged lift scale with the strength and duration of actuation. The results are compared with instantaneous and phase-averaged PIV data from the experiments, and the flow structures responsible for the lift response are identified.

Controlled separation of the entrance flow at the inlet of a branched channel is investigated experimentally in an air facility. Of particular interest is the formation and scaling of a separated flow downstream of the entrance plane into the secondary channel and its interaction with the flow surfaces at speeds up to $M = 0.4$. The separation is actively controlled using a spanwise array of fluidic actuators on the primary channel’s surface upstream of the inlet plane of the secondary duct. The effects of the actuation on the evolution of the separation and attachment of the vorticity layer between upstream surface of the primary duct and the surface of the secondary duct downstream of the branched inlet in the presence of a strong confined adverse pressure gradient are investigated using particle image velocimetry coupled with detailed static surface pressure distributions. The effects of the controlled separation within the secondary channel on the global flow within the primary duct and on flow split between primary and secondary channels are assessed, and it is demonstrated that actuation can effect significant changes in the flow fractions between the channels.

Copyright © 2015 Boeing. All rights reserved.

Aerodynamic Control using Distributed Active Bleed

Aerodynamic Control using Distributed Active Bleed

Aerodynamic Control using Distributed Active Bleed

Control of Trapped Vorticity in an Offset Diffuser

Control of Trapped Vorticity in an Offset Diffuser

Control of Trapped Vorticity in an Offset Diffuser

Aerodynamic Control of a Dynamically Pitching Airfoil using Transitory Pulsed Actuation

Aerodynamic Control of a Dynamically Pitching Airfoil using Transitory Pulsed Actuation

Aerodynamic Control of a Dynamically Pitching Airfoil using Transitory Pulsed Actuation

Aerodynamic Control of a Dynamically Pitching Airfoil using Transitory Pulsed Actuation

Aerodynamic Control of a Dynamically Pitching Airfoil using Transitory Pulsed Actuation

Aerodynamic Control of a Dynamically Pitching Airfoil using Transitory Pulsed Actuation

Supported by ONR
9:05AM A17.00006 Fast-flap Actuation for Attenuating Gust Response. MICHAEL OL, US Air Force Research Lab, KENNETH GRANLUND, North Carolina State University, ALBERT MEDINA, US Air Force Research Lab — Airfoil flow control actuators can respond at perhaps O(10E-3) convective-times, but the flowfield response requires typically 2-4 convective times, and initial force-transients can be negative. A conventional trailing-edge mechanical flap is “slow” to deflect in flight applications, of questionable efficacy in separated flows, and is plagued by a response nonlinear with deflection angle. We consider a half-chord airfoil flap actuated O(10) times faster than one convective time, taking advantage of scaling-effects in a water tunnel. The motivation is recent work on accelerating flat plate, and high incidence, where despite zero bound-circulation, the lift transients follow Wagner’s solution. Force-measurements for high-rate large-incidence flap deflection show similar trends, and offer promise in cancelling lift-transients from gusts (modeled by plunging or surging the airfoil). Parameter-studies of rate, amplitude and initial incidence suggest first-order-system relaxation to steady-state, with a time constant commensurate with 1-3 convective times, no negative transients and no discernible lag. Rapid flap actuation induces entrainment that augments the flow’s propensity to attach/reattach, perhaps paradoxically comporting with theory better, the faster the actuation.

9:18AM A17.00007 Control of flow separation in a turbulent boundary layer, MINJEONG CHO, SANGHO CHOI, HAECHOEN CHOI, Seoul National University — Towards the development of successful control methods for separation delay in a turbulent boundary layer, we adopt a model flow field, in which a turbulent separation occurs above a flat plate (Na and Moin 1998 JFM), and apply controls to this flow for reducing the size of the separation bubble and investigating the interaction between the forcing and flow near the separation bubble. We provide a single-frequency forcing with zero net mass flow rate at the upstream of the separation bubble. At low forcing frequencies, spanwise vortices are generated and travel downstream, bringing high momentum toward the wall and reducing the size of the separation bubble. Also, these vortices cause the separation and reattachment points to travel downstream. On the other hand, at high forcing frequencies, the size of the separation bubble becomes smaller and larger in time, respectively, due to the pressure gradient alternating favorably and adversely in time.

9:31AM A17.00008 Field Test Results from a 10 kW Wind Turbine with Active Flow Control. THOMAS RICE, VERONIKA BYCHKOVA, KEITH TAYLOR, Rensselaer Polytechnic Institute, DAN CLINGMAN, The Boeing Company, MICHAEL AMITAY, Rensselaer Polytechnic Institute — Active flow control devices including synthetic jets and dynamic vortex generators were tested on a 10 kW wind turbine at RPI. Previous work has shown that load oscillations caused by dynamic stall could be modified through the use of active flow control by injecting momentum into the flow field near the leading edge of a dynamically pitching model. In this study, this work has been extended to its logical conclusion, field-testing active flow control on a real wind turbine. The blades in the current study have a 0.28m chord and 3.05m span, no twist or taper, and were retrofitted with six synthetic jets on one blade and ten dynamic vortex generators on a second blade. The third blade of this turbine was not modified, in order to serve as a control. Strain gauges were installed on each blade to measure blades’ deflection. A simple closed loop control was demonstrated and preliminary results indicate reduced vibrational amplitude. Future testing will be conducted on a larger scale, 600kW machine at NREL, incorporating information collected during this study.

9:44AM A17.00009 Performance Comparison of Sweeping/Steady Jet Actuators. DAMIAN HIRSCH, California Institute of Technology, JUSTIN MERCIER, FLAVIO NOCA, HEPIA - University of Applied Sciences Western Switzerland, MORTEZA GHARIB, California Institute of Technology — Flow control through the use of steady jet actuators has been used on various aircraft models since the late 1950’s. However, the focus of recent studies has shifted towards the use of sweeping jets (fluidic oscillators) rather than steady jet actuators. In this work, experiments using various jet actuator designs were conducted at GALCIT’s Lucas Wind Tunnel on a NACA 0012 vertical tail model similar to that of the Boeing 767 vertical stabilizer at Reynolds numbers ranging from 0.5 to 1.2 million. The rudder angle was fixed at 20 degrees. A total of 32 jet actuators were installed along the wingspan perpendicular to the trailing edge and the rudder shoulder of the vertical stabilizer. It is known that these types of flow control prevent separation. However, the goal of this work is to compare different jet designs and evaluate their performance. Parameters such as the number of actuators, their volumetric flow, and the wind tunnel speed were varied. The lift generation capabilities of steady and sweeping jet actuators were then compared. Another set of experiments was conducted to compare a new sweeping jet actuator design with one of the standard versions.

1Supported by Boeing

Sunday, November 22, 2015 8:00AM - 9:57AM — Session A18 Vortex Dynamics: Energy Harvesting and Atmospheric Flows 206 - George Triantafyllou, National Technical University of Athens

8:00AM A18.00001 Sensitivity of two-dimensional flow past transversely oscillating cylinder to streamwise cylinder oscillations. GEORGE TRIANTAFYLLOU, Department of Naval Architecture and Marine Engineering, National Technical University of Athens, Athens Greece, SOFIA PEPPA, Department of Naval Architecture, Technological Educational Institute of Athens, Athens, Greece — The sensitivity of flow past transversely oscillating cylinder to streamwise harmonic perturbations is studied. The value of the Reynolds number is equal to 150. We start with a transversely oscillating cylinder and then impose a small streamwise (in-line) perturbation with a frequency equal to twice the transverse oscillation frequency. The cylinder is thus following an eight-shaped trajectory, which can be traversed in a counter-clockwise or clockwise direction. For the counter-clockwise mode of motion, we find that, for low to moderate values of the streamwise amplitude, the power transfer from the fluid to the structure increases with the amplitude of oscillation in the streamwise direction, even though the magnitude of the fluctuations of the forces is decreased. The increase of the power transfer becomes more important at higher values of the streamwise amplitude of oscillation. For the clockwise mode of motion the lift transients show similar trends, and offer promise in cancelling lift transients from gusts (modeled by plunging or surging the airfoil).
DIXIT & GOVINDARAJAN, JFM, 2010, sharp density variations, and therefore cannot be neglected under such circumstances (see, e.g., changes in inertia due to changes in the density. However, the non-Boussinesq terms can lead to a kind of centrifugal instability for small but

for Interdisciplinary Sciences — The Boussinesq approximation, commonly employed in weakly compressible or incompressible flows, neglects

, S. RAVICHANDRAN, TIFR Centre for Interdisciplinary Sciences, HARISH DIXIT, IIT Hyderabad, RAMA GOVINDARAJAN, TIFR Centre

1 Supported by ARPA-E award DE-AR0000296

8:26AM A18.00003 Buoyancy-Induced, Columnar Vortices1, MARK SIMPSON, ARI GLEZER, Georgia Institute of Technology — Free buoyancy-induced, columnar vortices (dust devils) that are driven by thermal instabilities of ground-heated, stratified air in areas with sufficient insolation convert the potential energy of low-grade heat in the surface air layer into a vortex flow with significant kinetic energy. A variant of the naturally-occurring vortex is deliberately triggered and anchored within an azimuthal array of vertical, stator-like flow vanes that form an open-top enclosure and impant tangential momentum to the radially entrained air. This flow may be exploited for power generation by coupling the vortex to a vertical-axis turbine. The fundamental mechanisms associated with the formation, evolution, and dynamics of an anchored, buoyancy-driven columnar vortex within such a facility are investigated experimentally using a heated ground plane. Specific emphasis is placed on the manipulation of the vortex formation and structure and the dependence of the vorticity production and sustainment mechanisms on the thermal resources and characteristic scales of the anchoring flow vanes using stereo-PIV. It is shown that manipulation of the formation and advection of vorticity concentrations within the enclosure can be exploited for increasing the available kinetic energy.

1 Supported by ARPA-E

8:39AM A18.00004 Numerical Investigation of Synthetic Buoyancy-Induced Columnar Vortices1, NICHOLAS MALAYA, ROY STOGNER, ROBERT MOSER, University of Texas at Austin — Much of the solar energy incident on the Earth's surface is absorbed into the ground, which in turn heats the air layer above the surface. This buoyant air layer contains considerable gravitational potential energy. The energy can drive the formation of columnar vortices (Dust-Devils) which arise naturally in the atmosphere. These Dust-Devils occur over a wide range of scales in many different locations across the Earth, as well as on Mars. A new energy harvesting approach makes use of this ubiquitous process by creating and anchoring the vortices artificially and extracting energy from them. In this talk we explore the characteristics of these vortices through numerical simulation. Computational models of the turning vane system used to generate the vortex have been developed. We will discuss the formulation of these models and their validation against available experimental measurements. We will also describe the use of these simulations to optimize the turning vane configuration to maximize the power extraction, as well as serving as a vehicle to probe the dynamics of the underlying physical processes. Finally, this talk will conclude with comparisons between the synthetic vortices and the naturally occurring phenomena.

1 This work supported by the Department of Energy [ARPA-E] under Award Number [DE-FOA-0000670]

8:52AM A18.00005 Non-Boussinesq effects on vorticity and kinetic energy production

, S. RAVICHANDRAN, TIFR Centre for Interdisciplinary Sciences, HARISH DIXIT, IIT Hyderabad, RAMA GOVINDARAJAN, TIFR Centre for Interdisciplinary Sciences — The Boussinesq approximation, commonly employed in weakly compressible or incompressible flows, neglects changes in inertia due to changes in the density. However, the non-Boussinesq terms can lead to a kind of centrifugal instability for small but sharp density variations, and therefore cannot be neglected under such circumstances (see, e.g., DIXIT & GOVINDARAJAN, JFM, 2010, 415). Here, we study the evolution of a light-cored Gaussian vortex and find that the non-Boussinesq terms can lead to significant changes in how vortices evolve. The problem is governed by three nondimensional numbers—Reynolds number (i.e., viscosity), Atwood number, and a ratio of gravitational and centrifugal Froude numbers. We find that the production of kinetic energy and vorticity in a light-cored Gaussian vortex are affected significantly by the non-Boussinesq terms, and varies non-monotonically with the parameters of the problem. In general, these non-Boussinesq effects depend both on the strength of gravity and on the Reynolds number associated with the initial vortex.

9:05AM A18.00006 Flow hydrodynamics and contaminant transport in the flow past a lateral square cavity1, CRISTIAN ESCAURIAZA, JUAN IGNACIO POLANCO, Pontificia Universidad Catolica de Chile, OLIVIA AUGUST, DIOGO BOLSTER, University of Notre Dame — Turbulent flows past lateral cavities play an important role in the transport of contaminants in rivers and streams. Cavities are surface storage zones, where large-scale unsteady coherent structures are the leading mechanisms that produce longer residence times and control the fate of contaminants in the river. In this work we study the recirculating flow and mass transport in a lateral square cavity, by performing numerical simulations with a hybrid URANS/LES turbulence model (DES-LR). We focus on the dynamics of the coherent structures and their impacts on the transport and storage of a passive scalar. In addition, we use the numerical results to develop new 1D models that improve the description of the evolution of the averaged concentration inside the cavity. By transferring the information to larger spatial scales, we provide new insights on the mechanisms of contaminant transport and analyze the overall effects of surface storage zones in open channel flows.

1 supported by Fondecyt grant 1130940
9:18AM A18.00007 Propulsive effects of vortex coupling between parallel pulsed jets. ATHANASIOS ATHANASSIADIS, DOUGLAS HART, Massachusetts Inst of Tech-MIT — For vehicles that use pulsed jet propulsion, nature provides inspiration for different ways to improve propulsive performance. Communities of marine invertebrates called salps improve the efficiency of cruising locomotion by aggregating into large multi-animal chains. In this process, the cylindrical animals physically connect to each other side-by-side to form an array of individual pulsed jets whose synchronous pulsing propels the entire chain forward. Some benefits of this chaining behavior can be described using existing models of pulsed jet propulsion for steady, cruising conditions. However, during unsteady conditions, the interactions between neighboring jets influence the overall propulsive performance. Using bench-top experiments, we investigate the unsteady interactions between two parallel pulsed jets. Under some conditions, the pulsed jets form vortex rings that coalesce before vortex formation is complete, coupling the hydrodynamics of the independent jets. We measure how different degrees of vortex coupling alter the energy and momentum transfer in the two-jet system. Finally, we explore the energy and momentum scalings that would guide the design of a vehicle using multi-jet maneuvering techniques.

1This work was supported by the Office of Naval Research

9:31AM A18.00008 Vortex dynamics in high-speed rarefied cavity flows. VISHNU VENUGOPAL, GRIMAJI SHARATH S, Texas A&M Univ — Space-access vehicles are frequently exposed to high non-equilibrium conditions, particularly during an atmospheric re-entry. Any cavity on the surface of such vehicles including suction chambers and impact damages can drastically alter the aerodynamic behavior around the vehicle. For instance, even if the freestream is rarefied, the flow within the cavity could be close to continuum due to the entrainment and accumulation of many molecules. This can significantly change the dynamics of vortex patterns, that are commonly present in a cavity flow, eventually affecting the surface properties along cavity walls. So, it is important to characterize the vortex dynamics in a cavity flow as a function of the degree of rarefaction (Knudsen number), cavity size and flow speed. Direct numerical simulations are performed for lid driven cavity flows using a Unified Gas Kinetic Scheme. A parametric study is performed to quantitatively possible vortex configurations for a given combination of Knudsen number, cavity aspect ratio and lid velocity. The underlying physical mechanisms involved in the production of different vortex structures are highlighted. Finally, an effort is made to develop a reference diagram that clearly classify the regions of physically possible vortex configurations.

9:44AM A18.00009 Vortex dynamics and flapping patterns of the inverted flag with a bluff body. HYEONSEONG KIM, JUNYOUNG KIM, DAEGYOUM KIM, KAIST — Flow-induced vibration of flexible structures for energy harvesting has drawn attention recently. The inverted flag whose trailing edge is clamped and leading edge is free to move was known to self-excite in a uniform flow of both water and air. In this study, we investigated the effect of an upstream bluff body, a flat plate, on the dynamics of the downstream inverted flag. Periodic vortical structures created by an upstream bluff body make the dynamics of the inverted flag quite different from those of the inverted flag without the bluff body. We examined the motion of the inverted flag by varying the relative displacement of the inverted flag from the bluff body and their relative size. Our results show that the inverted flap can flap with higher frequency and larger amplitude with the upstream bluff body. We also compared the dynamics of the inverted flag with those of the general flag with the upstream flag. In order to better understand the dynamics of the flag, the analysis of the flow patterns using particle image velocimetry was also conducted.

Sunday, November 22, 2015 8:00AM - 9:57AM — Session A19 Boundary Layers: General

8:00AM A19.00001 Time-resolved Tomo-PIV measurements of the interaction between a stationary held sphere and a turbulent boundary layer. RENE VAN HOUT, Technion Israel Institute of Technology - Israel Institute of Technology, JERKE EİŞMA, EDWIN OVERMARS, GERRIT ELSINGA, JERRY WESTERWEEL, TU-Delft — Time-resolved tomographic PIV measurements (acquisition rate 250Hz) were performed in a turbulent boundary layer (TBL) on the side wall of an open channel, water flow facility (cross section 60x60cm, W x H), 3.5m downstream of the inlet at a bulk flow velocity of \( U_b = 0.17m/s \) (\( Re_b = U_b D/\nu = 10^2 \times 10^5 \)). The measurement volume was a horizontal slab \( (6x1.5x0cm^3) \) extending from the side wall, 30cm above the bottom. The Tomo-PIV setup comprised four high-speed ImagerPro HS cameras (2016x2016pixels), a high-speed laser (Nd:YLF, Darwin Duo 80M, Quantronix), optics/prisms and data acquisition/processing software (LaVision, DaVis8.2). A sphere with diameter, \( D = 6mm \) \((D^+ = 51, +^+ \) denotes inner wall scaling\), was positioned at \( y = 37.5 \) and 5.4mm \((y^+ = 319 \) and 46) from the wall (measured from the sphere's center). The latter position covers most of the buffer layer while the former is well in the outer layer. Sphere Reynolds numbers based on \( D \) and the average streamwise velocity at the center of the sphere's center were \( 984 \) \((y^+ = 319) \) and 684 \((y^+ = 46)\). Results show the interaction between the coherent turbulence structures in the TBL and those generated in the sphere’s wake. Total and partial destruction of the log-law layer is observed when the sphere is positioned in the buffer and outer layer, respectively.

8:13AM A19.00002 Effect of viscosity stratification on stability of axisymmetric boundary layer. NIRMAL JAYAPRASAD, VINOD NARAYANAN, IIT Gandhinagar — Stability analysis of heated axisymmetric boundary layers explores the stability characteristics of different types of fluids flowing over a heated cylindrical boundary. In this work, an incompressible, laminar flow over a heated and cooled cylinder, where the flow direction is parallel to the axis of cylinder at steady state is numerically simulated by solving the Navier-Stokes equation and energy equation in cylindrical coordinates. Parallel flow assumption is used to obtain the stability equations. The viscosity variation with temperature is incorporated by using an empirical relation. Since air and water show opposite trend of viscosity variation with temperature, these fluids are considered. The analysis is performed for a range of Reynolds numbers and different wave numbers. The results show that heating stabilizes the flow of water but it has a destabilizing effect on air flow. The effect of Peclet number on the stability characteristics of the flow is also studied. Neutral stability curves of axisymmetric flow of air and water for various temperatures of the cylindrical body are also computed. More detailed results will be presented at the time of conference.

8:26AM A19.00003 Boundary layer similarity flow driven by power-law shear over a nonlinearly stretching surface. DANIEL KUBITSCHEK, PATRICK WEIDMAN, University of Colorado — Similarity solutions of the boundary layer equations describing wall-bounded flow driven by rotational velocities, \( U(y) = y^\alpha \), as \( y \to \infty \), over a nonlinearly stretching surface, \( U(x) = \lambda x^\alpha \) for permissible exponents, \( \sigma = \alpha/(\alpha + 2) \), are presented. An exact solution is presented for \( \alpha = -1/2 \) in terms of Airy functions. Numerical results for the wall shear stress and sample velocity profiles in the range \(-2/3 < \alpha < 5/4\) are computed. The limiting values for \( \lambda < 0 \) are determined, for each value of \( \alpha \), beyond which no solutions are found. The existence of solutions in the range \(-2/3 < \alpha < -1/2 \) is confirmed and the necessary condition, given by M. Guedda (2007), is shown to be satisfied.
8:39 AM A19.00004 The effect of thin liquid films on boundary-layer separation  
RADU CIMPEANU, DEMETRIOS PAPAGEORGIOU, MARINA KRAVTSOVA, ANATOLY RUBAN, Imperial College London — In this study we develop the theory for understanding the process of boundary-layer separation in the presence of a thin liquid film. The investigation is physically motivated by the accumulation of water on aircraft surfaces as a result of flying during adverse weather conditions, with implications in aircraft safety, certification and performance. We present an extension of the asymptotic framework of viscous-inviscid interaction and formulate a modified triple-deck model accounting for the strong density and viscosity contrast between the fluids in the system. The primary goal of the study is to address the question of whether the thin liquid layer acts to suppress or promote boundary-layer separation. We find that an increase in liquid film height (within its asymptotic scaling) contributes to a delay in the onset of separation. Furthermore, the main flow features, represented by local extrema in the perturbed flow quantities, are shifted further downstream within the interaction region. The consequences of the presence of the liquid film are illustrated through two typical examples encountered in flows past aircraft wings, namely surface roughness elements and corners/flare junctions.

8:52 AM A19.00005 The motion induced between radial extensional plates with one or both plates shrinking  
PATRICK WEIDMAN, ENRICO PEROCCO, University of Colorado — Flow between the radial extensional motion of parallel plates is studied when one plate stretches at rate \( a \) while the other shrinks at rate \( b \), and also when both plates shrink. The flow is governed by the stretching ratio \( \sigma = b/a \) and the Reynolds number \( R = \sigma b/h^2/\nu \), where \( h \) is the plate separation distance and \( \nu \) is the fluid kinematic viscosity. When both plates shrink one can find solutions in the region \( \sigma < -1 \) from those found in the region \( -1 \leq \sigma \leq 0 \). This feature is not available when one plate stretches and the other shrinks, and thus \( \sigma \) must be varied over the region \( \sigma \leq 0 \). The \( R = 0 \) solutions and their large-\( \sigma \) asymptotic behaviors are determined. Using two numerical techniques, no bifurcated solutions were encountered. Results are presented for upper and lower wall shear stresses, radial pressure gradients, and radial velocity profiles for these axisymmetric flows. A region of zero wall shear stress exists for stretching/shrinking plates while the wall shear stresses for shrinking/shrinking plates are never zero. An interesting singular limit in solution behavior as \( R \to \infty \) is found for the shrinking/shrinking plate flow.

9:05 AM A19.00006 Investigation of turbulence structure over impermeable and permeable rough walls with identical topography  
TAEOHON KIM, University of Illinois at Urbana Champaign, GIANLUCA BLOIS, University of Notre Dame, JIM BEST, University of Illinois at Urbana Champaign, KENNETH CHRISTENSEN, University of Notre Dame — Turbulent flow over complex topographies, both impermeable and permeable, is encountered in a broad range of natural and engineering systems. Wall permeability gives rise to significant modifications of the underlying flow structure owing to modified wall boundary conditions: slip and penetration. Across this interface, remarkable flow interactions occur and govern significant mass and momentum exchange resulting in apparent modification of the turbulence. In addition, the topography (roughness) of the surface modifies the near-wall flow. The current investigation explores the role of permeability and topography in turbulent flow through the use of sphere-based impermeable (single layer of hemispheres) and permeable (two layers of spheres) walls. Flow across the permeable interface was accessed using a refractive-index matching technique, that permitted high resolution particle-image velocimetry (PIV) measurements to be made in the streamwise-wall-normal \((x \times y)\) plane. This paper will detail analysis of the first and second order velocity statistics associated with these two different cases.

9:18 AM A19.00007 ABSTRACT WITHDRAWN —

9:31 AM A19.00008 Using refractive index matching to image flow above and within a highly-permeable laboratory stream bed  
DEREK LICHTNER, JIM BEST, University of Illinois, GIANLUCA BLOIS, TAEOHON KIM, KENNETH CHRISTENSEN, University of Notre Dame — Turbulent flow over a rough, porous gravel bed is investigated with particle image velocimetry (PIV) and refractive index matching (RIM). A model stream bed was constructed with 4224 pre-cast acrylic spheres \((D = 1.27 \text{ cm})\) in a fixed cubic pattern. The flow above and within the bed was measured in the streamwise-wall-normal plane at \( Re_b = 3.20 \times 10 \) with an image resolution of 11 Mpixel, and the flow was seeded with silver-coated hollow glass spheres \((\rho = 1700 \text{ kg m}^{-3})\). The high-permeability of the interface in these experiments permits large, instantaneous, near-bed streamwise momentum due to vertical exchange via turbulence. The mean velocity flow structure exhibits significant slip velocity at the bed interface. In the pore spaces, mean velocities are near-horizontal and 5-10% of the maximum free stream velocity. High Reynolds stresses near the bed, particularly around the crests of spherical roughness elements, suggest turbulence is produced by flow separation and the shedding of vortices from streambed grains. The dimensions of turbulent flow structures, determined via two-point correlations and Galilean decompositions, appear similar to those of hairpin vortices, although the resemblance remains unconfirmed without time-resolved data.

9:44 AM A19.00009 The law-of-the-wall in mixed convection flow in a vertical channel  
DUNCAN SUTHERLAND, Department of Mechanical Engineering, University of Melbourne Centre for Environmental Safety and Risk Engineering, Victoria University, Melbourne, DANIEL CHUNG, ANDREW OOI, Department of Mechanical Engineering, University of Melbourne, ELIE BOU-ZEID, Civil and Environmental Engineering, Princeton University — Direct numerical simulations (DNS) of mixed convection in a plane vertical channel are conducted over a range of Richardson numbers. The direction of the buoyant forces are parallel and anti parallel to the direction of the imposed mean flow resulting in buoyancy-aided flow on the hot wall and buoyancy opposed flow on the cold wall. In the absence of buoyant forces the mean normal velocity profile is logarithmic and independent of the orientation of the wall. In the case of a horizontal channel, where buoyancy is orthogonal to the direction of mean flow, a correction to the log-law for the mean normal velocity is given by Moinin-Ohbukov similarity theory in terms of empirically determined universal functions of momentum and temperature. We attempt an analysis of the law-of-the-wall for the aiding and opposing flows near the walls in a differentially heated vertical channel and develop analogous universal functions for temperature and momentum from the DNS data.

1Study supported by the Bushfire and Natural Hazards Cooperative Research Centre
8:00AM A20.00001 The Power Law and Log-law Behaviors of the Accelerated Thermal Turbulent Boundary Layer, LUCIANO CASTILLO, Texas Tech University, GUILLERMO ARAYA, Mechanical Engineering Department, University of Puerto Rico - Mayaguez, FAZLE HUSSAIN, Texas Tech University — DNS of spatially evolving thermal turbulent boundary layers with strong favorable pressure gradient (FPG) is performed by employing a multi-scale dynamic approach for generating realistic inflow turbulent information. Results reveal that the thermal fluctuation, $\theta$ and the Reynolds shear stress, $\langle u''v' \rangle$, both exhibit a logarithmic behavior in the meso-layer region (e.g., $30 \leq y' \leq 300$). The thickness of the log-region increases in the flow direction and with the strength of the acceleration. Moreover, the mean temperature profiles do not exhibit a log behavior even in the ZPG region, rather they show a power law. Furthermore, the maxima of the streamwise heat flux, $\langle u' \theta' \rangle$, increases linearly in the FPG region but remains constant in the ZPG region. On the contrary, the wall-normal heat flux remains frozen over the ZPG and acceleration regions. Meanwhile, $\nu'$, $u''$, and $\langle u''v' \rangle$ continue to decay along the flow direction. However, a surprising result is observed in the $\theta$ and $\langle u'' \theta' \rangle$ components which change from constant in ZPG to linear rise as the FPG increases. This increase occurs in spite the fact that turbulence production is drastically reduced in the accelerated region.

8:13AM A20.00002 Connections between density, wall-normal velocity, and coherent structure in a heated turbulent boundary layer, THERESA SAXTON-FOX, California Institute of Technology, STANISLAV GORDEYEV, ADAM SMITH, University of Notre Dame, BEVERLEY MCKEON, California Institute of Technology — Strong density gradients associated with turbulent structure were measured in a mildly heated turbulent boundary layer using an optical sensor (Malley probe). The Malley probe measured index of refraction gradients integrated along the wall-normal direction, which, due to the proportionality of index of refraction and density in air, was equivalently an integral measure of density gradients. The integral output was observed to be dominated by strong, localized density gradients. Conditional averaging and Pearson correlations identified connections between the streamwise gradient of density and the streamwise gradient of wall-normal velocity. The trends were suggestive of a process of pick-up and transport of heat away from the wall. Additionally, by considering the density field as a passive marker of structure, the role of the wall-normal velocity in shaping turbulent structure in a sheared flow was examined. Connections were developed between sharp gradients in the density and flow fields and strong vertical velocity fluctuations.

1This research is made possible by the Department of Defense through the National Defense & Engineering Graduate Fellowship (NDSEG) Program and by the Air Force Office of Scientific Research Grant # FA9550-12-1-0060

8:26AM A20.00003 Influence of wall roughness and thermal conductivity on turbulent natural convection, PAOLO ORLANDI, SERGIO PIROZZOLI, MATTEO BERNARDINI, Universita` di Roma La Sapienza — We study turbulent natural convection in enclosures with conjugate heat transfer. The simplest way to increase the heat transfer in this flow is through rough surfaces. In numerical simulations often constant temperatures are assigned on the walls, but this is an unrealistic condition in laboratory experiments. Therefore, in the DNS, to be of help to experimentalists, it is necessary to solve the heat conduction in the solid walls together with the turbulent flow between the hot and the cold walls. Here the cold wall, 0.5h thick is smooth, and the hot wall has 2D and 3D rough elements of thickness 0.2h above a solid layer 0.3h thick. The simulation is performed in a bi-periodic domain 4h wide. The Rayleigh number varies from $10^6$ to $10^8$. Two values of the thermal conductivity, one corresponding to copper and the other ten times higher were assumed. It has been found that the Nusselt number behaves as $Nu = aRa^\gamma$, with $\gamma$ increasing with the solid conductivity and depending of the roughness shape. 3D elements produce a heat transfer greater than 2D elements. An imprinting of the flow structures on the thermal field inside the walls is observed. The one-dimensional spectra at the center, one decade wide, agree with those of forced isotropic turbulence.

8:39AM A20.00004 How different is Buoyant Turbulence from Isotropic Turbulence?, JULIEN CLARET, ENSTA ParisTech, GUILLAUME BLANQUART, CalTech — This work seeks to establish a new approach for simulating variable density turbulent buoyant flows by extraction of the anisotropic component inherent to buoyant flows. This anisotropy is known to be the main difficulty when simulating buoyant flows. We perform for this an a priori analysis using the DNS data from [Caroll, Blanquart TCFD (2015)]. The anisotropy is quantified first through variance of the velocity field, two-point autocorrelation and energy spectra. The main observation is that for buoyant flows the velocity has a dependency on density shown by a non null conditional mean whereas this is not observed in isotropic flows. This allows to decompose Homogeneous Buoyant Turbulence (HBT) into three terms. The first corresponds to the conditional mean velocity on density that contains but is not reduced to the small scales anisotropy. The second term corresponds to the mean velocity averaged only in the direction of gravity which contains large scale anisotropy. The final term corresponds to a field of Homogeneous Isotropic Turbulence (HIT). This decomposition allows the reduction of the problem to the study of an HIT field which is well known. It also sheds light onto the development of a new Sub-Grid Scale (SGS) to simulate flows driven by buoyancy.

8:52AM A20.00005 Skin friction field and thermal plume formation in turbulent convection, JOERG SCHUMACHER, VINODH BANDARU, ANASTASIYA KOLCHINSKAYA, TU Ilmenau, JANET SCHEEL, Occidental College Los Angeles, KATHRIN PADBERG-GEHLE, TU Dresden — The dynamics in the thin boundary layers of temperature and velocity is the key to a deeper understanding of turbulent transport of heat and momentum in thermal convection. The velocity gradient right at the heated plate of a Rayleigh-Bénard convection cell forms the two-dimensional skin friction field and is related to the formation of thermal plumes in the boundary layer right above the plate. Our analysis is based on a direct numerical simulation of Rayleigh-Bénard convection in a closed cylindrical cell of aspect ratio $\Gamma = 1$ and focused on the critical points of the skin friction field. We identify triplets of critical points, which are composed of two unstable nodes and a saddle between them, as the characteristic building block of the skin friction field. Isolated triplets $Z_{++}$ and triplets $Z_{-+}$ coincide with thermal plumes, while triplets $Z_{++}$ reside in the meso-layer region. We focus on the separation of the triplets of critical points, which is the signature of the transition from isolated triplets to networks of triplets. These triplets are then compared with the triplets observed in the thermal field of the skin friction field. From a dynamical Lagrangian perspective, thermal plumes are formed together with an attractive hyperbolic Lagrangian Coherent Structure of the skin friction field. We discuss the differences from the skin friction field in turbulent channel flows from the perspective of the Poincaré-Hopf index theorem for two-dimensional vector field.

1This work is supported by the Deutsche Forschungsgemeinschaft.

9:05AM A20.00006 The effect of turbulent fluctuations on the relaxation of thermal non-equilibrium, SUALEKH KHURSHID, DIEGO DONZIS, Texas A&M University — In many engineering and natural systems, the microscopic behavior of constituent molecules can affect the macroscopic behavior of the flow. This interaction is significant when the two phenomena have commensurate time scales. We study the effect of turbulence on the relaxation of thermal non-equilibrium (TNE), in particular vibrational energy relaxation, using direct numerical simulation (DNS). First order effects are observed in the evolution of both vibrational energy and turbulence. For example, the rate of decay of kinetic energy is accelerated and temperature fluctuations are amplified. Analytic expressions for equilibrium vibrational energy, $E_v^*$, and characteristic relaxation time scale, $\tau_v$, are compared against DNS data and used to understand features of the decay. This decay can be divided into two regimes, one dominated by TNE exchanges in time scales of the order of $\tau_v$ followed by a turbulence decay. Between the two regimes, some vibrationally hot flows become cold before reaching equilibrium. This reflects an aspect of the strong coupling between turbulence and TNE in both regimes. Compressibility effects, quantified by turbulent Mach number ($M_t$), are also discussed.
9:18AM A20.00007 Construction of a Non-Equilibrium Thermal Boundary Layer Facility . DRUMMOND BILES, ALIREZA EBADI, ALLEN MA, CHRISTOPHER WHITE, Univ of New Hampshire — A thermally conductive, electrically heated wall-plate forming the bottom wall of a wind tunnel has been constructed and validation tests have been performed. The wall-plate is a sectioned wall design, where each section is independently heated and controlled. Each section consists of an aluminum 6061 plate, an array of resistive heaters affixed to the bottom of the aluminum plate, and a calcium silicate holder used for thermal isolation. Embedded thermocouples in the aluminum plates are used to monitor the wall temperature and for feedback control of wall heating. The wall-plate is used to investigate thermal transport in both equilibrium and non-equilibrium boundary layers. The non-equilibrium boundary layer flow investigated is oscillatory flow produced by a rotor-stator mechanism placed downstream of the test section of the wind tunnel.

9:31AM A20.00008 Scaling of co-spectra in grid turbulence with a mean cross-stream temperature gradient . CARLA BAHR, GILAD ARWATZ, MARCUS HULTMARK, MICHAEL MUELLER, Princeton University — Scaling of grid turbulence with a mean cross-stream temperature gradient is investigated using a combination of theoretical predictions and DNS. Conditions for self-similarity of the governing equations and particularly the scalar co-spectrum are investigated, which reveals necessary conditions for self-similarity to exist. These conditions provide a theoretical framework for scaling of the temperature flux spectrum, which offers new insights into the interaction of the turbulent velocity field with the scalar field. One necessary condition, predicted by the theory, is that the co-spectrum must vary as $\propto s^2$ for a self-similar solution to exist. DNS results are used to validate the theoretical predictions and good collapse of the co-spectrum is observed, which validates the self-similarity theory.

9:44AM A20.00009 ABSTRACT WITHDRAWN –

Sunday, November 22, 2015 8:00AM - 9:57AM – Session A21 Turbulence: Environmental Flows 209 - Lian-Ping Wang, University of Delaware

8:00AM A21.00001 Simulations of turbulence and dispersion in idealized urban canopies using a new kinetic scheme . LIAN-PING WANG, PABLO HUQ, University of Delaware, USA, ZHAOLI GUO, Huazhong University of Science and Technology, China — In this talk, we will demonstrate the capabilities of a new kinetic scheme, known as the Discrete Unified Gas Kinetic Scheme (DUGKS), by simulating turbulent flow and scalar dispersion in an idealized urban canopy. DUGKS is a finite-volume formulation of the Boltzmann equation which can incorporate a non-uniform grid. It could be used as an direct numerical simulation tool or as a large-eddy simulation tool for turbulent flow in a complex geometry. We will describe this mesoscopic CFD method, and details in setting up a non-uniform grid, no-slip boundary condition on solid surfaces, and far-field boundary conditions. The model urban canopy contains an array of buildings with a prescribed building-height-to-width aspect ratio. Several aspect ratios will be considered in the simulations. A passive scalar is continuously released from a near-ground point source. Profiles of mean velocity, turbulence statistics, and scalar concentration obtained from the simulations will be compared to data from water-tunnel measurements. Grid refinement will be performed to study the convergence of the simulated results on the grid resolution.

8:13AM A21.00002 Flow over and within large-scale porous topography: Impact of surface heterogeneity on turbulence structure . ALI M. HAMED, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, PRATEEK RANJAN, MATTHEW J. SADOWSKI, University of Illinois at Urbana-Champaign, HEIDI M. NEPF, Department of Civil and Environmental Engineering, Massachusetts Institute of Technology, LEONARDO P. CHAMORRO, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — An experimental investigation of the flow within and above model canopies was carried out to determine the effect of canopy height heterogeneity on the structure and spatial distribution of the turbulence. Two 800 mm long models with 20% blockage were placed in a 2.5 m long refractive-index-matching channel. The first model (base case) is constituted of equal height (h) square bar elements arranged in a staggered configuration. The other model bars had two heights (h+1/3h and h-1/3h) alternated every two rows. Particle image velocimetry was used to map the flow field at three locations spanning the length of the canopy under three confinement ratios H/h=2, 3, and 4, where H is the free surface height. The experiments were performed at Reynolds number $Re_H = 6800$, 10200, and 13600. Refractive index matching renders the canopy invisible and grants full optical access allowing the flow field within the canopy to be measured by PIV. Turbulence statistics complemented with POD, quadrant analysis, and LES decomposition reveal the distinctive effect of the height heterogeneity on the shear layer that forms on top of the canopy, and on the free flow over the canopies.

8:26AM A21.00003 A large-eddy simulation study on statistical attributes of urban-like geometries relevant to turbulence structure and bulk aerodynamic characteristics. XIAOWEI ZHU, WILLIAM ANDERSON, UT Dallas — The inherent spatial heterogeneity exhibited by real urban environments complicates a priori estimation of the roughness height needed to parameterize the inertial layer mean streamwise velocity. A large-eddy simulation study of turbulent flow over 3-D random urban-like topographies is conducted to explore the effects of surface geometry on bulk aerodynamic characterization. In a mean sense, we find that statistical attributes including surface height root mean square and skewness can adequately capture the spatial heterogeneities and randomness of real urban geometries. We find, however, that higher-order statistical moments have a negligible affect on aerodynamic drag (i.e. kurtosis may be omitted). The results enable exploration of applicability of some recently-proposed roughness parameterizations that are relevant to complex, urban-like roughness (including the model proposed by Flack and Schultz, 2010: J. Fluids Eng. 132, 041203-1). We evaluate empirical parameters needed in these models for the present urban-like cases. We find that two empirical parameters (relevant to height rms and skewness) can characterize the bulk aerodynamic roughness of topographies with statistical attributes comparable to dense urban environments.

1This work was supported by the Army Research Office, Atmospheric Sciences Program (PM: Dr. S. Collier) under Grant # W911NF-13-1-0474. Computational resources were provided by the Texas Advanced Computing Center at the University of Texas.
8:39AM A21.00004 LES for wind turbulence in canopy layer at large urban area. TETSURO TAMURA, HIDENORI KAWAI, Tokyo Institute of Technology, RAHUL BALE, KEIJI ONISHI, MAKOTO TSUBOKURA, Riken, KOJI KONDO, Kajima Corporation, TSUYOSHI NOZU, Shimizu Corporation — In order to accurately predict the wind flow in canopy layer of large urban area, we introduce LES based on BCM, Building Cube Method which is formulated on the very fine Cartesian mesh system. Houses and buildings were not aerodynamically modelled but directly reproduced their shapes in the numerical model, because the wind profile particularly in cities requires the correct estimation of local flow field in the canopy layer close to the ground. Recent high-performance computing, HPC technique has developed distinctly, so high-resolution computation can apply to flows around a complicated configuration. In this case we have to deal with buildings, vegetation and street etc. as a part of numerical model. Actually LES using the Cartesian coordinate encounters the non-correspondence of directions between the street lines and the discretized mesh lines. Very fine mesh system by BCM can solve this problem, supported by Immersed Boundary Method. Also, in this numerical scheme, the computational process is so simple that the parallel algorithm and the memory access obtain the perfect efficiency. In this study, we have applied LES by BCM to the wind flow estimation over the real complicated urban surface.

8:52AM A21.00005 Entrainment Across a Sheared Density Interface in High Richardson Number Cavity Flow. NICHOLAS WILLIAMSON, MICHAEL KIRKPARTICK, STEVE ARMFIELD, The University of Sydney — The turbulent entrainment of fluid across a sharp density interface has been examined experimentally in a purging cavity flow. In the experiments, a long straight cavity with sloped entry and exit boundaries is located in the base of a straight open channel. Saline fluid is entrained from the cavity into the overflow. The cavity geometry has been designed to ensure there is no separation of the overflow in the cavity region with the goal of obtaining a single mode of entrainment, related only to the interface properties rather than to cavity specific mechanisms. The bulk entrainment rate has been measured and correlated with bulk Richardson number over Ri = 1.0 – 20 at Reynolds number Re = 7100 – 15100. The entrainment rate is shown to scale with the local bulk Richardson number E ≃ CRi−1.38, very close to the established result for entrainment across a sharp two layer density interface in a recirculating water channel (Strang and Fernando, J Fluid Mech., 428, 2001) but with an order of magnitude lower coefficient C. Experiments instrumented with PIV/LIF were used to relate the bulk Ri to the local gradient Richardson number of the interface. In the cavity setting the interface appears to remain sharper, resulting in larger Ri and reduced entrainment.

9:05AM A21.00006 Entrainment of Vertical Jets in Turbulent Cross Flow. GRAHAM FREEDLAND, Portland State Univ, KAREN ROBERTS, Washington State Univ, LARRY MASTIN, U.S. Geological Survey, STEPHEN SOLOVITZ, Washington State Univ, RAUL CAL, Portland State Univ — Volcanic eruptions produce high concentrations of ash that produce clouds in the atmosphere that are hazardous for private and commercial aviation. Without accurate models to predict ash concentrations, air traffic is unable to safely navigate ash clouds downwind of an eruption. The present study focuses on the adjustment of turbulence and flow characteristics downstream of a model canopy. These experiments comprise three different canopy heights, three different wave conditions, and three different flowrates. Measurements are taken using an acoustic velocimeter and 2D particle image velocimetry. This work proposes the existence of four distinct regions downstream of a model canopy: the mixing layer, the transition region, the turbulence decay region, and the boundary layer. Each of these regions has distinct characteristics regarding the mean flow, bed stress, TKE, and Reynolds shear stress. The delineation and description of these four regions will allow ecosystem managers and sediment modelers to better understand coastal dynamics.

9:18AM A21.00007 Development of Turbulence Downstream of a Submerged Aquatic Canopy in Unidirectional and Combined Wave-Current Flows. ROBERT ZELLER, JEFFREY KOSEFF, The Bob and Norma Street Environmental Fluid Mechanics Laboratory, Stanford University — Seagrasses and corals form the essential building blocks of many coastal ecosystems, and the turbulence generated from these canopies has been investigated heavily. However, the effect of these canopies on the downstream flow is poorly understood, particularly for combined wave-current flows. Furthermore, the development of flow characteristics may have a profound impact on propagule transport and sediment dynamics downstream of the canopy. The present study focuses on the adjustment of turbulence and flow characteristics downstream of a model canopy. These experiments comprise three different canopy heights, three different wave conditions, and three different flowrates. Measurements are taken using an acoustic velocimeter and 2D particle image velocimetry. This work proposes the existence of four distinct regions downstream of a model canopy: the mixing layer, the transition region, the turbulence decay region, and the boundary layer. Each of these regions has distinct characteristics regarding the mean flow, bed stress, TKE, and Reynolds shear stress. The delineation and description of these four regions will allow ecosystem managers and sediment modelers to better understand coastal dynamics.

9:31AM A21.00008 Dynamic Mode Decomposition of Flow Around Interacting Barchan Dunes. NATHANIEL BRISTOW, GIANLUCA BLOIS, TAEOHON KIM, Department of Aerospace and Mechanical Engineering, University of Notre Dame, KOREYA RAKUKO, Department of Aerospace and Mechanical Engineering, University of Notre Dame, SADEK AMR, Department of Mathematics, Imperial College, London, England, JIM HOLLIS, Department of Geology, Geography, and Environmental Sciences, University of Illinois, Chicago, Illinois, U.S., KENNETH CHRISTENSEN, Department of Aerospace and Mechanical Engineering, University of Notre Dame, Notadome, Indiana, U.S. — Barchan dunes are crescentic bedforms located in environments with unidirectional flow and limited sediment supply, including deserts, river beds and the craters of Mars. The evolution of, and interactions between, barchans are highly dynamic, involving feedback mechanisms between the fluid flow, morphological change and sediment transport. A series of experiments were undertaken to discretely simulate the collapse of a smaller barchan with a larger, downstream one using fixed bedform models, each experiment representing a successive snapshot in the dune collision process. These experiments thus capture the turbulent flow over fixed-bed morphologies that correlate with rapid morphological change and high rates of sediment transport using time-resolved PIV in the wall-parallel plane. The use of a Refractive Index Matching (RIM) flow facility allows for the light to pass through the model, capturing areas which are otherwise obscured, such as around the horns of the dune and the sheltered region behind the crest. Dynamic Mode Decomposition is used to identify the most dominant modes contributing to flow dynamics in each collision stage.

1 NSF DGE-114747
2 Now at The Aerospace Corporation
Drifting snow has a significant impact on snow distribution in mountains, prairies as well as on glaciers and polar regions. In all these environments, the local mass balance is highly influenced by drifting snow. Despite most of the model approaches still rely on the assumption of steady-state and equilibrium saltation, recent advances have proven the mass-transport of drifting snow events to be highly intermittent. A clear understanding of such high intermittency has not yet been achieved. Therefore in our contribution we investigate mass- and momentum fluxes during drifting snow events, in order to better understand that the link between snow cover erosion and deposition. Experiments were conducted in a cold wind tunnel, employing sensors for the momentum flux measurements, the mass flux measurement and for the snow depth estimation over a certain area upstream of the other devices. Preliminary results show that the mass flux is highly intermittent at scales ranging from eddy turnover time to much larger scales. The former scales are those that contribute the most to the overall intermittency and we observe a link between the turbulent flow structures and the mass flux of drifting snow at those scales. The role of varying snow properties in inducing drifting snow intermittency goes beyond such link and is expected to occur at much larger scales, caused by the physical snow properties such as density and cohesiveness.

The interaction between the small and large scales of turbulence is investigated in a mixing layer achieving a Reynolds number based on the Taylor microscale ($\text{Re}_\lambda = 250$). Positive fluctuations of the large-scale velocity correspond to large vorticity r.m.s. on the low-speed side of the mixing layer and to low vorticity r.m.s. on the high-speed side, respectively. The relationship between large and small scales thus depends on the position if the vorticity r.m.s. is correlated with the large-scale velocity fluctuations. However, when correlating the vorticity r.m.s. with the large-scale velocity gradients, the correlation coefficient is nearly constant throughout the mixing layer and close to unity. This observation reveals that large and small scales are characterized by a strong interaction independent of the flow position when the large-scale velocity gradients are considered instead of the large-scale velocity fluctuations usually employed in the existing literature on amplitude modulation. The vorticity from unfiltered (small scales) and from low-pass filtered velocity fields tend to be aligned when examined within vortical tubes, suggesting that part of the large-scale characteristics is not lost at the smallest scales.

Turbulent mixing for jets in crossflow is important in numerous applications. Reynolds-averaged models for turbulent scalar transport are usually based on the gradient diffusion hypothesis (GDH), with a scalar eddy diffusivity calculated from the model eddy viscosity. Such models are not accurate in the near jet region causing poor prediction of the scalar concentration distribution. We use 3D mean velocity and concentration data acquired using magnetic resonance imaging to infer improved diffusivity models. The transport equation is solved using the experimental velocity data and a prescribed functional form for the scalar diffusivity. An evolutionary algorithm then optimizes the model constants to minimize the difference between the calculated and measured scalar concentration fields. Tests of multiple model forms for seven different jet in crossflow configurations provide insight into the required characteristics of advanced models. The GDH with a weakly anisotropic diffusivity is very accurate beyond 4 hole diameters downstream of the injection point. However, standard turbulent diffusivity models overestimate turbulent mixing in the separation region; in most cases, the optimization procedure inferred counter-gradient diffusion in this region. New models that adjust automatically depending on the characteristics of the mean velocity and concentration fields are under development.

This research is supported by Honeywell Inc.
the momentum balance for the suspension – the mixture of active particles plus fluid – only external particles may also display ‘action at a distance’ and accumulate adjacent to (or be depleted from) a boundary without any external forces. In the run length of the active particles and gives a Boltzmann-like distribution from a balance of the swim force and the swim pressure. Active and the ‘weight’ of the active particles. A continuum mechanical description is possible when variations occur on scales larger than

observed in experiments. Our analysis implies that active liquid crystals are governed by the same generic ordering principles that determine with recently published data and, in particular, predicts correctly a previously unexplained regime of long-range nematic ordering of defects interest, yet a satisfactory mathematical description remains elusive. Here, we present and validate a continuum theory for this new class of self-assemble into two-dimensional active liquid crystals that exhibit a rich creation and annihilation dynamics of topological defects, reminiscent to understanding the physics of microbiological systems. Recent experiments demonstrated that ATP-driven microtubule-kinesin bundles can

As a result, the particle-pressure exerted on a container wall is the sum of the swim pressure [Takatori et al., Phys. Rev. Lett. 2014, 113, 028103] and the ‘weight’ of the active particles. A continuum mechanical description is possible when variations occur on scales larger than the run length of the active particles and gives a Boltzmann-like distribution from a balance of the swim force and the swim pressure. Active particles may also display ‘action at a distance’ and accumulate adjacent to (or be depleted from) a boundary without any external forces. In the momentum balance for the suspension – the mixture of active particles plus fluid – only external body forces appear.
8:26AM A23.00003 Diffusion in active suspension of microswimmers. ERIC CLIMENT, BLAISE DELMOTTE, FRANCK PLOURABOUE, Institut de Mecanique des Fluides de Toulouse - France, ERIC KEAVENY, Imperial College - Dept. of Math. - London - UK, MATTHIEU MARTIN, SALIMA RAFAI, PHILIPPE PEYLA, ERIC BERTIN, Laboratoire Interdisciplinaire de Physique - Grenoble, France, IMFT TEAM, IC TEAM, LIPHY TEAM — The presence of microswimmers in a fluid generates flow agitation due to multi-body hydrodynamic interactions. This agitation of the fluid leads to random trajectories of passive tracers particles and the swimmers themselves, and from a microscopic point view, it can be interpreted as a diffusive mechanism. By means of experiments (videomicroscopy of suspensions of chlamydomonas reinhardtii) and numerical simulations (Stokesian fluid populated with squirmers), we investigate the evolution of the effective diffusion coefficient when the volumetric concentration of the active suspension varies. By comparing the experimental and numerical results, we quantify the role of active swimming on the measured diffusion and identify the physical mechanisms that lead to diffusion enhancement. Our results aim to provide a better understanding of how swimming organisms affect micron-scale transport in the environment.

8:39AM A23.00004 Efficient Simulation of a Large Number of Microswimmers Using Fast Multipole Method. MINGHAO ROSTAMI, SARAH OLSON, Worcester Polytechnic Institute — Regularized Stokes formulation has been shown to be very effective at modeling fluid-structure interactions when the fluid is highly viscous. However, its computational cost grows quadratically with the number of particles immersed in the fluid. We demonstrate how fast multipole method can be applied to significantly reduce the computational cost of regularized Stokes method. Numerical results will be presented for simulating the dynamics of a large number of microswimmers immersed in 3D Stokes flows. Furthermore, we also investigate the swimming efficiency of the microswimmers when they are placed in various geometric configurations.

8:52AM A23.00005 Biogenic mixing induced by intermediate Reynolds number swimming at pycnoclines1. SHIYAN WANG, University of Notre Dame and Purdue University, AREZOO ARDEKANI, Purdue University — Recently, there has been a debate regarding the contribution of marine organisms to ocean mixing. To address this question, we study fully-resolved motion of interacting swimmers in a density stratified fluids using a “squirmer” model to quantify their contribution to mixing. In the aphotic ocean (i.e. regions that are 200 m beneath the sea surface), zooplankton are the most abundant organisms leading to vertical fluid transport. Their body size ranges from millimeter to centimeter, and their Reynolds number is in the range of O(1-100). Therefore, it is important to examine the biogenic mixing in this inertial regime. Our numerical results suggest that biogenic mixing increases with inertia, and in local hot spots, the vertical water transport induced by centimeter-sized organisms is comparable to the turbulent mixing. In the presence of background turbulence, the biogenic mixing is determined by the magnitude of dissipation of kinetic energy introduced by the organisms.

This work is supported by NSF grants CBET-1066545 and CBET-1414581.

9:05AM A23.00006 Collective motion of microswimmers in viscoelastic fluids1, GAOJIN LI, AREZOO ARDEKANI, Purdue University — The dynamics of suspension of self-propelled microorganisms show fascinating hydrodynamic phenomena, such as, large scale swarming motion, locally correlated motion, enhanced particle diffusion, and enhanced fluid mixing. Even though many studies have been conducted in a Newtonian fluid, the collective motion of microorganisms in non-Newtonian fluids is less understood. The non-Newtonian fluid rheological properties, such as viscoelasticity and shear-dependent viscosity in saliva, mucus and biofilm, significantly affect the swimming properties and hydrodynamic interaction of microorganisms. In this work, we use direct numerical simulation to investigate the collective motion of rod-like swimmers in viscoelastic fluids. Two swimming types, pusher and puller, are investigated. The background viscoelastic fluid is modeled using an Oldroyd-B constitutive equation.

This work is supported by NSF CBET-1445955 and Indiana CTSI TR001108.

9:18AM A23.00007 Numerical study of the generation of metachronal waves in layers of beating cilia using a Lattice Boltzmann method. Application to the generation of fluid motion at the cell scale. JEAN MERCAT, ZHE LI, JULIEN FAVIER, UMBERTO D’ORTONA, Aix Marseille University, M2P2, UMR7340, France, SEBASTIEN PONCET, Sherbrooke University, Quebec, Canada, M2P2 TEAM — Cilia are flexible elongated whip-like structures which are ubiquitous in nature. Indeed, the collective beating of arrays of thousands of cilia can transport fluid (mucus in airways) or induce locomotion on microorganisms swimming in water. From a purely hydrodynamical point of view, cilia do not beat randomly, but rather generate typical metachronal waves at their surface. In this work, we study the self-organization of the beating motion of large fields of beating cilia in a two-component flow environment, made of water and a much more viscous fluid. The numerical solver is based on an immersed boundary-lattice Boltzmann method in the context of single- and multi-component fluid flows, and in the presence of fixed or moving solid boundaries. The solver has been validated in previous studies. Various parameters are varied, such as length, spacing and phase motion of individual cilia. The energetic performances of different kind of waves are studied to understand the emergence of antiplectic metachronal waves, commonly observed in nature. It is found that a purely hydrodynamical coupling between fluid and cilia can explain the onset of metachronal waves in cilia arrays, and that these waves are maximizing a performance ratio.

9:31AM A23.00008 A fast method to compute triply-periodic Brinkman flows1, HOANG-NGAN NGUYEN, KARIN LEIDERMANN, Univ of California, Merced, SARAH OLSON, Worcester Polytechnic Institute — A fast method is developed to efficiently compute three-dimensional Brinkman flows induced by triply-periodic arrays of points forces and regularized forces. For point forces, we decompose the periodic Brinkman velocity into the sum of two series: one in real space and one in Fourier space. To do the splitting, we make use of a regularized solution with special decay properties so that both summands will decay in a Gaussian manner. For regularized forces, the same methodology is used to split the regularized velocity, and again, Gaussian decay of the summands is achieved. When there are N forces (N periodic arrays), the overall complexity is O(N^2). We discuss different ways to reduce the complexity to O(N^3/2) and to O(N log N). Finally, we present two sets of numerical results. The first validates the computational complexity of the algorithm and the second illustrates how this method can be used to study microscopic flows of organisms in a porous medium. A simple dumbbell model of swimmers is implemented that exhibits a large scale flow varying as a function of resistance within the porous medium.

This work is supported by NSF Grant number 1413078.
explosions

1 Experimental results with Brownian dynamics simulations and analytical theory. We corroborate all systems originating from the force required to confine them by boundaries. We apply a strong trap to collect the swimmers into a close-packed active crystal and then turn off the trap which causes the crystal to “explode” due to an imbalance of the active pressure. We understand the evolution of the intestine as an ideal mixer.

8:00AM A24.00001 Transcapillary Trafficking of Clustered Circulating Tumor Cells

8:13AM A24.00002 Transversal mixing in the gastrointestinal tract

8:26AM A24.00003 Fluid dynamic modelling of renal pelvic pressure during endoscopic stone removal

8:39AM A24.00004 The intestine is a blender

8:52AM A24.00005 Fluid-solid modeling of lymphatic valves

1ST is supported by a Gates Millennium Scholars fellowship and a NSF Fellowship No. DGE-1144469. RDD is supported by a doctoral fellowship of the fund for scientific research (FWO-Vlaanderen). This work is also supported by NSF Grant CBET 1437570.
A reduced order model for fluid-structure interaction of thin shell structures conveying fluid for physiological applications, GARY HAN CHANG, YAHYA MODARRES-SADEGHI, Univ of Mass - Amherst — In this work, a reduced-order model (ROM) is constructed to study fluid-structure interaction of thin shell structures conveying fluid. The method of snapshot Proper Orthogonal Decomposition (POD) is used to construct the reduced-order bases based on a series of CFD results, which then are improved using a QR-factorization technique to satisfy the various boundary conditions in physiological flow problems. In the process, two sets of POD modes are extracted: those due to the shell wall’s motion and those due to the pulsatile flow. The Modal Assurance Criterion (MAC) technique is used for selecting the final POD modes used in the reduced-order model. The structure model is solved by Galerkin’s method and the FSI coupling is done by adapting a coupled momentum method. The results show that the dynamic behavior of thin shells conveying fluid is closely related to the distribution of the shell’s Gaussian curvature, the existence of imperfections and the physiological flow conditions. This method can effectively construct a computationally efficient FSI model, which allows us to examine a wide range of parameters which exist in real-life physiological problems.

A reduced order model for fluid-structure interaction of thin shell structures conveying fluid for physiological applications, GARY HAN CHANG, YAHYA MODARRES-SADEGHI, Univ of Mass - Amherst — In this work, a reduced-order model (ROM) is constructed to study fluid-structure interaction of thin shell structures conveying fluid. The method of snapshot Proper Orthogonal Decomposition (POD) is used to construct the reduced-order bases based on a series of CFD results, which then are improved using a QR-factorization technique to satisfy the various boundary conditions in physiological flow problems. In the process, two sets of POD modes are extracted: those due to the shell wall’s motion and those due to the pulsatile flow. The Modal Assurance Criterion (MAC) technique is used for selecting the final POD modes used in the reduced-order model. The structure model is solved by Galerkin’s method and the FSI coupling is done by adapting a coupled momentum method. The results show that the dynamic behavior of thin shells conveying fluid is closely related to the distribution of the shell’s Gaussian curvature, the existence of imperfections and the physiological flow conditions. This method can effectively construct a computationally efficient FSI model, which allows us to examine a wide range of parameters which exist in real-life physiological problems.

A reduced order model for fluid-structure interaction of thin shell structures conveying fluid for physiological applications, GARY HAN CHANG, YAHYA MODARRES-SADEGHI, Univ of Mass - Amherst — In this work, a reduced-order model (ROM) is constructed to study fluid-structure interaction of thin shell structures conveying fluid. The method of snapshot Proper Orthogonal Decomposition (POD) is used to construct the reduced-order bases based on a series of CFD results, which then are improved using a QR-factorization technique to satisfy the various boundary conditions in physiological flow problems. In the process, two sets of POD modes are extracted: those due to the shell wall’s motion and those due to the pulsatile flow. The Modal Assurance Criterion (MAC) technique is used for selecting the final POD modes used in the reduced-order model. The structure model is solved by Galerkin’s method and the FSI coupling is done by adapting a coupled momentum method. The results show that the dynamic behavior of thin shells conveying fluid is closely related to the distribution of the shell’s Gaussian curvature, the existence of imperfections and the physiological flow conditions. This method can effectively construct a computationally efficient FSI model, which allows us to examine a wide range of parameters which exist in real-life physiological problems.

A reduced order model for fluid-structure interaction of thin shell structures conveying fluid for physiological applications, GARY HAN CHANG, YAHYA MODARRES-SADEGHI, Univ of Mass - Amherst — In this work, a reduced-order model (ROM) is constructed to study fluid-structure interaction of thin shell structures conveying fluid. The method of snapshot Proper Orthogonal Decomposition (POD) is used to construct the reduced-order bases based on a series of CFD results, which then are improved using a QR-factorization technique to satisfy the various boundary conditions in physiological flow problems. In the process, two sets of POD modes are extracted: those due to the shell wall’s motion and those due to the pulsatile flow. The Modal Assurance Criterion (MAC) technique is used for selecting the final POD modes used in the reduced-order model. The structure model is solved by Galerkin’s method and the FSI coupling is done by adapting a coupled momentum method. The results show that the dynamic behavior of thin shells conveying fluid is closely related to the distribution of the shell’s Gaussian curvature, the existence of imperfections and the physiological flow conditions. This method can effectively construct a computationally efficient FSI model, which allows us to examine a wide range of parameters which exist in real-life physiological problems.

Scaling of phloem structure and optimality of sugar transport in conifer needles, KAARE H. JENSEN, Technical University of Denmark, HENRIK RONELLENFITSCH, Max Planck Institute for Dynamics and Self-Organisation, JOHANNES LIESCHE, University of Copenhagen, N. MICHELE HOLBROOK, Harvard University, ALEXANDER SCHULZ, University of Copenhagen, ELENI KATIFORI, University of Pennsylvania — The phloem vascular system facilitates transport of energy-rich sugar and signalling molecules in plants, thus permitting long-range communication within the organism and growth of non-photosynthesizing organs such as roots and fruits. The flow is driven by osmotic pressure, generated by differences in sugar concentration between distal parts of the plant. The phloem is an intricate distribution system, and many questions about its regulation and structural diversity remain unanswered. Here, we investigate the phloem structure in the simplest possible geometry: a linear leaf, found, for example, in the needles of conifer trees. We measure the phloem structure in four tree species representing a diverse set of habitats and needle sizes, from 1 cm (Picea omorika) to 35 cm (Pinus palustris). We show that the phloem shares common traits across these four species and find that the size of its conductive elements obeys a power law. We present a minimal model that accounts for these common traits and takes into account the transport strategy and natural constraints. This minimal model predicts a power law phloem distribution consistent with transport energy minimization, suggesting that energetics are more important than translocation speed at the leaf level.
8:13AM A25.00002 Biophysical analysis of water filtration phenomenon in the roots of halophytes1, KIWOONG KIM, SANG JOON LEE, Center for Biofluid and Biomimic Research, Department of Mechanical Engineering, Pohang University of Science and Technology (POSTECH) — The aim of this work is to understand the physics underlying the mechanisms of two-dimensional aquatic pollen dispersal, known as hydrophylls, that have evolved in several genera of aquatic plants, including Halodule, Halophila, Lepilena, and Ruppia. We selected Ruppia, which grows in the wetlands of the New Jersey/New York metropolitan area, for this study. We observed two mechanisms by which the pollen released from male inflorescences of Ruppia maritime is adsorbed on a water surface: 1) inflorescences rise above the water surface and after they mature their pollen mass falls onto the surface as clumps and disperses as it comes in contact with the surface; 2) inflorescences remain below the surface and produce air bubbles which carry pollen mass to the surface where it disperses. In both cases dispersed pollen masses combine with others to form pollen rafts. The formation of porous pollen rafts increases the probability of pollination since the attractive capillary force on a pollen raft towards a stigma is much larger than on a single pollen grain.

1This research was financially supported by the National Research Foundation (NRF) of Korea (Contract grant number: 2008-0061991).

8:26AM A25.00003 Fluid dynamics of two-dimensional pollination in Ruppia (widggon grass)1, NAGA MUSUNURI, DANIEL BUNKER, SUSAN PELL, IAN FISCHER, PUSHPENDRA SINGH, NJIT — The aim of this work is to understand the physics underlying the mechanisms of two-dimensional aquatic pollen dispersal, known as hydrophylls, that have evolved in several genera of aquatic plants, including Halodule, Halophila, Lepilena, and Ruppia. We selected Ruppia, which grows in the wetlands of the New Jersey/New York metropolitan area, for this study. We observed two mechanisms by which the pollen released from male inflorescences of Ruppia maritime is adsorbed on a water surface: 1) inflorescences rise above the water surface and after they mature their pollen mass falls onto the surface as clumps and disperses as it comes in contact with the surface; 2) inflorescences remain below the surface and produce air bubbles which carry pollen mass to the surface where it disperses. In both cases dispersed pollen masses combine with others to form pollen rafts. The formation of porous pollen rafts increases the probability of pollination since the attractive capillary force on a pollen raft towards a stigma is much larger than on a single pollen grain.

1The work was supported by National Science Foundation

8:39AM A25.00004 ABSTRACT WITHDRAWN —

8:52AM A25.00005 Scientific designs of pine seeds and pine cones for species conservation1, KAHYE SONG, EUNSEOP YEOM, HYEJEONG KIM, SANG JOON LEE, Pohang Univ of Sci & Tech — Reproduction and propagation of species are the most important missions of every living organism. For effective species propagation, pine cones fold their scales under wet condition to prevent seeds from short-distance dispersal. They open and release their embedded seeds on dry and windy days. In this study, the micro-/macro-scale structural characteristics of pine cones and pine seeds are studied using various imaging modalities. Since the scales of pine cones consist of dead cells, the folding motion is deeply related to structural changes. The scales of pine cones consist of three layers. Among them, bract scales are only involved in collecting water. This makes pine cones reduce the amount of water and minimize the time spent on structural changes. These systems also involve in drying and recovery of pine cones. In addition, pine cones and pine seeds have advantageous structures for long-distance dispersal and response to natural disaster. Owing to these structural features, pine seeds can be released safely and efficiently, and these types of structural advantages could be mimicked for practical applications.

1This research was financially supported by the Creative Research Initiative of the Ministry of Science, ICT and Future Planning (MSIP) and the National Research Foundation (NRF) of Korea (Contract grant number: 2008-0061991).

9:05AM A25.00006 Switchable and Tunable Aerodynamic Drag on Cylinders , MARK GUTTAG, FRANCISCO LOPEZ JIMENEZ, PEDRO REIS, MIT — We report results on the performance of Smart Morphable Surfaces (Smporphs) that can be mounted onto cylindrical structures to actively reduce their aerodynamic drag. Our system comprises of an elastomeric thin shell with a series of carefully designed sub-surface cavities that, once depressurized, lead to a dynamic deformation of the surface topography, on demand. Our design is inspired by the morphology of the giant cactus (Carnegia gigantea) which possesses an array of axial grooves, which are thought to help reduce aerodynamic drag, thereby enhancing the structural robustness of the plant under wind loading. We perform systematic wind tunnel tests on cylinders covered with our Smporphs and characterize their aerodynamic performance. The switchable and tunable nature of our system offers substantial advantages for aerodynamic performance when compared to static topographies, due to their operation over a wider range of flow conditions.

9:18AM A25.00007 The effect of porosity and flexibility on the hydrodynamics behind a mangrove-like root model , AMIRKHOSRO KAZEMI, Florida Atlantic Univ, SAMANTHA PARRY, Ocean and Mechanical Engineering, Florida Atlantic University, KEITH VAN DE RIET, School of Architecture, Florida Atlantic University, OSCAR CURET, Ocean and Mechanical Engineering, Florida Atlantic University — Mangroves play a prominent role in coastal areas in subtropical regions. Mangrove forests are of special interest to protect shorelines against storm surges, hurricane winds, sea-level rise and tsunamis. In addition, mangroves play a critical role in filtering water and providing habitat to different organisms. In this work we study the complex interaction of water flow and mangrove roots which were modeled with a circular array of cylinders with different spacing between them as well as different configurations. In addition, we modeled the flexibility of the roots by attaching rigid cylinders to torsional connectors. The models were tested in a water tunnel for a range of Reynolds number from 2200 to 12000. In a series of experiments we measured the drag force, instant and mean velocity behind the models. We also performed 2D flow visualization for the models in a flowing soap film setup. The results show that the minimum velocity of the wake is highly dependent on the porosity and flexibility of the roots. We observed that there is a small-scale turbulent region. This turbulence is recombined downstream in a larger vortex structure eventually forming a von Karman vortex street wake. We compare the results from rigid cylinder and the flexible counterpart.
9:31AM A25.00008 Optimal root arrangement of cereal crops, YEONSU JUNG, KEUNHWAN PARK, HO-YOUNG KIM¹, Seoul National University — The plant root absorbs water from the soil and supplies it to the rest part of the plant. It consists of a number of root fibers, through whose surfaces water uptake occurs. There is an intriguing observation that for most of cereal crops such as maize and wheat, the volume density of root in the soil declines exponentially as a function of depth. To understand this empirical finding, we construct a theoretical model of root water uptake, where mass transfer into root surface is modeled just as heat flux around a thin plate. It consists of a number of root fibers, through whose surfaces water uptake occurs. There is an intriguing observation that for Most previous studies have examined steady swimming, but a few have looked at linear accelerations, even though most fish do not often swim steadily. During steady swimming, thrust and drag forces are balanced, which makes it difficult to separate the two, but during acceleration, thrust exceeds drag, making it easier to measure. This study used particle imaging velocimetry (PIV) to compare the structure of the wake during steady swimming and acceleration and to estimate the axial force. Axial force increased during the orientation of the vortices did not differ between steady swimming and acceleration, which is different than anguilliform swimming, whose wakes change structure during acceleration. This difference may point to fundamental differences between the two swimming modes.

1Department of Mechanical and Aerospace Engineering, Seoul National University, Seoul 08826, Korea

9:44AM A25.00009 Moisture-driven actuators inspired by motility of plants, BEOMJUNE SHIN, MINHEE LEE, HO-YOUNG KIM, Seoul National University — We report design and fabrication of moisture-driven actuators mimicking pine cones, wild wheat and seeds of Erodium cicutarium, which can bend and even helically coil with variation of environmental humidity. The actuators adopt a bilayer configuration, one of whose layers is hygroscopically active while the other is inactive. In order to enhance the degree and speed of deformation which critically depends on moisture-responsivity of the active layer, nanofibers of hydrogel are directionally deposited on the inactive layer via electrospinning. As a result, several designs of soft robots are demonstrated which are capable of locomotion by harvesting environmental humidity energy. The dynamics of the robots are analyzed by coupling moisture diffusion kinetics and elastic theory of multi-layer bending. The theoretical predictions are compared with the experimental results, to lead to the optimal design to maximize the locomotion speed measured by travel distance normalized by body length per unit time.

Sunday, November 22, 2015 8:00AM - 9:57AM –
Session A26 Biofluids: Flexible Swimmers I

8:00AM A26.00001 The hydrodynamics of linear accelerations in bluegill sunfish, Lepomis macrochirus¹, TYLER WISE, ALEX BODEN, MARGOT SCHWALBE, ERIC TYTELL, Tufts University — As fish swim, their body interacts with the fluid around them in order to generate thrust. In this study, we examined the hydrodynamics of linear acceleration by bluegill sunfish, Lepomis macrochirus, which swims using a carangiform mode. Carangiform swimmers primarily use their caudal fin and posterior body for propulsion, which is different from anguilliform swimmers, like eels, that undulate almost their whole body to swim. Most previous studies have examined steady swimming, but a few have looked at linear accelerations, even though most fish do not often swim steadily. During steady swimming, thrust and drag forces are balanced, which makes it difficult to separate the two, but during acceleration, thrust exceeds drag, making it easier to measure. This study used particle image velocimetry (PIV) to compare the structure of the wake during steady swimming and acceleration and to estimate the axial force. Axial force increased during the orientation of the vortices did not differ between steady swimming and acceleration, which is different than anguilliform swimming, whose wakes change structure during acceleration. This difference may point to fundamental differences between the two swimming modes.

1This material is based upon work supported by the U. S. Army Research Office under grant number W911NF-14-1-0494.

8:13AM A26.00002 Effect of aspect ratio in free-swimming plunging flexible plates, PETER YEH, ALEXANDER ALEXEEV, Georgia Institute of Technology — Using three dimensional fully-coupled fluid-structure interaction simulations, we investigate the free swimming of plunging elastic rectangular plates with aspect ratios ranging from 0.5 to 5 in a viscous fluid with Reynolds number 250. We find that maximum velocity occurs near the first natural frequency regardless of aspect ratio, while the maximum swimming economy occurs away from the first natural frequency and corresponds to a specific swimmer bending pattern characterized by reduced displacement of the swimmer’s center of mass. Furthermore, we find that swimmers with wider span are both faster and more economical than narrow swimmers. These faster speeds are due to decreased drag for low aspect ratio plunging swimmers, which is in agreement with a recently proposed vortex-induced drag model that suggests that the smaller relative size of side vortices in low aspect ratio swimmers creates less drag per unit width. Our results are useful for the design of small autonomous micro-swimming devices and also provide insights on the physics of aquatic locomotion using oscillating fins.

8:26AM A26.00003 Numerical simulations of chordwise flexible pitching foils: are expanding or contracting forms more efficient?, KAI SCHNEIDER, M2P2-CNRS & CMI Aix-Marseille University, Marseille, France, THOMAS ENGELS, M2P2-CNRS, Aix-Marseille University, Marseille, France & Institut für Strömungsmechanik und Technische Akustik (ISTA), TU Berlin, Germany — We present three-dimensional direct numerical simulations of chord-wise flexible plates of different shape with driven pitching motion. We focus on the tip vortices originating from three-dimensional effects due to the finite span. These vortices are important when predicting the swimmers cruising velocity, since they contribute significantly to the drag force. First we consider rectangular swimmers with different aspect ratios and compare with an experimental study (Raspa et al., Phys. Fluids 26, 2014). Then we study expanding and a contracting shapes. We find the cruising velocity of the contracting swimmer to be higher than the rectangular one, which in turn is higher than the expanding one, while the power requirements are the lowest for the contracting shape. We provide evidence that this finding is due to the tip vortices interacting differently with the swimmer.

8:39AM A26.00004 Numerical Investigation of the C-start in an Elastic Plate, DANIEL CANUTO, JEFF ELDREDGE, University of California Los Angeles, ROBERTO ZENIT, Universidad Autonoma de Mexico — The C-start is a swimming mechanism employed by certain fish to achieve rapid acceleration from rest. In addition to its relatively low energy cost, the agility that this mechanism permits makes its understanding important for the design of biomimetic swimmers. To investigate the dynamics of C-starts, we conduct two-dimensional numerical simulations of an elastic plate rotated about its edge through a specified angle. The plate is free to translate in one direction, normal to its rotational axis and parallel to the plate’s final orientation. The results obtained are compared with experimental data. Based on the swimming velocity, it is found that the C-start can be divided into three distinct stages: motion begins with a period of nearly constant acceleration, continues with a transient period as the tail’s rotation ends, and concludes with a very gradual deceleration, or “coasting”. These stages are analyzed, as are the effects of important design parameters (e.g., body density, bending stiffness) on the dynamics observed in each stage.
8:52AM A26.00005 3D Kinematics and Hydrodynamic Analysis of Freely Swimming Cetacean1, YAN REN, DUSTIN SHEINBERG, GENG LIU, HAIBO DONG, University of Virginia, FRANK E. FISH, West Chester University, JOVERIA JAVED, Robert E. Lee High School — It’s widely thought that flexibility and the ability to control flexibility are crucial elements in determining the performance of animal swimming. However, there is a lack of quantification of both span-wise and chord-wise deformation of Cetacean’s flukes and associated hydrodynamic performance during actively swimming. To fill this gap, we examined the motion and flexure of both dolphin fluke and orca fluke in steady swimming using a combined experimental and computational approach. It is found that the fluke surface morphing can effectively modulate the flow structures and influence the propulsive performance. Findings from this work are fundamental for understanding key kinematic features of effective Cetacean propulsors, and for quantifying the hydrodynamic force production that naturally occurs during different types of swimming.

This work is supported by ONR MURI N00014-14-1-0533 and NSF CBET-1313217

9:05AM A26.00006 Investigation of the Role of Planform Shape and Swimming Gait in Cetacean Propulsion1, FATMA AYANCIK, Lehigh University, FRANK E. FISH, West Chester University, KEITH W. MOORED, Lehigh University — Dolphins and whales, known as cetaceans, have morphological characteristics associated with enhanced thrust production, high propulsive efficiency and reduced drag. These animals oscillate their moderate aspect ratio flukes in a heaving and pitching motion to propel themselves through the water. Surprisingly, these animals display a large variation in their fluke shape and swimming gait. The present study aims to probe the connection between the fluke shape and swimming gait in high performance swimming. The planform shape of cetacean flukes is parameterized with a NACA-inspired function where the coefficients are fit to several species. An unsteady three-dimensional boundary element method is used to identify the thrust production, energetics and wake structure of free-swimming flukes with an added virtual body drag. The shape and gait parameters of the different species are exchanged to gain a broader understanding of the connection between shape and gait. The numerical results are compared with lunate tail theory to assess the limitations of the theory and its predictions of force and energetic scalings.

Supported by the Office of Naval Research under Program Director Dr. Bob Brizzolara, MURI grant number N00014-14-1-0533.

9:18AM A26.00007 Performance of an unsteady plate with a two-dimensional body attached upstream1, RODRIGO LISAZO, TYLER VAN BUREN, DANIEL FLORYAN, DEVON HARTSOUGH, EMILE OSHIMA, CLARENCE W. ROWLEY, Princeton University, ALEXANDER SMITS, Princeton University and Monash University — We present results from experiments and simulations on a biologically-inspired two-dimensional heaving and pitching rigid plate. Two cases are considered: an isolated pitching plate actuated sinusoidally at the leading edge, and a pitching plate attached at the trailing edge of a stationary, streamlined two-dimensional panel. Experiments were conducted in a water channel facility equipped with a six component load cell and particle image velocimetry (PIV). Simulations were performed using an immersed boundary method. Of particular interest are (1) the impact of leading edge separation on propulsive efficiency, (2) the influence of the incoming boundary layer on the trailing vortex structure and (3) the effects of the unsteady motion on the behavior of the boundary layer.

The work was supported by ONR under MURI Grant N00014-14-1-0533.

9:31AM A26.00008 A robotic device with a passive undulating ribbon fin: kinematics and propulsive performance1, HANLIN LIU, OSCAR CURET, Florida Atlantic University — Many aquatic animals swim with high maneuverability using undulating ribbon fins. In this type of swimming, the organism propels by sending one or multiple traveling waves along an elongated fin. In previous work, robotic models with fully actuated fins where the parameters of the traveling waves are fully prescribed have been used to study the propulsive performance and fluid dynamics of this type of propulsion. However, less work has been done in ribbon fins with passively undulating waves. In this work, we use a robotic device to study the kinematics and propulsive performance of a passively undulating ribbon fin. The physical model is composed of fifteen rays interconnected with a membrane. Only two rays are actuated while the other rays are free to rotate through a common axis. The robotic fin was tested in a flume at different flow conditions. In a series of experiments we measured fin kinematics, propulsive forces and power consumption. As the leading two rays are actuated, a traveling wave with decaying amplitude passes through the undulating waves. As the frequency of the actuated rays increases, the enclosed area of the undulating wave and the traveling wave frequency increase while the wavelength decreases. Our data also show that the propulsive force generated by the fin scaled with the enclosed area and the square of the relative velocity between incoming flow and traveling wave. These results suggest that both natural swimmers and underwater vehicles using ribbon-fin-based propulsion can potentially take advantage of passive undulating waves.

National Science Foundation Grant No. 1420774

9:44AM A26.00009 Dynamic Surface Morphing of Sunfish Caudal Fin Enhances Its Propulsive Efficiency in Steady Swimming1, GENG LIU, CHENGYU LI, HAIBO DONG, University of Virginia, GEORGE LAUDER, Harvard University — In this work, an integrated experimental and computational approach has been used to investigate the correlation between the propulsive performance and surface morphology of bluegill sunfish’s caudal fin in steady swimming. 3D sunfish caudal fin kinematics and surface morphology were reconstructed based on the output of a high-speed photogrammetry system. Hydrodynamic performance and wake structures were numerically studied by an in-house immersed-boundary-method flow solver. It is found that the spanwise surface morphing enhances both the thrust and the propulsive efficiency by more than 30%. Further investigation of the near-field and far-field wakes has shown that the enhanced span edge vortices were responsible for the performance improvement. Vortex dynamics analyses of such unsteady flow are expected to provide physical insight into the understanding of a potential performance enhancement mechanism in bluegill sunfish caudal fin propulsion.

This work is supported by ONR MURI N00014-14-1-0533 and NSF CBET-1313217.

Sunday, November 22, 2015 8:00AM - 9:57AM – Session A27 Biofluids: Flapping 308 - Matthew Ringuette, State University of New York at Buffalo
8:00AM A27.00001 Vortex Loop Topology During the Stroke Reversal of a Flapping Wing

This work is supported by the National Science Foundation, Award Number 1336548, supervised by Dr. Dimitrios Papavassiliou.

8:13AM A27.00002 Schooling of two tandem flapping wings: Simulations and theory

FANG FANG, SOPHIE RAMANANARIVO, LEIF RISTROPH, MICHAEL SHELLEY, Courant Inst, APPLIED MATH LAB, NYU TEAM — We examine theoretically the hydrodynamic interaction of two tandem flapping wings. The two wings heave vertically with the same prescribed sinusoidal motion and each wing is free to choose its locomotion speed in the horizontal direction. We model the wings as flat plates and apply an improved vortex sheet simulation method to study their interaction through the fluid. Multiple stable schooling states are found from simulations and are consistent with experimental results. By applying an external load on the follower wing, we map out an effective hydrodynamic potential acting on the follower as a function of the ‘schooling number’, which is defined as the tail-to-head separation distance over the wake wavelength. The hydrodynamic potential and drag-induced dissipation function are also calculated theoretically by applying a linear theory for the motion of the leader, the wake it produces, and for its effect on the follower.

8:26AM A27.00003 “Schooling” of wing pairs in flapping flight

SOPHIE RAMANANARIVO, Courant Institute, NYU, JUN ZHANG, NYU, and NYU Shanghai, LEIF RISTROPH, Courant Institute, NYU, AML, COURANT COLLABORATION, PHYSICS NYU COLLABORATION — The experimental setup implements two independent flapping wings swimming in tandem. Both are driven with the same prescribed vertical heaving motion, but the horizontal motion is free, which means that the swimmers can take up any relative position and forward speed. Experiments show however clearly coordinated motions, where the pair of wings crystallize into specific stable arrangements. The follower wing locks into the path of the leader, adopting its speed, and with a separation distance that takes on one of several discrete values. By systematically varying the kinematics and wing size, we show that the set of stable spacings is dictated by the wavelength of the periodic wake structure. The forces maintaining the pair cohesion are characterized by applying an external force to the follower to perturb it away from the ‘stable wells’. These results show that hydrodynamics alone is sufficient to induce cohesive and coordinated collective locomotion through a fluid, and we discuss the hypothesis that fish schools and bird flocks also represent stable modes of motion.

8:39AM A27.00004 Experimental Investigation of the Unsteady Flow Structures of Two Interacting Pitching Wings

MELIKE KURT, KEITH MOORED, Lehigh Univ — Birds, insects and fish propel themselves with unsteady motions of their wings and fins. Many of these animals are also found to fly or swim in three-dimensional flocks and schools. Numerous studies have explored the three-dimensional steady flow interactions and the two-dimensional unsteady flow interactions in collectives. Yet, the characterization of the three-dimensional unsteady interactions remains relatively unexplored. This study aims to characterize the flow structures and interactions between two sinusoidally pitching finite-span wings. The arrangement of the wings varies from a tandem to a bi-plane configuration. The vortex structures for these various arrangements are quantified by using particle image velocimetry. The vortex-wing interactions are also characterized as the synchrony between the wings is modified.

8:52AM A27.00005 Investigating the Force Production of Functionally-Graded Flexible Wings in Flapping Wing Flight

DURLAV MUDBHARI, MALCOLM ERDOGAN, KAI HE, DANIEL BATEMAN, RORY LIPKIS, KEITH MOORED, Lehigh University — While it is understood that flexibility can improve the propulsive performance of flapping wings and fins, the flexion ratio distribution leading to optimal performance has not been explored. Using 2D small-amplitude theory and a fast Chebyshev method, we examine how thrust depends on the chord-wise distribution of wing stiffness. Through numerical optimization, we find that focusing flexibility at the wing’s front, e.g. through a torsional spring, maximizes thrust. A wing with an optimally chosen spring ratio, that is, the ratio of the length of the rigid section compared to the total span length. By varying the flexion ratio as well as the material properties of the flexible section, the study aims to examine the force production and energetics of flapping flight with functionally-graded flexible wings.

9:05AM A27.00006 Torsional spring is the optimal flexibility arrangement of a flapping wing

NICK MOORE, Florida State University — While it is understood that flexibility can improve the propulsive performance of flapping wings and fins, the flexion ratio distribution leading to optimal performance has not been explored. Using 2D small-amplitude theory and a fast Chebyshev method, we examine how thrust depends on the chord-wise distribution of wing stiffness. Through numerical optimization, we find that focusing flexibility at the wing’s front, e.g. through a torsional spring, maximizes thrust. A wing with an optimally chosen spring constant typically generates 36% more thrust than a wing of optimal uniform stiffness. These results may relate to material distributions found in nature, such as insect wings, and may apply to the design of biomimetic swimmers and flyers, such as ornithopters.

9:18AM A27.00007 Flapping locomotion of a flexible wing with heaving motion

SUNGYUK IM, HYUNG JIN SUNG, KAIST — The flapping locomotion of a freely heaving flexible wing was experimentally explored in a merry-go-round equipment. Two rectangular wings were attached at the both ends of a horizontal support bar submerged in a dodecagonal water tank. The center of the support bar was connected to the vertically flapping axis which is freely rotating. This experimental apparatus generated a pure heaving motion in the vertical direction to the flapping wings in the frequency range of 0 to 5 Hz. The propulsion due to the heaving wing was expressed by a horizontally rotating speed of the support bar. The heaving motion and the rotating speed were retained with a linear encoder and sensor, and the signal was measured with different experimental parameters. Compared to a rigid wing, the flexible wing in the heaving motion showed a better propulsive performance in some conditions. The effects of the flexibility, the aspect ratio, and the thickness of the heaving wing on the propulsive performance were examined.

This work was supported by the Creative Research Initiatives (No. 2015-001828) program of the National Research Foundation of Korea (MSIP).

1Supported by the Office of Naval Research under Program Director Dr. Bob Brizolara, MURI grant number N00014-14-1-0533.
9:31AM A27.00008 The role of tip deflection on the thrust produced by rigid flapping fins, FRANCISCO HUERA-HUARTE, Department of Mechanical Engineering, Universitat Rovira i Virgili, MORTEZA GCHARIB, Division of Engineering and Applied Science, California Institute of Technology — It is well known that flexibility plays an important role in the propulsion performance and efficiency of oscillating fin based propulsion systems. Compliance is one of the aspects that has received more attention, as it seems to be a common feature in nature’s flyers and swimmers. Active control strategies are also common in nature. We will show how by deflecting only the last 10% of length of a rigid fin, at the tip, the thrust can be changed dramatically. This can be thought as an alternative to passive flexibility for controlling very efficiently the momentum transfer in the wake and therefore the thrust generation when flapping. A series of experiments have been carried with a robotic fin that allowed the control of its flapping kinematics as well as the control of the motions of its tip independently. We will be showing situations in which the tip was kept at a certain fixed position during a power stroke, and others in which it moved either in-phase or out-of-phase with the fin. The observed thrust and wake dynamics will be discussed for all these situations. The authors would like to acknowledge the financial support provided by the Gordon and Betty Moore Foundation and by the Spanish Ministerio de Economía y competitividad (MINECO) through grant DPI2012-37904.

9:44AM A27.00009 Fanning the Optimal Breeze with an Abanico, GRACE GOON, JOEL MARTHELOT, PEDRO REIS, Massachusetts Inst of Tech-MIT — Flexible hand-held fans, or abanicos, are universally employed as cooling devices that are both portable and sustainable. Their to and fro axial motion about one’s hand generates an airflow that increases the evaporation rate near the skin and refreshes. We study this problem in the context of fluid-structure interaction, through precision model experiments. We first characterize the elastic properties of a semi-circular thin plates with various thickness and evaluate their aerodynamic performance in a custom built apparatus. The air velocity profile that results from the flapping motion of the fan is characterized for different driving conditions. A systematic variation of the geometric and elastic parameters, along with an exploration of the parameter space of the periodic driving motion (amplitude and frequency), allows us to establish optimal design and operational conditions for maximal output of the generated airflow, while minimizing the input power.

Sunday, November 22, 2015 8:00AM - 9:57AM — Session A28 Biofluids: Motility in Newtonian and Non-Newtonian Fluids

8:00AM A28.00001 Motility modes of the parasite Trypanosoma brucei1, FATMA ZEYNEP TEMEL, ZIJIE QU, MICHAEL MCALLASTER, CHRISTOPHER DE GRAFFENRIED, KENNETH BREUER, Brown University — The parasitic single-celled protozoan Trypanosoma brucei causes African Sleeping Sickness, which is a fatal disease in humans and animals that threatens more than 60 million people in 36 African countries. Cell motility plays a critical role in the developmental phases and dissemination of the parasite. Unlike many other motile cells such as bacteria Escherichia coli or Caulobacter crescentus, the flagellum of T. brucei is attached along the length of its awl-like body, producing a unique mode of motility that is not fully understood or characterized. Here, we report on the motility of T. brucei, which swims using its single flagellum employing both rotating and undulating propulsion modes. We tracked cells in real-time in three dimensions using fluorescent microscopy. Data obtained from experiments using both short-term tracking within the field of view and long-term tracking using a tracking microscope were analyzed. Motility modes and swimming speed were analyzed as functions of cell size, rotation rate and undulation pattern.

8:13AM A28.00002 The fluid dynamics of the ciliate Pseudotontonia sp. jumping by “tail” contraction, HOUSHUO JIANG, Woods Hole Oceanographic Institution, Woods Hole, MA 02543, BRAD GEMMELL, EDWARD BUSKEY, University of Texas, Marine Science Institute, Port Aransas, TX 78373 — The marine planktonic ciliate Pseudotontonia sp. (~ 80 µm in cell size) possesses two sets of propulsive machinery: (1) an anteriorly located ciliary band that beats to let the cell swim backward, and (2) a long, contractile appendage (i.e. the ‘tail’) that at times contracts rapidly to pull the cell body backward, resulting in the tail contraction and body jumping motion being oppositely directed inwards towards the same location. We use high-speed microscopy imaging and micro-scale image velocimetry techniques to measure the ciliate swimming and jumping kinematics and imposed flow fields. We show that the cilia-propelled swimming achieves a sustained swimming speed ~ 10 mm s⁻¹ that can last more than 100 ms. The swimming imposed flow conforms to the steady stresslet flow field that decays spatially at r⁻². On the other hand, the tail contraction causes the cell to jump at a peak speed ~ 55 mm s⁻¹ and cover a jumping distance 2-4 cell lengths within ~ 12 ms jumping time. The jumping imposed flow fits quite well to the unsteady impulsive stokeslet flow field that decays spatially at r⁻³. Based on the measured jumping kinematics, we develop a fluid dynamics model to explain the thrust generation due to the tail contraction.

8:26AM A28.00003 Investigation of the swimming mechanics of Schistosoma cercariae and its role in disease transmission, DEEPAK KRISHNAMURTHY, ARJUN BHARGAVA, GEORGIOS KATSIKIS, MANU PRAKASH, Stanford University — Schistosomiasis is a Neglected Tropical Disease responsible for the deaths of an estimated 200,000 people annually. Human infection occurs when the infectious forms of the worm known as cercariae swim through freshwater, detect humans and penetrate the skin. Cercarial swimming is a bottleneck in disease transmission since cercariae have finite energy reserves, hence motivating studies of their swimming mechanics. Here we build on earlier studies which revealed the existence of two swimming modes: the tail-first and head-first modes. Of these the former was shown to display a novel symmetry breaking mechanism enabling locomotion at low Reynolds numbers. Here we propose simple models for the two swimming modes based on a three-link swimmer geometry. Using local slender-body-theory, we calculate the swimming gait for these model swimmers and compare with experiments, both on live cercariae and on scaled-up robotic swimmers. We use data from these experiments and the models to calculate the energy expended while swimming in the two modes. This along with long-time tracking of swimming cercariae in a lab setting allows estimation of the decrease in activity of the swimmer as a function of time which is an important factor in cercarial infectivity. Finally, we consider, through experiments and theoretical models, the effects of gravity since cercariae are negatively buoyant and sink in the water column while not swimming. This sinking affects cercarial spatial distribution which is important from a disease perspective.
How an organism gets its shape remains an open question of fundamental science. In this study, we measure the 3D shape of a bacterium, Caulobacter crescentus, using a computational graphic technique for free-swimming microorganisms to analyze thousands of image frames of the same individual bacterium. Rather than having a crescent shape, the cell body of the organism is found to be twisted with a helical pitch angle around 45 degrees. Moreover, the detailed size and geometry of the cell body, matches the optimized cell body obtained by the slender body theory for swimming at fixed power. This result shed new light on the shape evolution of microorganisms, and suggests that C. crescentus has adapted to its natural habitat of fresh-water lakes and streams, lacking nutrients.

At low compression rates, the ribbon buckles quasi-statically to form the standard one-mode shape in agreement with the fundamental Euler body theory for swimming at fixed power. This result shed new light on the shape evolution of microorganisms, and suggests that C. crescentus has adapted to its natural habitat of fresh-water lakes and streams, lacking nutrients.

In all cases, flexing a uniflagellated bacterium, caulobacter crescentus, in solutions of a number of polymers of several different sizes. Our findings confirm the peaked speed-viscosity curve, only as the molecular weight of the flexible polymers used surpassed ~ 50,000 da. The threshold molecular weight required to augment swimming speed varies somewhat with the polymer species, but it generally corresponds to radius of gyration over tens of nanometers. This general feature is consistent with the model of Powers et al. (Physics of Fluid, 2009), predicting that nonlinear viscoelasticity of the fluid enhances swimming motility.

1 Work Supported by the NSF Fluid Physics Program (Award number CBET 1438033).

9:18AM A28.00007 Buckling Instabilities and Complex Dynamics in a Model of Unflagellated Bacterial Locomotion1, FRANK NGUYEN, MICHAEL GRAHAM, University of Wisconsin, Madison — Locomotion of microorganisms at low Reynolds number is a long studied problem. Of particular interest are organisms using a single flagellum to undergo a wide range of motions: pushing, pulling, and tumbling or flicking. Recent experiments have connected the stability of the hook protein, connecting cell motor and flagellum, to deviations from typical straight swimming trajectories. We seek physical explanations to these phenomena by developing a computationally inexpensive, rigid-body dynamic model of a uniflagellated organism with a flexible hook connection that captures the fundamental dynamics, kinematics, and configurations. Furthermore, the model addresses the effects of hook loading and geometry on the stability of the system. Simulations with low hook flexibility produce the classic straight trajectory, but a large flexibility produces helical trajectories, leading to directional changes when coupled with transient hook stiffening. Minima for critical flexibilities are found in certain subsets of parameter space, implying preferred geometries for certain swimming dynamics. The model verifies proposed mechanisms for swimming in various modes and highlights the role of flexibility in the biology of real organisms and the engineering of artificial microswimmers.

1 This work was supported by NSF grant PHY-1304942

9:31AM A28.00008 MOVED TO M26.003

9:44AM A28.00009 Dynamics of Buckling of an Elastic filament in a Viscous fluid1, MOUMITA DASGUPTA, ARSHAD KUDROLLI, Clark University — We study the buckling of an elastic filament when immersed in a Newtonian fluid as it undergoes a uniaxial compression. Although there have been investigations of buckling of semi-flexible filaments in complex materials including locomotion of microorganisms, in cytoskeleton of microtubules and helical plant roots, there is a gap in the understanding of the dynamics of buckling instability for the simpler Newtonian case. Therefore, we investigated the growth of buckled modes of an elastic ribbon under various compression rates which buckles into configurations which depend of the relative magnitude of the elastic and viscous forces. At low compression rates, the ribbon buckles quasi-statically to form the standard one-mode shape in agreement with the fundamental Euler buckling mode. As the compression rate is increased, the ribbon undergoes systematic increase in the number of modes at onset. In all cases, the ribbon relaxes after compression stops to the fundamental Euler mode. We will discuss the fits to the shape in terms of sums of Euler modes as well as Fourier modes, and their growth and decay. Finally, the effect of the fluid viscosity on the evolution of the buckled mode will be discussed.

1Supported under NSF Grant # DMR1508186

Sunday, November 22, 2015 8:00AM - 9:57AM
Session A29 Nonlinear Dynamics: Coherent Structures I
310 - Nicholas Ouellette, Stanford University
are in full (qualitative) agreement with observed behavior in the actual flows by way of a linearized representation of the equations of motion near the periodic lines. Predictions on the basis of this investigation yet the underlying mechanisms are to date only partially understood. This study deepens insight into the (perturbed) Lagrangian dynamics of trajectories embedded in chaotic regions. Weak perturbation destroys the periodic lines and causes said trajectories to coalesce into families of concentric closed lines generically consist of elliptic and/or hyperbolic points and thus give rise to 3D flow topologies made up of families of concentric closed of periodic points are key organizing entities in the Lagrangian flow topology of certain three-dimensional (3D) time-periodic flows. Such

The importance of coherent structures in transport and mixing is well established in many fields, including oceanography, meteorology, and atmospheric sciences. Coherent Lagrangian structures (LCSs) are particularly useful in understanding the transport of material tracers in unsteady flows where traditional Eulerian diagnostics are less effective.

At the core of LCS analysis is the fact that Lagrangian vortices can be viewed as material surfaces, and thus as surfaces moving with the fluid. LCSs provide a way to identify and analyze these surfaces in a way that is directly linked to particle motion. This approach can be used to study the organization of fluid flow and the transport of material tracers.

The technique of LCS analysis involves identifying Lagrangian structures as clusters of Lagrangian trajectories. These trajectories are computed by tracking passive particles advected by the fluid flow. When particles are distributed densely, as can be achieved in laboratory, the fluid velocity field can be reconstructed through Particle Tracking Velocimetry (PTV), enabling computation of Lyapunov exponents or other numerical analyses.

When particles are sparse, as in drifter measurements of oceans, the velocity field cannot be reliably reconstructed. Nevertheless, the amount of stretching and mixing in the flow can still be measured by tracking the motion of topological loops. This approach is particularly useful in regions of chaotic and regular flow, where traditional Eulerian diagnostics may be less effective.

In summary, LCS analysis provides a powerful tool for studying transport and mixing in unsteady flows. By identifying coherent Lagrangian vortices, we can gain insight into the organization of fluid flow and the transport of material tracers, which is crucial for understanding many natural and industrial processes.

---

1 Funded by NSF CMMI-1233935
9:31AM A29.00008 Lagrangian coherent structures as mesoscale transport barriers in atmospheric flows, SHIBABRAT NAIK, SHANE ROSS, Biomedical Engineering and Mechanics, Engineering Science and Mechanics program, Virginia Tech — Coherent structures in two-dimensional flows have long been studied in the context of transport in fluid dynamics. However, for geophysical systems a small vertical velocity can lead to nontrivial three-dimensional motion of airborne biological populations affecting agriculture or hazardous outputs from natural disasters. The pathways and barriers in the lower atmosphere, from ground level to a kilometer altitude and over a horizontal scale of several kilometers—which bridge the scale of, for example, local farmlands to the larger regional scale—are still unclear. This requires exploring relevant spatiotemporal scales related to advection in the space of 3D + time. In this talk, we will present the application of finite-time Lyapunov exponent based three-dimensional Lagrangian coherent structures (LCS) to address questions of transport using historical data sets from satellite observations, field measurements and the Weather Research and Forecasting (WRF) model.

9:44AM A29.00009 The domain dependence of chemotaxis in two-dimensional turbulence, WENBO TANG, KIMBERLY JONES, PHILLIP WALKER, Arizona State University — Coherent structures are ubiquitous in environmental and geophysical flows and they affect reaction-diffusion processes in profound ways. In this presentation, we show an example of the domain dependence of chemotaxis process in a two-dimensional turbulent flow. The flow has coherent structures that form barriers that prohibit long-range transport of tracers. Accordingly, the uptake advantage of nutrient by motile and nonmotile species differs significantly if the process start in different locations of the flow. Interestingly, the conventional diagnostic of finite-time Lyapunov exponents alone is not sufficient to explain the variability — methods to extract elliptic transport barriers are essential to relate to the explanation. We also offer some explanations of the observed scalar behaviors via analyses of bulk quantities.

Support: NSF-DMS-1212144

Sunday, November 22, 2015 8:00AM - 9:44AM
Session A30 DFD: Geophysical Fluid Dynamics: Rotating and Stratified Convection
311 - Keith Julien, University of Colorado

8:00AM A30.00001 Asymptotically reduced equations for rapidly rotating and stably stratified flow, DAVID NIEVES, KEITH JULIEN, Univ of Colorado - Boulder — Observations by van Haren & Millot (2005) of the deep Western Mediterranean Sea and by Timmermans et al. (2006) of the deep Canadian Basin find vertical fluid motions to be as significant as horizontal motions for ocean dynamics. Since the classical quasi-geostrophic equations do not allow for such vertical motions reduced equations for geostrophically balanced flow with O(1) vertical motions are presented alongside their numerical solutions and results. The reduced equations describe flow constrained by rapid rotation and stable stratification and, in fact, are the stably stratified counterpart to the reduced equations used by Julien et al. in successful studies of rapidly rotating Rayleigh-Bénard convection. Specifically, the equations are valid in the small Rossby number (Ro 1) and O(1) Froude number limit. The focus here is a comparison to similar studies of rotating and stratified flow by Smith & Waleffe (2002), Wingate et al. (2011), and Marino et al. (2013) among others.

8:13AM A30.00002 Rotating Rayleigh-Bénard convection with Ekman pumping, MEREDITH PLUMLEY, KEITH JULIEN, PHILIPPE MARTI, University of Colorado, Boulder, JONATHAN AURNOU, Department of Earth, Planetary and Space Sciences, University of California, Los Angeles, STEPHAN STELLMACHT, Institut für Geophysik, Westfälische Wilhelms-Universität Münster, Germany — Rotating Rayleigh-Bénard convection is of interest in many geoscience applications, with examples like deep ocean convection or the magnetic field generation of planets occurring in the regime where convectively driven motions are dominated by the effects of rotation. To better understand the dynamics of these large physical systems, several techniques are used including asymptotic methods, DNS and experiments. While these three methods have seen good agreement in results for stress free boundary conditions, the case of rigid no-slip boundaries presents an interesting difference. Along the no-slip boundaries, Ekman layers form and Ekman pumping occurs. It has been thought that the effect of these boundary layers is negligible for small Ekman number because of how thin they become. However, new DNS of the 3D Boussinesq equations have provided evidence that this is not the case. A new asymptotic model has been developed to include these boundary layers and verify the impact of the Ekman boundaries on the flow. Results from simulations of this new model will be compared with DNS and experimental results. The results support the findings of increased global heat transfer due to the presence of Ekman pumping.

8:26AM A30.00003 Mixing efficiency of buoyancy forced circulation in a rotating basin, CATHERINE VREUGDENHIL, BISHAKHDATTA GAYEN, ROSS GRIFFITHS, Australian National University — We ask whether rotation influences the mixing efficiency in horizontal convection in a rectangular basin. Direct numerical simulations are reported for a rotating f-plane ocean with an applied basal temperature differential over a wide range of Ekman number $E_L = 6 \times 10^{-6} - 1 \times 10^{-2}$, with Prandtl number $Pr = 5$. Two values of the Rayleigh number are considered which, in the absence of rotation, relate to the viscous $(Ra = 7.4 \times 10^{11})$ and inertial $(Ra = 7.4 \times 10^{11})$ regimes. The heat flux decreases and boundary layer thickness increases with rotation rate, consistent with geostrophic scaling. At very high rotation rates and the smaller $Ra$ a regime dominated by Ekman pumping is revealed, with strong interior stratification. For the larger $Ra$ turbulent convective plumes in the boundary layer region form cyclonic vortices that extend through the depth, weakening the stratification. The global mixing efficiency $\eta$ is consistent with the theoretical prediction $\eta = 1 - (Hu/L)^{-1}$ (where $Hu$ is the Nusselt number, $H$ is height and $L$ is length of the domain) for $Nu \gg 10$. Independent of rotation, $\eta$ approaches unity at large $Nu$, and therefore at large $Ra$. Laboratory experiments in the inertial regime with an applied heat flux are also considered.

8:39AM A30.00004 Spinup of a stratified fluid in a sliced, circular cylinder, M.R. FOSTER, The Ohio State University, R.J. MUNRO, University of Nottingham — Experiments were performed in a linearly salt-stratified fluid in a circular cylindrical tank, with a planar bottom boundary slope at a small angle $\alpha$ to the horizontal. The container rotated initially at an angular velocity $\Omega$, so that the Ekman number, $E$, was typically $10^{-5}$. We examined the adjustment when the container’s angular speed is abruptly increased by $\epsilon \Omega$, with $\epsilon \sim 0.1$. Further, $\alpha \gg E^{1/2}$, and the Burger number $S$ is large. There are similarities and differences between this spinup and that in a sliced square cylinder (Munro & Foster, Phys. Fluids 26, 2014, denoted by MF14). Unlike MF14, the axisymmetry of the initial core motion means there are no core eddies generated by boundary-layer eruption. In fact, since the core motion is nearly axisymmetric for all time at large $S$, eddy formation is confined to the region of height $O(S^{1/2})$ near the lower slope, within which the Rossby waves are confined. Just as in MF14, after several “spinup times,” the cross-container velocity profiles agree very well with a linear asymptotic theory for small $\epsilon$, $E$ and large $S$, provided one properly accounts for the Rayleigh layers on the cylinder’s sidewall.
8:52AM A30.00005 Regimes of axisymmetric flow in a rotating annulus with local convective forcing1

HELENE SCOLAN, Department of Physics, University of Oxford, Atmospheric, Oceanic & Planetary Physics, SYLVIE SU, Ecole Normale Superieure de Lyon, France, ROLAND M.B. YOUNG, PETER L. READ, Department of Physics, University of Oxford, Atmospheric, Oceanic & Planetary Physics — We present a numerical study of axisymmetric flows in a rotating annulus convectively forced by local thermal forcing via a heated annular ring at the bottom near the external wall and a cooled circular disk near the centre at the top surface. This new configuration is a variant of the classical thermally-driven annulus analogue of the atmosphere circulation, where thermal forcing was previously applied on the sidewalls. Two vertically and horizontally displaced heat sources/sinks are arranged so that, in the absence of rotation, statically unstable convection would be induced above the source and beneath the sink, thereby relaxing strong constraints placed on background temperature gradients in previous setup. By using the Met Office / Oxford Rotating Annulus Laboratory code, we investigated a series of equilibrated, 2D axisymmetric flows for a large range of dimensionless parameters and characterized them in terms of velocity and temperature fields. Several distinct flow regimes were identified, depending upon the rotation rate and strength of differential heating. These regimes will be presented with reference to variations of horizontal Ekman layer thickness versus the thermal boundary layer thickness and corresponding scalings for various quantities such as the heat transport.

1Grants: EPSRC EP/K029428/1 and studentship Met Office/Oxford

9:05AM A30.00006 A Laboratory Study of Vortical Structures in Rotating Convective Plumes1, HAO FU, SHIWEI SUN, YUAN WANG, BOWEN ZHOU. None, THERMAL TURBULENCE RESEARCH TEAM2 — A laboratory study of the columnar vortex structure in rotating Rayleigh-Bénard convection is conducted. A rectangular water tank is uniformly heated from below and cooled from above, with $Ra = (6.35 \pm 0.77) \times 10^7$, $Ta = 9.84 \times 10^7$, $Pr = 7.34$. The columnar vortices are vertically aligned and quasi steady. Two 2D PIV systems were used to measure velocity field. One system performs horizontal scans at 9 different heights every 13.6s, covering 62% of the total depth. The other system scans vertically to obtain the vertical velocity profile. The measured vertical vorticity profiles of most vortices are quasi-linear with height while the vertical velocities are nearly uniform with only a small curvature. A simple model to deduce vertical velocity profile from vertical vorticity profile is proposed. Under quasi-steady and axisymmetric conditions, a “vortex core” assumption is introduced to simplify vertical vorticity equation. A linear ODE about vertical velocity is obtained whenever a vertical vorticity profile is given and solved with experimental data as input. The result is approximately in agreement with the measurement.

1This work was supported by Undergraduates Training Project (J1103410).
2School of Atmospheric Sciences, Nanjing University

9:18AM A30.00007 Local Available Potential Energy in Simulations of Stratified Turbulence with Uniform and Non-uniform Ambient Density Gradients1, GAVIN PORTWOOD, STEPHEN DE BRUYN KOPS, University of Massachusetts Amherst, TURBULENCE SIMULATION LABORATORY TEAM — In stratified flows, the maximum amount of potential energy that can be converted to kinetic energy is the difference between the potential energy in the instantaneous flow and that in the flow if the fluid parcels were adiabatically sorted to produce the lowest energy configuration. Lorenz (1955) defines this global quantity as available potential energy (APE). Holliday and McIntyre (1981) introduces the concept of local available potential energy ($E_{ap}$) associated with a fluid parcel, and Molemaker and McWilliams (2010) develop the transport for this quantity for a viscous, Boussinesq fluid. Here, we characterize $E_{ap}$ in simulations of a vortex street with uniform and non-uniform stabilizing ambient density gradients. In pseudo-spectral direct numerical simulations resolved on up to $4096 \times 2048 \times 2048$ grid points, we find that the majority of APE is due to fluid parcels displaced a small distance, relative to the buoyancy length scale, from their locations in the sorted density field. By computing each term in the transport equation for $E_{ap}$, we observe by how much $E_{ap}$ of a fluid parcel changes in time due to local dipycnal mixing, and by how much global mixing alters the position of the local parcel in the sorted density field.

1This work is funded by DoD HPCMP though Frontier Project FPCFD-FY14-007 and the Office of Naval Research via grant N00014-15-1-2248.

9:31AM A30.00008 Heat flux in a penetrative convection experiment in water2, YOANN CORRE, THIERRY ALBOUSSIÈRE, STEPHANE LABROSSE, LGL - Laboratoire de Géologie de Lyon, Université Lyon 1, ENS Lyon, PHILIPPE ODIER, SYLVAIN JOUBAUD, Phys-ENS, Laboratoire de Physique de l’ENS Lyon — In geophysical systems, stably stratified fluids adjacent to the top surface. This new configuration is a variant of the classical thermally-driven annulus analogue of the atmosphere circulation, where thermal forcing was previously applied on the sidewalls. Two vertically and horizontally displaced heat sources/sinks are arranged so that, in the absence of rotation, statically unstable convection would be induced above the source and beneath the sink, thereby relaxing strong constraints placed on background temperature gradients in previous setup. By using the Met Office / Oxford Rotating Annulus Laboratory code, we investigated a series of equilibrated, 2D axisymmetric flows for a large range of dimensionless parameters and characterized them in terms of velocity and temperature fields. Several distinct flow regimes were identified, depending upon the rotation rate and strength of differential heating. These regimes will be presented with reference to variations of horizontal Ekman layer thickness versus the thermal boundary layer thickness and corresponding scalings for various quantities such as the heat transport.

2This work is funded by DoD HPCMP though Frontier Project FPCFD-FY14-007 and the Office of Naval Research via grant N00014-15-1-2248.

Session A31: Drops: Surface Interactions

8:00AM A31.00001 Formation of surface nanodroplets under controlled flow conditions1, DETLEF LOHSE, University of Twente

1Grants: EPSRC EP/K029428/1 and studentship Met Office/Oxford

Sunday, November 22, 2015 8:00AM - 9:57AM – Session A31: Drops: Surface Interactions

8:00AM A31.00001 Formation of surface nanodroplets under controlled flow conditions1, DETLEF LOHSE, Univ of Twente, XUEHUA ZHANG, RMIT Melbourne and Univ of Twente, ZIYANG LU, RMIT Melbourne, HUANSHU TAN, Univ. of Twente, LEI BAO, RMIT Melbourne, YINGHE HE, James Cook University, Townsville City, CHAO SUN, Univ. of Twente — Nanodroplets on a solid surface (i.e. surface nanodroplets) have practical implications for high-throughput chemical and biological analysis, lubrications, lab-on-chip devices, and near-field imaging techniques. Oil nanodroplets can be produced on a solid-liquid interface in a pseudo-spectral direct numerical simulations resolved on up to $4096 \times 2048 \times 2048$ grid points, we find that the majority of APE is due to fluid parcels displaced a small distance, relative to the buoyancy length scale, from their locations in the sorted density field. By computing each term in the transport equation for $E_{ap}$, we observe by how much $E_{ap}$ of a fluid parcel changes in time due to local dipycnal mixing, and by how much global mixing alters the position of the local parcel in the sorted density field.

1Grants: EPSRC EP/K029428/1 and studentship Met Office/Oxford

Session A31: Drops: Surface Interactions

8:00AM A31.00001 Formation of surface nanodroplets under controlled flow conditions1, DETLEF LOHSE, Univ of Twente, XUEHUA ZHANG, RMIT Melbourne and Univ of Twente, ZIYANG LU, RMIT Melbourne, HUANSHU TAN, Univ. of Twente, LEI BAO, RMIT Melbourne, YINGHE HE, James Cook University, Townsville City, CHAO SUN, Univ. of Twente — Nanodroplets on a solid surface (i.e. surface nanodroplets) have practical implications for high-throughput chemical and biological analysis, lubrications, lab-on-chip devices, and near-field imaging techniques. Oil nanodroplets can be produced on a solid-liquid interface in a simple step of solvent exchange in which a good solvent of oil is displaced by a poor solvent. In this work, we experimentally and theoretically investigate the formation of nanodroplets by the solvent exchange process under well-controlled flow conditions. We find significant effects from the flow rate and the flow geometry on the droplet size. We develop a theoretical framework to account for these effects. The main idea is that the droplet nuclei are exposed to an oil oversaturation pulse during the exchange process. The analysis gives that the volume of the nanodroplets increases with the Peclet number $Pe$ of the flow as $\propto Pe^{3/4}$, which is in good agreement with our experimental results. In addition, at fixed flow rate and thus fixed Peclet number, larger and less homogeneously distributed droplets formed at less narrow channels, due to convection effects originating from the density difference between the two solutions of the solvent exchange.
This structure allows the interface to undergo large deformations including the rupture and coalescence of fluid interfaces. Tracking/Level Set technique which defines the interface both by a discontinuous density field as well as by a local triangular Lagrangian mesh. Communication is handled by MPI message passing procedures. The method for the treatment of the fluid interfaces uses a hybrid Front where the velocity field is solved by a parallel GMRes method for the viscous terms and the pressure by a parallel multigrid/GMRes method.

We simulate the evolution of a three-dimensional pendant droplet through pinch-off using a new parallel two-phase flow solver called BLUE. The parallelization of the code is based on the technique of algebraic domain decomposition

SHIN, Hongik University, Republic of Korea — We simulate the evolution of a three-dimensional pendant droplet through pinch-off using a

ZHANG, RMIT University, Melbourne, Australia, XUEHUA ZHANG TEAM, DETLEF LOHSE COLLABORATION — Nanoscale droplets on a substrate are of great interest because of their relevance for droplet-based technologies for light manipulation, lab-on-chip devices, miniaturised reactors, encapsulation and many others. In this work, we establish a basic principle for the symmetrical arrangement of surface nanodroplets during their growth under simple flow conditions. In our model system, nanodroplets nucleate at the rim of spherical cap microstructures on a substrate, as a pulse of oversaturation is supplied by a solvent exchange process. We find that, while growing, the nanodroplets self-organise into highly symmetric arrangements, with respect to position, size, and mutual distance. The angle between the neighbouring droplets is four times the ratio between the base radii of the droplets and the spherical caps. We show and explain how the nanodroplets acquire the symmetrical spatial arrangement during their competitive growth and why and how the competition enhances the overall growth rate of the nucleated nanodroplets. This mechanism behind the nanodroplet self-organisation promises a simple approach for the location control of droplets with a volume down to attoliters.

DI KANG, MARINA CHUGUNOVA, ALI NADIM, Claremont Graduate University — We examine the behavior of a thin viscous liquid film on a rotating solid sphere under the influence of gravity, centrifugal force and surface tension. The model is based on the lubrication approximation in axisymmetric spherical coordinates, with no-slip at the liquid-solid interface and with normal and tangential stress balances, including Marangoni effects, at the liquid-air interface. The rotation axis is assumed to be aligned with the direction of gravity and the Coriolis force is neglected, identifying parameter regimes when the latter is justified. We show that for constant surface tension, the energy-minimizing steady states are of three different types: uniformly positive film thickness, or states with one or two dry zones on the sphere. The transient dynamics in approaching those states are also described. A stability analysis when Marangoni effects are present but in the absence of gravity and rotation identifies the parameter regimes for instability to occur and the corresponding unstable modes.

IVAN DEVIC, University of Twente, SHUHUA PENG, RMIT Melbourne, HUANSHU TAN, DETLEF LOHSE, University of Twente, XUEHUA ZHANG, RMIT Melbourne and University of Twente — Wetting of micro-patterned surfaces is of the great interest in the fundamental research and many practical applications such as open microfluidics, metal corrosion, pesticide spray and water collection. In this work, we investigate nanodroplets, partially wetting a flat surface and partially wetting a spherical surface (spherical microlens) which has a small contact angle with the flat surface. We have developed a theoretical approach for minimising the free surface energy of the nanodroplet at the rim of microlens and have also connected to data from experiments. Since the diffusion length scale is long enough for nanodroplets to obtain quasi-static shape in our experiments, with our approach we are also able to obtain growing or shrinking dynamics of the nanodroplet in this system. Of particular interest is behaviour of the contact angle of nanodroplet on the spherical surface of microlens. We find that contact angle of the minimum free surface energy shape deviates more from Young’s angle as the nanodroplet gets smaller compared to the spherical microlens, while contact angle of larger drops asymptotically approaches Young’s angle. Theoretical results partially agree with our experimental data, due to the surface heterogeneity of both substrates in our experiments.

YI XIA, CHUN-TI CHANG, PAUL STEEN, Cornell University — A water droplet placed on a hydrophobic plate is driven by plate-normal oscillations. Resulting droplet motions are largely inviscid, having Reynolds number ¿100 (Ohnesorge 0.002). We are interested in isolating the effective damping, sometimes called Davis dissipation, owing to a moving contact line that is not completely mobile. In this talk, we report energy budgets as influenced by contact angle – contact line speed relationships for variously prepared surfaces.

THANH-VINH NGUYEN, KIYOSHI MATSUMOTO, ISAO SHIMOYAMA, Univ of Tokyo — We directly measure the normal force distribution on the contact area during the 1st mode resonant vibration of a droplet using an array of MEMS based cantilever. The measurement result shows that the normal force change is the largest at the periphery of the contact area. The ratio between the amplitude of the normal force change at the periphery of the contact area at the center of the contact area was approximately 20 times, in the case of 1.8 µL water droplet whose equilibrium contact angle is 140 degrees. We also demonstrate a method to estimate viscosity based on the measurement of the droplet vibration using MEMS cantilevers. The proposed method is able to estimate viscosity using less than 3 µL water droplet whose equilibrium contact angle is 140 degrees. We also demonstrate a method to estimate viscosity based on the measurement of the droplet vibration using MEMS cantilevers. The proposed method is able to estimate viscosity using less than 3 µL sample and has a simple operating principle. We believe that this method is suitable for point-of-care testing and characterization of chemical and biological solutions.

XIN LIN, LEWIS JOHNS, RANGA NARAYANAN, University of Florida — The instability of a pendant drop is explained. The liquid in the drop is heavier than the surrounding fluid. The scaled groups that describe the stability are the scaled volume and the Bond number. We show without computations that the volume has a maximum value beyond which it must break catastrophically. However this upper bound on volume is not the instability limit for the drop for all Bond numbers. There exists a critical Bond number, above which the drop breaks before the upper bound on volume can be reached. We discuss why this occurs and what it means for the physics of the break-up.

Support for XL from NSF 0968313 is gratefully acknowledged.

CARLOS PENA, Universitat Jaume I, Spain, LYES KAHOUDAJJ, OMAR MATAR, Imperial College London, JALEL CHERGUI, DAMIR JURIC, LIMSI-CNRS, SEUNGWON SHIN, Hongik University, Republic of Korea — We simulate the evolution of a three-dimensional pendant droplet through pinch-off using a new parallel two-phase flow solver called BLUE. The parallelization of the code is based on the technique of algebraic domain decomposition where the velocity field is solved by a parallel GMRes method for the viscous terms and the pressure by a parallel multigrid/GMRes method. Communication is handled by MPI message passing procedures. The method for the treatment of the fluid interfaces uses a hybrid Front Tracking/Level Set technique which defines the interface both by a discontinuous density field as well as by a local triangular Lagrangian mesh. This structure allows the interface to undergo large deformations including the rupture and coalescence of fluid interfaces.

1 EPSRC Programme Grant, MEMPHIS, EP/K0039761/1
From the TIRFM measurement, changes in the bulk-averaged velocity (O(10 μm/s)) and the shear rate (O(100 s⁻¹)) are present. We study surfactant covered drops in uniform and density-stratified ambients, as well as clean drops entering a dissolved surfactant layer. We quantify the effects of entrainment for various Reynolds and Marangoni numbers. We also report a brief acceleration followed by a significant deceleration as a clean drop enters a surfactant layer, and describe how the adsorption rate affects those dynamics.

The approach also points towards the relationship between microscopic triple line contortions and CAH. Infusion-withdrawal experiments are used to generate distributions of the local contact angles for several such drops but of the same liquid-substrate pairs. A statistical distribution of local contact angles is used as the basis for this approach. Drops with randomly shaped triple lines but of fixed volumes were deposited on a substrate and their triple line shapes were extracted by imaging. Using a solution developed by Prabhala et al. (Langmuir, 2010), the complete three dimensional shape of the sessile drop was generated. A distribution of the local contact angles for several such drops but of the same liquid-substrate pairs is generated. This distribution is a result of several microscopic advancing and receding processes along the triple line. This distribution is used to yield an approximation of the CAH associated with the substrate. This is then compared with measurements of CAH by means of a liquid infusion-withdrawal experiment. Static measurements are shown to be sufficient to measure quasistatic contact angle hysteresis of a substrate. The approach also points towards the relationship between microscopic triple line contortions and CAH.

Sunday, November 22, 2015 8:00AM - 9:57AM — Session A32 Drops: Surface Tension Effects 313 - Francois Blanchette, UC Merced

8:00AM A32.00001 Surfactant-laden drops rising in a stratified ambient, FRANCOIS BLANCHETTE, DAVID MARTIN, UC Merced — We present results of a numerical study of the dynamics of rising drops in the presence of both surfactants and stratification. Our simulations model oil drops rising in the oceans, where naturally occurring or man-made surfactants are present. We study surfactant covered drops in uniform and density-stratified ambients, as well as clean drops entering a dissolved surfactant layer. We quantify the effects of entrainment for various Reynolds and Marangoni numbers. We also report a brief acceleration followed by a significant deceleration as a clean drop enters a surfactant layer, and describe how the adsorption rate affects those dynamics.

8:13AM A32.00002 Measurement of strong Marangoni flow near a contact line of a water droplet on hydrophobic surfaces, JOONSIK PARK, KENNETH S. BREUER, Brown University — Strong Marangoni flow from a water droplet on hydrophobic substrate has been theoretically predicted but not been quantitatively measured. Using two different experimental techniques, multi-layer flood illumination and Total Internal Reflection Fluorescence Microscopy (TIRFM), we report Marangoni flows with large (O(100 μm/s)) velocity near a contact line of a water droplet on hydrophobic substrates. The flow is measured by tracking the motion of nanoparticles with respect to the contact line, using statistical particle tracking velocimetry combined with sub-pixel edge detection algorithm. Under multi-layer flood illumination, the recirculating convective flow is identified within 5 μm vertically from the substrate. From the TIRFM measurement, the changes in the bulk-averaged velocity (O(100 μm/s)) and the shear rate (O(100 s⁻¹)) as the distance from the contact line are identified within 550 nm vertically from the substrate, and compared to the characteristic shear rate and speed from Marangoni effect, respectively. Surprisingly, both Flood and TIRFM measurements indicate high slip velocities extending as far as 33 μm from the contact line. One possible explanation is that the high slip velocity is due to the accumulation of nanobubbles near the contact line which were formed at the deposition of a droplet.

8:26AM A32.00003 The stability of a rising droplet: an inertialess nonmodal growth mechanism, GIACOMO GALLINO, LAILAI ZHU, FRANCOIS GALLAIRE, Laboratory of Fluid Mechanics and Instabilities, EPFL — Past modal stability analysis (Kojima et al. 1984) predicted that a rising or sedimenting droplet in a viscous fluid is stable in the presence of surface tension no matter how small, in contrast with experimental and numerical results. By performing a non-modal stability analysis, we demonstrate the potential for transient growth of the interfacial energy of a rising droplet in the limit of inertialess Stokes equations. The predicted critical capillary numbers agree well with that from direct numerical simulations reported in the literature (Koh & Leal 1989). Boundary integral simulations are used to delineate the critical amplitude of the most destabilizing perturbations. The critical amplitude is negatively correlated with the linear optimal energy growth, implying that the transient growth is responsible for reducing the necessary perturbation amplitude required to escape the basin of attraction of the spherical solution.

A European Research Council is acknowledged for funding the work through a starting grant (ERC SimCoMiCs 280117).

8:39AM A32.00004 Three-dimensional simulations of a rising bubble in a self-rewetting fluid, AMARNATH PREMLATA, MANOJ TRIPATHI, KIRTI SAHU, IIT Hyderabad, GEORGE KARAPETSAS, University of Thessaly, KHELLIL SEFIANE, University of Edinburgh, OMAR MATAR, Imperial College London — The motion of a gas bubble in a square channel with linearly increasing temperature in the vertical direction is investigated via 3D numerical simulations. The channel contains a so-called self-rewetting fluid whose surface tension exhibits a parabolic dependence on temperature with a well-defined minimum. An open-source finite-volume fluid flow solver, Gerris, is used with a dynamic adaptive grid refinement technique, based on the vorticity magnitude and position of the interface. We find that in self-rewetting fluids, the buoyancy-induced upward motion of the bubble is retarded by a thermocapillary-driven flow, which occurs as the bubble crosses the location at which the surface tension is minimum. The bubble then migrates downwards when thermocapillarity exceeds buoyancy. In its downward path, the bubble encounters regions of horizontal temperature gradients, which lead to bubble motion towards one of the channel walls. These phenomena are observed at sufficiently small Bond numbers and have no analogue for fluids whose surface tension decreases linearly with temperature. The mechanisms underlying these phenomena are elucidated by considering how the surface tension dependence on temperature affects the thermocapillary stresses in the flow.

EPSRC Programme Grant, MEMPHIS, EP/K0039761/1
controlled at relatively lower range of frequency was used. In the result, the internal velocity increases with the increase of the plate temperature and both flow directions of Marangoni and flow. In order to estimate the temperature gradient inside and surface tension on the droplet, a commercial software Comsol Multiphysics evaporating, the variation of contact angle and internal volume of droplet was observed by using the combination of a continuous light and a stain effect in the end. This study aims to visualize and control the Marangoni flow by using periodic vertical vibration. While the droplet is possessing small finite contact angles reckoned to be evaporation-induced. The experimentally determined droplet profiles are shown here to confirm by the predictions of a lubrication model accounting for the impact of the Marangoni effect on the droplet shape.

Overall, the experimental findings are quantitatively and the measurements are found to be in a good agreement with the classical static shape. These deviations are attributed to a Marangoni flow, due to evaporation-induced thermal gradients along the liquid-air interface, and are mostly observed in conditions of high evaporation. Unlike the classical static shapes, the distorted experimental profiles exhibit an inflection point at the contact line area. When a poorly volatile liquid is considered, however, the temperature differences and the Marangoni stresses are weak, operating in the spherical shape range (Eötvös number, $E_0 = 1.95$, and Morton number, $M = 78.20$) with creeping flow particle Reynolds number ($Re_p = 0.053$). A improved technique for measurement and processing of data acquired from simultaneous planar PIV-PLIF is used to obtain velocity and concentration fields around the drop. A progressive non-Gaussian behaviour from large scales to small scales is seen, in scale wise wavelet energy decomposition of vorticity and concentration fields. This suggests similarity with high Schmidt and Reynolds number intermittent turbulence, even in the creeping flow region. Fourier spectra of concentration and velocity shows the plethora of length scales generated by the Marangoni instabilities.

1 financial support by DAE-India, and TEQIP-India (COE-PI)
8:00AM A33.00001 Drying of a coffee drop: differences between dry and wet tables1, FRANCOIS BOULONNE, FRANÇOIS INGRéEMEAU, HOWARD STONE, Princeton University — We have all experienced that a coffee drop drying on a table leaves a ring stain. The radial flow in the drop coupled with a larger drying flux at its edge are the reasons for the particle accumulation in the liquid wedge. However, if the substrate is wet, the liquid surrounding the drop modifies the vapor distribution, and thus the drop evaporation dynamics. Our experimental observations show that the drying kinetics and the particle motion are affected by the ambient conditions. Our experimental findings indicate that the spatially varying evaporation rate and the temporal evolution of the particles forming the ring. We believe that these results are of practical interest for printing applications involving multiple drop systems or drying surfaces.

1F.B. acknowledges that the research leading to these results received funding from the People Programme (Marie Curie Actions) of the European Union’s Seventh Framework Programme (FP7/2007-2013) under REA grant agreement 623541.

8:13AM A33.00002 Evaporation of sessile droplets on smooth and structured substrates1, PIERRE COLINET, MARIE FLANDROY, SAM DEHAECK, ALEXEY REDNIKOV, BENJAMIN SOBAC, YANNIS TSOUmpAS, Universite Libre de Bruxelles — Evaporation of sessile droplets remains a topic of active research nowadays, not only due to its widespread occurrence both in nature and technology, but also because it raises a number of interesting fundamental questions. Among these, the influence of the substrate topography is far from being understood. In this presentation, we focus on the difference between droplet evaporation dynamics on a smooth substrate and on substrates with controlled geometrical heterogeneities (square or hexagonal arrays of cylindrical pillars). While the wetting dynamics of droplets on such structured substrates has already been studied (in particular it is known that the droplet shape typically tends to adopt the symmetry of the pattern it lies upon), the influence the structure has on the evaporation dynamics (including the influence of pinning) has not yet received much attention. From interferometric measurements of droplet shapes, we highlight in particular that the evaporation from structured substrates is generally faster than on smooth substrates, and that the scaling exponents characterizing the evolution of radius versus time are also different. Pinning is also shown to be favored above a certain surface fraction of pillars, both for hexagonal and square arrays.

1Financial support of FRS-FNRS, ESA-BELSPO PRODEX, ULB, and BELSPO IAP 7/38 is gratefully acknowledged.

8:26AM A33.00003 Coffee-ring effect beyond the dilute limit, JIN YOUNG KIM, SEUL-A RYU, SKKU Advanced Institute of Nanotechnology (SAINT), Sungkyunkwan University, HYUNGDAE KIM, Department of Nuclear Engineering, Kyung Hee University, JOON HEON KIM, JUNG SU PARK, Advanced Photonics Research Institute (APRI), Gwangju Institute of Science and Technology (GIST), YONG SEOK PARK, JEONG SU OH, Department of Genetic Engineering, Sungkyunkwan University, BYUNG MOOK WEON, School of Advanced Materials and Engineering, SKKU Advanced Institute of Nanotechnology (SAINT), Sungkyunkwan University — The coffee-ring effect, which is a natural generation of outward capillary flows inside drying coffee drops, is valid at the dilute limit of initial solute concentrations. If the solute is not dilute, the ring deposit is forced to have a non-zero width; higher initial concentration leads to a wider ring. Here we study the coffee-ring effect in the dense limit by demonstrating differences with various initial coffee concentrations from 0.1% to 60%. The coffee drops with high initial concentrations of real coffee particles show interesting evaporation dynamics: dense coffee drops tend to be less cylindrical. This dilute limit result is difficult to explain. We propose that the slow evaporation of dense coffee drops is associated with the ring growth rate. The coffee-ring effect becomes more significant in modern technologies such as self-assembly of nanoparticles, ink-jet printing, painting, and ceramics. The complexity in evaporation dynamics of colloidal fluids would be able to be understood by boosting the coffee-ring effects in the dilute as well as the dense limits.

8:39AM A33.00004 Coffee ring effect resulted conductive nanowire patterns by evaporating colloidal suspension droplets without sintering process1, XIAOFENG WANG, BAEKHoon SEONG, HADI TEGUH YUDISTIRA, DOYOU NG BYUN, Sungkyunkwan University — Drying colloidal suspensions containing non-volatile solute will form a ring like pattern, which is called “coffee ring effect.” Here, we present the coffee ring effect with silver nanowires dispersing into DI water, resulting in a highly dense-packed nanowire ring patterns. The effect of nanowire length, concentration, droplet size, and substrate temperature were investigated. With shorter nanowires, a distinct ring could be obtained. Meanwhile, the concentration of the colloidal suspension was found to affect the ring width. The droplet size and nanowire length played a significant role in affecting the occurrence of the coffee ring effect. When smaller droplets (i.e., less than 150 µm) containing long nanowires (≈ 20 µm), the coffee ring effect was suppressed. While smaller droplets containing short nanowires (≈ 1 µm), the coffee ring effect was not affected. By increasing the temperature of the substrate, nano-ring pattern was formed inside the original ring. The resistivity of the semi-circle of the nanowire ring was measured, and had a minimum value of 1.32 Ω cm without any sintering process. These findings could be exploited to basic study of ring stain effect as well as the practical use, such as evaporative lithography and ink-jet printing for conductive film and display.

1This research was supported by the Basic Science Research Program through the National Research Foundation of Korea (NRF) (Grant number: 2014-023284).

8:52AM A33.00005 Dependence of fluid flows in an evaporating sessile droplet on the characteristics of the substrate, LEV BARASH, Landau Institute for Theoretical Physics — Temperature distributions and the corresponding vortex structures in an evaporating sessile droplet are obtained by performing detailed numerical calculations. A Marangoni convection induced by thermal conduction in the drop and the substrate is demonstrated to be able to result not only in a single vortex, but also in two or three vortices, depending on the ratio of substrate to fluid thermal conductivities, on the substrate thickness and the contact angle. The “phase diagrams” containing information on the number, orientation and spatial location of the vortices for quasistationary fluid flows are presented and analysed. The results obtained demonstrate that the fluid flow structure in evaporating droplets can be influenced in a controlled manner by selecting substrates with appropriate properties.

9:05AM A33.00006 Dynamics of surfactant-laden evaporating droplets, GEORGE KARAPETSAS, National Technical University of Athens, KIRTI CHANDRA SAHU, Indian Institute of Technology Hyderabad, OMAR K. MATAR, Imperial College London — We consider the flow dynamics of a thin evaporating droplet in the presence of an insoluble surfactant and small particles in the bulk. Evolution equations for the film height, the interfacial surfactant and bulk particle concentration are derived using a lubrication model coupled by a constitutive relation for the dependence of the viscosity on local particle concentration. An important ingredient of our model is that it takes into account the fact that the surfactant adsorbed at the surface hinders the evaporation. Time-dependent simulations are performed to determine how the presence of surfactants affects the evaporation and flow dynamics with and without the presence of particles in the bulk. We discuss the various mechanisms that affect the shape of the droplet as it evaporates as well as the resulting pattern of particle deposition.
9:18AM A33.00007 On thin evaporating drops: when is the $d^2$-law valid? MATTHEW SAXTON, JONATHAN WHITELEY, DOMINIC VELLA, JAMES OLIVER, University of Oxford — We study the evolution of a thin, axisymmetric, partially wetting drop as it evaporates. The stress singularity at the contact line is regularized using slip and we perform a matched-asymptotic analysis in the limit of small slip. A generalization of Tanner’s law to account for the effect of mass transfer is derived and the behaviour of the drop close to extinction is analysed. We find a criterion for when the contact-set radius close to extinction evolves as the square-root of the time remaining until extinction — the famous $d^2$-law. However, for a sufficiently large rate of evaporation, our analysis predicts that a $d^{13/7}$-law should be more appropriate. Our asymptotic results are validated by comparison with numerical simulations.

9:31AM A33.00008 Evaporation of a drop on a periodic solid substrate with moving contact-line, AMIRHOSSEIN AMINI, Mechanical Engineering UW, GEORGE. M. HOMSY, Mechanical Engineering — Experiments on evaporating droplets on structured surfaces have shown that the contact line does not move with constant speed, but rather in a step-like, “stick-slip” fashion. As a first step in understanding this phenomenon, we study the evolution of a 2D incompressible Newtonian droplet evaporating on a non-flat substrate. We deploy a standard one-sided model which, together with the lubrication approximation, results in an evolution equation for the local height of the droplet. This is solved numerically for the case of constant fixed contact angle by utilizing numerical slip. We present results for sinusoidal substrates of different dimensionless amplitudes and wavelengths, with the droplet profile, evaporative mass flux and contact-line position evaluated for each case. In contrast with our previous results for a flat substrate, for which the contact line recedes in a nearly constant speed, we observe that the contact line speed and position show significant time variation, with the contact line moving approximately in a step-like fashion for relatively steep substrates.

9:44AM A33.00009 Evaporation dynamics of non-spherical sessile drops of pure fluids and binary mixtures1 PEDRO J. SAENZ, OMAR K. MATAR, Imperial College London, KHELLIL SEFIANE, PRASHANT VALLURI, The University of Edinburgh, JUNgho Kim, The University of Maryland — The dynamics of pure axisymmetric volatile sessile droplets have been meticulously examined over the last four decades but remain poorly understood. Studies focusing on more realistic non-spherical configurations are virtually non-existent. The dynamics of the latter are examined in this investigation by means of experiments and numerical simulations. We show that the lifetime and bulk flow characteristics of these drops depend on their size and shape. The irregular geometries lead to the emergence preferential convection currents in the liquid as well as differential local evaporation rates noticeable along the contact line. Similarly, we inspect the thermocapillary stability of the flow, which results as the liquid volatility increases, and find that this is also affected by the non-uniform wettabillity along the triple line. The Marangoni-driven instabilities grow in an intricate spatio-temporal fashion leading to the emergence of different flow regimes. Finally, we also provide new insights into the evaporation process of binary-mixture drops.

1 Memphis Multiphase (EPSRC EP/K003976/1) & ThermoPOWER (EU IRSES-PIRSES GA-2011-294905)

Sunday, November 22, 2015 8:00AM - 9:57AM —
Session A35 Drops: Impact and Splashing Ballroom B - Arnout Boelens, University of Chicago

8:00AM A35.00001 Computational comparison of high and low viscosity micro-scale droplets splashing on a dry surface, ARNOTT BOELENS, Institute for Molecular Engineering, University of Chicago, ANDRZEJ LATKA, Department of Physics, University of Chicago, Juan De Pablo, Institute for Molecular Engineering, University of Chicago — Depending on viscosity, a droplet splashing on a dry surface can splash immediately upon impact, a so called prompt splash, or after initially spreading on the surface, a late splash. One of the open questions in splashing is whether the mechanism behind both kinds of splashing is the same or not. Simulation results are presented comparing splashing of low viscosity ethanol with high viscosity silicone oil in air. The droplets are several hundred microns large. The simulations are 2D, and are performed using a Volume Of Fluid approach with a Finite Volume technique. The contact line is described using the Generalized Navier Boundary Condition. Both the gas phase and the liquid phase are assumed to be incompressible. The results of the simulations show good agreement with experiments. Observations that are reproduced include the effect of reduced ambient pressure suppressing splashing, and the details of liquid sheet formation and breakup. While the liquid sheet ejected in an early splash breaks up at its far edge, the liquid sheet ejected in a late splash breaks up close to the droplet.

8:13AM A35.00002 The role of substrate wetting and drop shape in splashing dynamics, ANDRZEJ LATKA, SIDNEY NAGEL, James Franck Institute, University of Chicago, Chicago, Illinois 60637, USA — The impact of a liquid drop on a solid surface yields a beautiful splash via an intricate interaction of the liquid, the substrate and, most surprisingly, the surrounding air. Varying the liquid’s viscosity or surface tension and the substrate’s roughness or elasticity results in strikingly different splash morphologies. While one might also have expected the wetting properties to affect splashing, we show here that changing the substrate from fully wetting to non-wetting does not significantly alter the splashing behaviour. We also investigate how the drop’s evolving shape influences the dynamics. When the drop first contacts the surface, there is a region of high negative curvature; the bulk of the descending liquid must feed the sheet rapidly spreading over the substrate by flowing around this concavity. After the drop has spread sufficiently, this concavity disappears and the flows within the drop change their shape. We find that the effect of air on splashing is significantly stronger in the initial regime. This dependence sheds light on low-velocity drop impacts.

8:26AM A35.00003 Time-resolved interference imaging of the air disc under an impacting drop, E. Q. Li, S. T. Thordodsson, King Abdullah University of Science and Technology — Water drop impacting on dry, solid surface, is rapidly decelerated by an air cushion. This thin air layer is formed by lubrication pressure in the gas, which is strong enough to stop the inertia of the drop liquid and deform its bottom tip. The contact of the drop with the solid therefore occurs along a ring, entrapping a central bubble. For very large impact velocities the lubrication pressure becomes large enough to compress the gas. We use the Kirana ultra-high-speed video camera and 50 ns pulsed laser-diodes for interferometric imaging, at time-resolution of 200 ns. We capture the evolution of the air-layer thickness profile over the entire bubble entrainment process. The maximum diameter of the air disc is in perfect agreement with earlier theoretical models, if one uses the bottom radius of curvature of the drop. The air-layer thickness is also in agreement with available theoretical models, if one assumes adiabatic compression of the gas. For the largest impact velocities the air is compressed by more than a factor of 10. Immediately after first contact, the air disc expands rapidly in the vertical. The outer edge of the air-disc forms a kink in the free surface. This kink can move radially outwards just before contact, at speed as large as 50 times the impact velocity.
We study the early-time fluid mechanical phenomena of the splash of a liquid drop on a solid surface, focusing on the dynamics before contact through the intervening air layer. Previous theoretical work (e.g. Mani, Mandre and Brenner [Journal of Fluid Mech., (2010), vol. 647, pp. 163185]) on this problem neglected viscous effects in the liquid. However, a set of recent experiments show definitively that even at early times viscous effects in the liquid are important, and in particular have the ability to dramatically change the shape of the interface before contact.

The simulations couple liquid flow via the gas layer with nonsteady Stokes flow in the liquid, and surface tension at the liquid-air interface.

8:52AM A35.00005 Drop impact on flowing liquid films: asymmetric splashing1, RENAD ISMAIL, ZHIHHAO CHE, Imperial College London, LAUREN ROTKOVITZ, MIT, IDRIS ADEBAYO, OMAR MATAR, Imperial College London — The splashing of droplets on flowing liquid films is studied experimentally using high-speed photography. The flowing liquid films are generated on an inclined substrate. The flow rate of the liquid film, the inclination angle, and the droplet speed are controlled and their effects on the splashing process studied. Due to the flow in the liquid film and the oblique impact direction, the splashing process is asymmetric. The propagation of the asymmetric crown and the generation of secondary droplets on the rim of the crown are analysed through image processing. The results show that the flow in the liquid films significantly affects the propagation of the liquid crown and the generation of secondary droplets.

1EPSRC Programme Grant, MEMPHIS, EP/K0039761/1

9:05AM A35.00006 Droplet Impact onto an Immiscible, Floating Oil Layer: Splash Behavior and Droplet Sizes1, DAVID MURPHY, CHENG LI, VINCENT D’ALBIGNAC, DAVID MORRA, JOSEPH KATZ, Johns Hopkins University — The high speed impact of a raindrop on a fluid surface at We0 = µu/d/σ = 2000 affects environmental processes like marine aerosol production. High speed imaging shows that a floating immiscible oil layer, such as a crude oil slick, modifies the splash behavior. Tests performed for a wide range of layer thicknesses (h), viscosities, and surface and interfacial tensions facilitate behavioral categorization in terms of We0 = µu/h/σ, and ReFr = µu/d/µgh, where h and d subscripts refer to layer and droplet properties, respectively. Included are multi-layer/level crowns, and due to the high Oh = µ/(µ)^{1/2} of oil, formation of an intact ejecta sheet within 50 µs after impact, which subsequently ruptures to form aerosolized oil droplets. High speed holographic microscopy provides the size and spatial distributions of airborne droplets, which are bimodal with peaks at 50 and 225 µm. Small droplets (50 µm) are ejected primarily at shallow angles and remain at low elevation by microligament breakup within the first 50 µs of impact. Larger droplets (225 µm) are ejected at a steeper angle and produced later by breakup of larger ligaments protruding vertically from the splash crown. Small droplet frequency at high elevation increases when crude oil is introduced, mostly as satellite droplets resulting from the large ligament breakup.

1Funding provided by the Gulf of Mexico Research Initiative

9:18AM A35.00007 Crown-splash by a cylinder impact, JALIL HASANYAN, SEAN GART, SUNGHWAN JUNG, Virginia Tech — The impact of a droplet onto a liquid bath creates a crown splash of the thin liquid sheet. Similarly, we can observe a crown-splash phenomenon when a rigid hydrophilic cylinder impacts a liquid bath. After the cylinder impacts the air/liquid interface, the liquid sheet splashes upwards and creates a crown-forming instability. However, unlike the drop-generated splash, the solid-generated splash does not expand radially, but stays on the side of the cylinder. In this present study, we examined the vertical splash depending on the cylinder size, impact speed, and liquid properties. Also, the instability of the leading edge of the splash is characterized and compared with capillary instability theories.

9:31AM A35.00008 Numerical simulation of drop impact on a thin film: the origin of the droplets in the splashing regime1, ZHIHUA XIE, ZHIHHAO CHE, RENAD ISMAIL, CHRIS PAIN, OMAR MATAR, Imperial College London — Drop impact on a liquid layer is a feature of numerous multiphase flow problems, and has been the subject of numerous theoretical, experimental and numerical investigations. In the splashing regime, however, little attention has been focused on the origin of the droplets that are formed during the splashing process. The objective of this study is to investigate this issue numerically in order to improve our understanding of the mechanisms underlying splashing as a function of the relevant system parameters. In contrast to the conventional two-phase flow approach, commonly used to simulate splashing, here, a three-dimensional, three-phase flow model, with adaptive, unstructured meshing, is employed to study the liquid (droplet) gas (surrounding air) liquid (thin film) system. In the cases to be presented, both liquid phases have the same fluid property, although, clearly, our method can be used in the more general case of two different liquids. Numerical results of droplet impact on a thin film are analysed to determine whether the origin of the droplets following impact corresponds to the mother drop, or the thin film, or both.

1EPSRC Programme Grant, MEMPHIS, EP/K0039761/1

9:44AM A35.00009 Numerical simulation of droplet impact on interfaces1, LYES KAHOUADJI, ZHIHHAO CHE, OMAR MATAR, Imperial College London, SEUNGWON SHIN, Hongik University, Republic of Korea, JALEL CHERGUI, DAMIR JURIC, LIMSI-CNRS — Simulations of three-dimensional droplet impact on interfaces are carried out using BLUE, a massively-parallel code based on a hybrid Front-Tracking/Level-Set algorithm for Lagrangian tracking of arbitrarily deformable phase interfaces. High resolution numerical results show fine details and features of droplet ejection, crown formation and rim instability observed under similar experimental conditions.

1EPSRC Programme Grant, MEMPHIS, EP/K0039761/1
8:00AM A36.00001 Deformation and stability of surfactant - or particle - laden drop, QUENTIN BROUSSEAU, GERARDO PRADILLO, ANDREW OBERLANDER, PETIA VLAHOVSKA, Brown University, SOFTMECH@BROWN TEAM — We present an experimental study of the behavior of a drop covered with insoluble surfactant or colloidal particles in a uniform DC electric field. Steady drop shapes, drop evolution upon application of the field, and drop relaxation after the field is turned off are observed for leaky dielectric fluids: Polybutadiene (PB), Silicon oil (PDMS), and Castor oil (CO). The surfactant is generated at the drop interface by reaction of end-functionalized hydrophobic polybutadiene (PB) and polydimethylsiloxane (PDMS). The meniscus behavior is examined under the presence of charged surfactant covered droplet, and adjusted models taking into account the presence of colloidal spheres with range of electric properties. We will discuss the complex interplay of shape deformation, surfactant elasticity, particle redistribution, and interfacial charging in droplet electrohydrodynamics. Our results are important for understanding electrorheology of emulsions commonly found in the petroleum industry. We acknowledge grant NSF CBET 1437545 for funding.

However, meniscus evolution from a rounded shape to a cone was a long-standing puzzle as it overlaps with spontaneous fluid ejection. We conducted experiments on deionized (DI) water, DI-water with 0.1M KCl, polyethylene glycol, polymer solution simulating human saliva, lubricant with 0.02wt% graphene. Experiments on DI water under microgravity in International Space Station enabled us to extend the measured cone lengths from 0.5 mm (Earth) to 5 cm. The meniscus evolution to a cone was found to exhibit a universal self-similarity scaled by the fluid surface tension and density and strikingly insensitive to the forcing field while a 50% increase in applied voltage shortens the overall time for the meniscus to decay. The quasi-steady narrow size distribution of emulsified water droplets with diameters close to 1 μm was formed after a few minutes. The generated emulsion was confined near the needle electrode due to the dielectrophoretic force. The emulsion had a well-defined boundary with a shape resembling a pendant drop suspended on the pin electrode.

8:13AM A36.00002 Water-in-oil emulsification in a non-uniform alternating electric field, SUHWAN CHOI, ALEXEI SAVELEV, North Carolina State University — The emulsification of a water microdroplet placed in castor oil was performed using a non-uniform alternating electric field formed in the pin-to-plate geometry. A non-uniform electric field of ~40 kV/mm alternating with a frequency of 0.67 kHz first pin electrode. The applied frequency exceeded charge relaxation frequency of castor oil (0.3 Hz) and was below charge relaxation frequency of deionized water (7.8 kHz) used in the experiments. The emulsification process was captured with a CCD camera. The emulsification process started with entrainment of the water droplet in the high electric field region near the pin electrode under the dielectrophoretic force. Upon touching the pin, the microdroplet was disintegrated in numerous channels and secondary droplets. The process continued by entrainment of secondary droplets and continuous size reduction. Three droplet breakup mechanisms were identified: drop elongation and capillary breakup, ac electrospaying of individual droplets, chain and bridge formation and decay. The quasi-steady narrow size distribution of emulsified water droplets with diameters close to 1 μm was formed after a few minutes. The generated emulsion was confined near the needle electrode due to the dielectrophoretic force. The emulsion had a well-defined boundary with a shape resembling a pendant drop suspended on the pin electrode.

8:26AM A36.00003 Interaction between electrically charged droplets in microgravity

1. MARTIN BRANDENBOURGER, HERVE CAPS, JEROME HARDOUN, GRASP, Universit de Lige, Lige, Belgium, YOUREN VITRY, Beams, Universite Libre de Bruxelles, Bruxelles, Belgium, BERNARD BOIGELOT, Department of Electrical Engineering and Computer Science, University of Lige, 4000 Lige, Belgium, STEPHANE DORBOLO, GRASP, Universit de Lige, Lige, Belgium, GRASP TEAM, BEAMS COLLABORATION — The past ten years, electrically charged droplets have been studied tremendously for their applications in industry (electrospray, electrowetting,...). However, charged droplets are also present in nature. Indeed, it has been shown that the droplets falling from thunderclouds possess an excess of electric charges. Moreover, some research groups try to use the electrical interaction between droplets in order to control the coalescence between cloud droplets and control rain generation. The common way to study this kind of system is to make hypothesis on the interaction between two charged droplets. Then, these hypothesis are extended to a system of thousands of charged droplets. Thanks to microgravity conditions, we were able to study the interaction between two electrically charged droplets. In practice, the charged droplets were propelled one in front of the other at low speed (less than 1 m/s). The droplets trajectory is studied for various charges and volumes. The repulsion between two charged droplets is correctly fitted by a simple Coulomb repulsion law. In the case of attractive interactions, we discuss the collisions observed as a function of the droplets speed, volume and electric charges.

1. Thanks to FNRS for financial support.

8:39AM A36.00004 How does electricity make liquid bristle?1

1. BORIS KHUSID, EZINWA ELELE, YUEYANG SHEN, New Jersey Institute of Technology, Newark, NJ, DONALD R. PETIT, NASA Johnson Space Center, Houston, TX — Electrified fluid forms pointed cones triggering sparks, flashes of light, and ejecting droplets. This phenomenon is encountered in lightning and utilized in a number of technologies. Taylor showed that surface tension and electric forces form a conical meniscus with a semivertex angle of 49.3°. However, meniscus evolution from a rounded shape to a cone was a long-standing puzzle as it overlaps with spontaneous fluid ejection. We developed a method to control the cone-shaped spikes just shy of droplet ejection (PRL 114, 054501, 2015). Experiments were conducted on deionized (DI) water, DI-water with 0.1M KCl, polyethylene glycol, polymer solution simulating human saliva, lubricant with 0.02wt% graphene. Experiments on DI water under microgravity in International Space Station enabled us to extend the measured cone lengths from 0.5 mm (Earth) to 5 cm. The meniscus evolution to a cone was found to exhibit a universal self-similarity scaled by the fluid surface tension and density and strikingly insensitive to the forcing field while a 50% increase in applied voltage shortens the overall time for the meniscus to rise by more than an order of magnitude. Field induced flow inside the cone offers possibilities for non-contact control of separation and mixing inside tiny droplets.

1. NASA grants NNX09AK06G and NNX13AQ53G

8:52AM A36.00005 Effect of The Viscosity Ratio on Equilibrium Shapes and Instability of Liquid Drops in Electric Field, ASGHAR ESMAEELI, Southern IL Univ-Carbondale — Electrohydrodynamics of liquid drops is currently the focus of increased attention because of its relevance in a host of processes such as micro- and bio-fluidics. In a weak electric field the drop acquires an equilibrium shape, deforming to an oblate or a prolate spheroid. However, beyond a critical electric field strength it will disintegrate through tip-streaming or bulbous breakup. The modes and mechanisms of drop disintegration has been reasonably well-studied, assuming the drop viscosity to be the same as that of the ambient. However, there are evidences that suggest the viscosity ratio can dramatically affect the dynamics, even leading to new breakup modes. The goal of this study is to provide further insight regarding this issue through computational simulations. To this end, we use a front tracking/finite difference scheme in conjunction with Taylor leaky-dielectric model to solve the governing electrohydrodynamic equations.

9:05AM A36.00006 Electrohydrodynamics-driven pattern formation of liquid drops, ALI BEHJATIAN, ASGHAR ESMAEELI, Southern IL Univ-Carbondale — Direct Numerical Simulations are performed to explore pattern formation of suspension of leaky dielectric liquid drops in uniform DC electric field. The applied electric field strength is moderate so that the drops do not disintegrate, but they go through tangible deformation. The results show that the drops form columnar structures or horizontal rafts, depending on the ratio of the dielectric properties of the drop liquid and the ambient liquid. Scaling arguments are used to characterize the time scale of column and raft formation and their breakup, when the electric field is disconnected.
9:18AM A36.00007 Enhanced fog collection with electric fields, MAHER DAMAK, SEYED REZA MAHMOUDI, KRIPA VARANASI, Massachusetts Institute of Technology — Fog harvesting is a promising source of fresh water in remote areas. However, the efficiency of current collectors, consisting in fine meshes standing perpendicularly to the wind, is dramatically low. Fog-laden flows generally have low Stokes numbers, which leads to the deviation of fog droplets in the vicinity of the mesh wires. Here, we propose to overcome this aerodynamic limitation using a combination of electric fields and specific collecting surfaces. We show that our system largely increases the fog collection efficiency. We study the trajectories of individual particles and use the results to derive a model to predict the collection efficiency of the system. We finally identify and quantify the mechanisms that can limit the collection of fog particles. The understanding of these mechanisms leads us to construct a design chart that can be used to determine the optimal design parameters that should be used in fog collection applications as a function of the field conditions.

9:31AM A36.00008 Dispersion and vaporization of a spray of electrically charged droplets in a coflowing hot gas, ANGEL PERENA, FRANCISCO HIGUERA, Universidad Politecnica de Madrid — A numerical model of a dilute spray of electrically charged liquid droplets vaporizing in a hot gas is formulated based on a Eulerian description of the gas and Lagrangian tracking of the droplets. The model is used to simulate the dispersion and vaporization of a spray in a cylindrical chamber where the droplets are axially injected with a coflow of hot gas. The effects of the initial size of the droplets and of the gas-to-liquid mass flux and inlet temperature ratios are analyzed, and the conditions under which the droplets fully vaporize in the chamber without impacting on its walls are determined. The ranges of operation where these requirements are met widen when the flow rate of liquid is split into a number of sources and injected through different orifices, which also improves the uniformity of the temperature and vapor mass fraction distributions at the outlet of the chamber. The effects of satellite droplets and of a high voltage applied between the bases of the chamber are considered.

1This work has been supported by the Spanish MINECO through projects DPI2013-47372-C02-2 and CSD2010-00010

Sunday, November 22, 2015 8:00AM - 9:44AM
Session A39 CFD: Reactive Flows I Sheraton Back Bay C - Xinyu Zhao, University of Connecticut

8:00AM A39.00001 DNS evaluation of Reynolds stress models and Generalized Langevin models using velocity-acceleration correlation, XINYU ZHAO, ALEXANDROS MATHIOUDAKIS, University of Connecticut — Velocity-acceleration correlation is used to evaluate the pressure-rate-of-strain term for Reynolds-stress based models and the drift coefficient in the generalized Langevin model. The direct numerical simulations (DNS) of a non-premixed temporally-evolving slot jet flame and a premixed temporally-evolving jet flame are used. Both flames feature moderate Reynolds numbers, as well as highly anisotropic and inhomogeneous flow environment. Good agreement is achieved between turbulent statistics obtained from velocity-acceleration correlation and those obtained directly from DNS. Different filter sizes are then applied to the DNS database to further test the feasibility of representing pressure-rate-of-strain term and the drift coefficient using velocity-acceleration correlation in experiments or large eddy simulations. Behaviors of turbulent statistics obtained from the premixed flame and those from the nonpremixed flame are analyzed. Finally, the applicability of existing generalized Langevin model coefficients to flame simulations is discussed.

1The authors acknowledge Dr. Jacqueline Chen for providing access to the DNS database and computing resources

8:13AM A39.00002 Direct Numerical Simulation of a supersonic reacting jet with thermochemical nonequilibrium, ROMAIN FIÉVET, University of Michigan, STEPHEN VOELKEL, The University of Texas at Austin, HEESEOK KOO, University of Michigan, PHILIP VARGHESE, The University of Texas at Austin, VENKAT RAMAN, University of Michigan — In flows that exhibit nonequilibrium of internal energies, the ignition and stabilization of flames can exhibit complex dependences. Common to shock-containing flows, the lack of equilibrium between vibrational and translational motion of the molecules can significantly alter the initiation of the fuel oxidation process. In this study, direct numerical simulations (DNS) of a non-premixed temporally-evolving jet flame and a premixed temporally-evolving jet flame are used. An important aspect of this work is the determination of chemical reaction rates consistent with such nonequilibrium. For this purpose, quasi-classical trajectory analysis based two-temperature reaction rates have been formulated. The nonequilibrium multi-species mixture is described using species-specific temperature, leading to an enhanced set of momentum, species, and energy equations. A jet-in-crossflow simulation is used to understand the onset of chemical reactions under such nonequilibrium conditions.

8:26AM A39.00003 Semi-implicit iterative methods for low Mach number turbulent reacting flows, JONATHAN F. MACART, MICHAEL E. MUELLER, Princeton University — A formally second-order accurate Lie splitting approach has been developed and applied to the solution of scalar transport/reaction equations for Direct Numerical Simulation (DNS) of low Mach number turbulent reacting flows. The temporal discretization errors of this scheme are analyzed and compared with a formally first-order accurate Lie splitting approach and variations on a second-order accurate monolithic preconditioned scheme, utilizing two different preconditioners: the full Jacobian of the chemical source term and its diagonal approximation. The effect of chemical mechanism size on the relative performance of these schemes is assessed with a simple one-dimensional unsteady test case. The improved stability of the full Jacobian preconditioner is found to outpace its increased cost per time step compared to the diagonal approximation, and this advantage is found to increase with mechanism size. Likewise, the Strang splitting scheme is demonstrated to achieve better performance than the other approaches due to greater stability at larger time steps, despite greater cost per step. Finally, the schemes are evaluated with a three-dimensional unsteady turbulent planar jet flame, and similar conclusions are found as for the one-dimensional test case for relative performance.

8:39AM A39.00004 Effects of High-Order Schemes and Turbulence Models on Supersonic Combustion of Liquid Jet in Cross Flow, KEN ALABI, TTC Technologies, Inc., Centereach, NY 11720 USA, FULUSO LADENIDE, Stony Brook University, Stony Brook, NY 11794-2300 USA, WENHAI LI, TTC Technologies, Inc., Centereach, NY 11720 USA — In this study, we investigate the effects of various numerical schemes, which are mostly high-order, and different turbulence models, on combustion in liquid jets in supersonic cross flow. The baseline models for two-phase (liquid/gas) flow and evaporation, which include the modeling of the primary and secondary breakup of liquid injectants and the modeling of the evaporation and fate of the spherical particles that result from the breakup models, are based on Sirignano’s approach, with Spalding-like closures. The Eulerian-Lagrangian approach is implemented. Several variations of the weighted essentially-non-oscillatory (WENO) schemes are used for spatial discretization, with different turbulence-combustion modeling approaches, including the approaches of laminar flamelets and transported mass fractions. The results show the advantages of high-order (spatial) schemes relative to the second-order approximate factorization procedure of Beam-Warming. Approaches investigated for improving the computational efficiency of the otherwise expensive two-phase flow calculations will be discussed.
8:00AM A40.00001 Immersed boundary methods for viscoelastic particulate flows
SREENATH KRISHNAN, ERIC SHAQFEH, GIANLUCA IACCARINO. Stanford Univ — Viscoelastic particulate suspensions play key roles in many energy applications. Our goal is to develop a simulation-based tool for engineering such suspensions. This study is concerned with fully resolved simulations, wherein all flow scales associated with the particle motion are resolved. The present effort is based on Immersed Boundary methods, in which the domain grids are fixed, but particles move. In this approach, the conservation of momentum equations, which include both Newtonian and non-Newtonian stresses, are solved over the entire domain including the region occupied by the particles. The particles are defined on a separate Lagrangian mesh that is free to move over an underlying Eulerian grid. The development of an immersed boundary forcing technique for moving bodies within an unstructured-mesh, massively parallel, non-Newtonian flow solver is thus developed. The flow inside the reactor chamber is not that simple. It would often develop recirculation at various locations inside the reactor due to reactor geometry, flow conditions, buoyancy effects from temperature differences and rotational effects cause by the rotating substrate. This recirculation causes hot spots and affects the overall performance of the reactor. A recirculation fluid packet experiences a longer residence time inside the reactor and, thus, it heats up to higher temperatures causing unwanted chemical reactions and decomposition. It decreases the grow rate and uniformity on the substrate. A mathematical and computational model has been developed to help identify these unwanted hot spots occurring inside the CVD reactor. The model can help identify the user parameters needed to reduce the recirculation effects and better control the flow. Flow rates, pressures, rotational speeds and temperatures can all affect the severity of the recirculation within the reactor. The model can also help assist future designs as the geometry plays a big role in controlling fluid flow. The model and the results obtained are discussed in detail.

8:52AM A39.00005 LES/FMDF of turbulent jet ignition in a rapid compression machine
ABDOULAHAD VALIDI, HAROLD SCHOCK, ELISA TOULSON, FARHAD JABERI. Michigan State University, CFD AND ENGINE RESEARCH LABS, MICHIGAN STATE UNIVERSITY COLLABORATION — Turbulent Jet Ignition (TJI) is an efficient method for initiating and controlling combustion in combustion systems, e.g. internal combustion engines. It enables combustion in ultra-rich mixtures by utilizing hot product turbulent jets emerging from a pre-chamber combustor as the ignition source for the main combustion chamber. Here, we study the TJI-assisted ignition and combustion of lean methane-air mixtures in a Rapid Compression Machine (RCM) for various flow/combustion conditions with the hybrid large eddy simulation/filtered mass density function (LES/FMDF) computational model. In the LES/FMDF model, the filtered form of compressible Navier-Stokes equations are solved with a high-order finite difference scheme for the turbulent velocity, while the Navier-Stokes equation is solved with a stochastic method to obtain the scalar (species mass fraction and temperature) field. The LES/FMDF data are used to study the physics of TJI and combustion in RCM. The results show the very complex behavior of the reacting flow and the flame structure in the pre-chamber and RCM.

9:05AM A39.00006 Computational Efficiency and Accuracy of the Two Forms of the Rate-Controlled Constrained-Equilibrium Method
FATEMEH HADI, Northeastern University, REZA SHEIKH, 61124602 — In this study, the Rate-Controlled Constrained-Equilibrium (RCCE) method in constraint potential and constraint forms have been investigated in terms of accuracy and numerical performance. The RCCE originates from the observation that chemical systems evolve based on different time scales, dividing reactions into rate-controlling and fast reactions. Each group of rate-controlling reactions imposes a slowly changing constraint on the allowed states of the system. The fast reactions react the system to the associated constrained-equilibrium state on a time scale shorter than that of constraints. The two RCCE formulations are equivalent mathematically; however, they involve different numerical procedures and thus show different computational performance. In this work, the RCCE method is applied to study methane oxygen combustion in an adiabatic, isobaric stirred reactor. The RCCE results are compared with those obtained by direct integration of detailed chemical kinetics. Both methods are shown to provide very accurate representation of the kinetics. It is also evidenced that while the constraint form involves less numerical stiffness, the constraint potential implementation results in more overall saving in computation time.

9:18AM A39.00007 Computational Study of Fluid Flow in a Rotational Chemical Vapor Deposition (CVD) Reactor
SUN WONG, YOGESH JALURIA. Rutgers Univ — In a typical Chemical Vapor Deposition (CVD) reactor, the flow of the reacting gases is one of the most important considerations that must be precisely controlled in order to obtain desired film quality. In general, the fluids enter the reactor chamber, travel over to the heated substrate area, where chemical reactions lead to deposition, and then exit the chamber. However, the flow inside the reactor chamber is not that simple. It would often develop recirculation at various locations inside the reactor due to reactor geometry, flow conditions, buoyancy effects from temperature differences and rotational effects cause by the rotating substrate. This recirculation causes hot spots and affects the overall performance of the reactor. A recirculation fluid packet experiences a longer residence time inside the reactor and, thus, it heats up to higher temperatures causing unwanted chemical reactions and decomposition. It decreases the grow rate and uniformity on the substrate. A mathematical and computational model has been developed to help identify these unwanted hot spots occurring inside the CVD reactor. The model can help identify the user parameters needed to reduce the recirculation effects and better control the flow. Flow rates, pressures, rotational speeds and temperatures can all affect the severity of the recirculation within the reactor. The model can also help assist future designs as the geometry plays a big role in controlling fluid flow. The model and the results obtained are discussed in detail.

9:31AM A39.00008 LES study of intermittency in soot formation in a model aircraft combustor
HEESEOK KOO, VENIKAT RAMAN, Univ of Michigan - Ann Arbor, MICHAEL MUELLER, Princeton University, KLAUS PETER GEIGEL, German Aerospace Center — Intermittent soot formation is one of the modeling challenges that prevent accurate predictions of soot concentration in a turbulent reacting flow. Due to the highly unsteady and irregular sooting behavior, formation of soot is acutely sensitive to the flow and gas phase history. Therefore, we need to accurately capture interactions between soot chemistry, particle dynamics, and turbulent flame as well as the turbulent reacting flow. In this study, large eddy simulation (LES) is used to understand the model sensitivity to these interactions. Hybrid methods of moment (HMM) soot model is used that accommodates detailed process of soot particle and soot precursor evolution. Gas phase chemistry uses flamelet progress variable approach with an additional enthalpy dimension to include soot radiation effect. The developed numerical model is tested on the DLR swirl combustor that emulates the rich-queen-lean (RQL) configuration using secondary oxidation air injection.

Sunday, November 22, 2015 8:00AM - 9:31AM — Session A40 CFD: Particulate Flows
Sheraton Back Bay D - Sreenath Krishnan, Stanford University

8:00AM A40.00001 Immersed boundary methods for viscoelastic particulate flows
SREENATH KRISHNAN, ERIC SHAQFEH, GIANLUCA IACCARINO. Stanford Univ — Viscoelastic particulate suspensions play key roles in many energy applications. Our goal is to develop a simulation-based tool for engineering such suspensions. This study is concerned with fully resolved simulations, wherein all flow scales associated with the particle motion are resolved. The present effort is based on Immersed Boundary methods, in which the domain grids are fixed, but particles move. In this approach, the conservation of momentum equations, which include both Newtonian and non-Newtonian stresses, are solved over the entire domain including the region occupied by the particles. The particles are defined on a separate Lagrangian mesh that is free to move over an underlying Eulerian grid. The development of an immersed boundary forcing technique for moving bodies within an unstructured-mesh, massively parallel, non-Newtonian flow solver is thus developed and the present work presents fundamental advancements that parallelization and transfer of information between the underlying fluid grid and the particle mesh. Several validation test cases will be presented including sedimentation under orthogonal shear, a key flow in drilling muds and fracking fluids.

8:13AM A40.00002 Influence of Non-homogeneous Particle Distributions on Drug Release in a Couette in vitro Dissolution Device
BALAJI JAYARAMAN, JAMES BRASSEUR. Penn State, YANXING WANG, Georgia Tech — Drug dissolution rates from powdered formulations are commonly measured in in vitro devices. Both measurements and models commonly assume perfect mixing of drug and particle within the device. In this study we analyze the potential importance of heterogeneity in particle concentration and distribution using CFD that incorporates physically accurate mathematical representations of hydrodynamic enhancement of mass transport from shear as applicable to drug dissolution in vitro. We have developed a high-fidelity computational formulation using the Lattice Boltzmann Method (LBM) with the parallel particle tracking for a polydisperse collection transported by the flow. Drug release from the small (<100 μm) Lagrangian ‘point’ particles is modeled using a mathematical framework that is built on a validated first principles ‘quasi-steady state’ approximation with correlations for shear enhancement and integrated with the coarser Eulerian LBM flow field using a subgrid formulation Our Eulerian-Lagrangian formulation takes into account spatial variations in particle ‘bulk’ concentration from polydisperse particle distributions with specified particle distribution heterogeneities. We shall discuss the primary influences of heterogeneous bulk concentrations surrounding individual particles and non-homogeneous particle distributions in an in vitro Couette flow device to quantify the relative influences of shear enhancement on drug dissolution in vitro vs. in vivo.
8:26AM A40.00003 Comparison of Strongly Coupled Diffuse and Sharp Interface Fluid-Structure Interaction Approaches for Particle-Laden Flows, FAZLOLAH MOHAGHEGH, H.S. UDAYKUMAR. The aim of this study is to find a proper method for the simulation of blood as a particulate fluid. Since the blood cell density is almost the same as plasma, the high added mass effect necessitates implementation of a strongly coupled FSI method in the numerical simulation. Therefore, three different FSI approaches are compared, two Smoothed Profile Methods (SPM) with one and two projection steps as diffuse interface approaches and the Sharp Interface Method (SIM). Stable FSI computations can be achieved by using sub-iterations within each time step, i.e. by updating the fluid and structure and their boundary conditions at each time step multiple times to reach a desired tolerance as the convergence criteria. Various cases were used to benchmark the methods, including particles motion in a channel and particles sedimentation. The results show that the number of sub-iterations plays a key role in the efficiency. While use of SPM with two projection steps has the most expensive sub-iteration process, it has the best efficiency as it requires the lowest number of sub-iterations within each time step. Moreover, the method is more stable than SIM and the SPM with one projection. SIM is faster than SPM with one projection and it has better stability.

8:39AM A40.00004 A hierarchical Cartesian method for particle-laden flows with conjugate heat transfer. GONZALO BRITO GADESCHI, MATTHIAS MEINKE, WOLFGANG SCHROEDER. The approach is validated against results available from the literature and its applicability is demonstrated it by studying canonical equation in the particle phase is discretized based on a finite volume method. The moving surface of the particles is tracked using a level-set and is especially suited for moving boundary problems. The fluid phase is solved with a thermal Lattice-Boltzmann method while the heat conduction in the particles. The algorithm is based on hierarchical Cartesian grids and is especially suited for moving boundary problems. The fluid phase is solved with a thermal Lattice-Boltzmann method while the heat equation in the particle phase is discretized based on a finite volume method. The moving surface of the particles is tracked using a level-set method. The approach is validated against results available from the literature and its applicability is demonstrated it by studying canonical problems of suspended rigid particles in channel-like flows.

8:52AM A40.00005 Direct numerical simulation of evaporation-induced particle motion. HOCHAN HWANG, GIHUN SON, Sogang Univ. — A sharp-interface level-set (LS) method is presented for direct numerical simulation (DNS) of evaporation-induced particle motion. The liquid surface is tracked by the LS function, which is defined as a signed distance from the liquid-gas interface. The conservation equations of mass, momentum, energy for the liquid and gas phases and vapor mass fraction for the gas phase are solved accurately imposing the coupled temperature and vapor fraction conditions at the evaporating liquid-gas interface. A dynamic contact angle model is also incorporated into the LS method to account for the change between advancing and receding contact angles at the liquid-gas-solid contact line. The solid surface is tracked by another LS function, which is defined as a signed distance from the fluid-solid interface. The conservation equations for multiphase flows are extended to treat the solid particle as a high-viscosity non-evaporating fluid phase. The velocity inside the solid domain is modified to enforce the rigid body motion using the translational velocity and angular velocity of the particle centroid. The DNS results demonstrate the particle accumulation near the evaporating interface and the contact line pinning and stick-slip motion near the evaporating contact line.

9:05AM A40.00006 Numerical simulation of contact line motion and particle distribution in dip coating. GIHUN SON, JAEWON LEE, Sogang University — A level-set method is presented for computing contact line motion and particle distribution in dip coating, which is a popular process for production of thin films and has received new attention as a simple particle deposition process for patterning microstructures. Assuming that the interface temperature is below the saturation temperature, we obtain the evolution equation of mass, momentum, and energy for the liquid-gas phases, the vapor mass fraction in the gas phase, and the particle concentration in the liquid phase. To consider the case where the particle concentration reaches the maximum value (in random packing), the diffusion coefficient of particles is determined from the generalized Stokes-Einstein equation. The temperature and vapor fraction at the interface and the evaporation mass flux are simultaneously determined from the coupled equations for the mass and energy balances at the interface and the thermodynamic relation. The present computations demonstrated that the plate withdrawal velocity significantly affects the liquid film formation and particle distribution pattern. In the regime of a low plate velocity, the computed liquid-gas-solid contact line reaches a quasi-steady state and the particle accumulation is pronounced near the stationary contact line.

9:18AM A40.00007 Clustering Instability in Sedimenting Gas-Solid Suspensions and its Influence on Flow Properties. XIAOQI LI, XIAOLONG YIN, Colorado School of Mines, GUODONG LIU, Harbin Institute of Technology. — It is well known that sedimentation or fluidization of solid particles through gas is unstable. Instability is usually recognized as particle clusters when the solid fraction is low, or as void ‘bubbles’ when the solid volume fraction is high. Using particle-resolved numerical simulations, we studied cluster formation in gas-solid systems with gas-to-solid density ratio being 0.01 and 0.001. The particles are uniformly distributed with a terminal velocity. The solid fraction is 0.25. Up to 4000 particles were used such that the clustering phenomena can be adequately examined. In periodic computational domains whose lateral dimension is about eight particle diameters, nucleated particle clusters quickly coalesce and grow into traveling waves that span the entire width of the domain. Consequently, gas-solid drag is significantly increased compared to that in a homogeneous liquid-solid suspension, the lateral velocity variance is suppressed, and the particle velocity distributions are strongly non-Gaussian. When lateral dimension is increased to about thirty particle diameters, particle clusters never turn into width-spanning traveling waves. As results, the drag is similar to that in a homogeneous suspension, the lateral velocity variance is strongly enhanced and the vertical variance reduced, and particle velocity distributions are nearly Gaussian. These results suggest that the effect of particle clusters should be examined in domains with large lateral dimensions.
Fish locomotion: insights from both simple and complex mechanical models

GEORGE LAUER, Harvard University — Fishes are well-known for their ability to swim and maneuver effectively through the water, and recent years have seen great progress in understanding the hydrodynamics of aquatic locomotion. But studying freely-swimming fishes is challenging due to difficulties in controlling fish behavior. Mechanical models of aquatic locomotion have many advantages over studying live animals, including the ability to manipulate and control individual structural or kinematic factors, easier measurement of forces and torques, and the ability to abstract complex animal designs into simpler components. Such simplifications, while not without their drawbacks, facilitate interpretation of how individual traits alter swimming performance and the discovery of underlying physical principles. In this presentation I will discuss the use of a variety of mechanical models for fish locomotion, ranging from simple flexing panels to complex biomimetic designs incorporating flexible, actively moved, fin rays on multiple fins. Mechanical devices have provided great insight into the dynamics of aquatic propulsion and, integrated with studies of locomotion in freely-swimming fishes, provide new insights into how fishes move through the water.
**2:10PM D1.00001** Numerical study of thermally stratified shear flows at the interface between porous and pure-fluid layers\(^1\), MILTIADIS V. PAPA-XANDRIS, PANAGIOTIS D. ANTONIADIS, Université catholique de Louvain — In this talk we are concerned with thermally stratified flows at the interface between superposed porous and pure-fluid layers. In our study we employ a thermo-mechanical model for the flows of interest that was recently developed by our team. According to this model, both the fluid and the solid matrix are treated as two separate and identifiable continua that are in thermal non-equilibrium with each other. This allows for the derivation of a single set of equations that are simultaneously valid both in the porous and pure-fluid regions. First, we formally present the basic steps of deriving the equations of the numerical model and sketch the algorithm for its numerical treatment. Then, we present and discuss numerical results for transient shear flows in the domains of interest, under both stable and unstable thermal stratification. Emphasis is placed on the effects of buoyancy to the evolution of the flow structures at the interface and on the mechanisms that induce thermal non-equilibrium inside the porous medium.

\(^1\)This work is supported by the National Fund for Scientific Research (FNRS), Belgium

**2:23PM D1.00002** Experimental Investigation of Dissolution-Driven Convection in Heterogeneous Porous Medium, RUI NI, ASHWANTH K. R. SALIBINDLA, ASHIK ULLAH MOHAMMAD MASUK, JIKANG SHEN, Department of Mechanical and Nuclear Engineering, Pennsylvania State University — Subsurface carbon sequestration in saline aquifers has emerged as one promising method to mitigate anthropogenic emission of CO\(_2\) because of the potential storage capacity of the accessible formations. Being injected into the porous formation underground, the buoyant CO\(_2\) will start to migrate upward and may eventually leak back to the surface through faults in the overlying caprock. This leaking process may be hindered or even completely stopped due to the dissolution of CO\(_2\) into the brine. For those locations, where the supercritical CO\(_2\) is above the brine, the dissolution between the two fluids leads to a mixture with higher density than both CO\(_2\) and brine; and thus the resultant solution on the interface is unstable, drawing the CO\(_2\)—brine mixture down by gravity. Previous laboratory experiments on dissolution-driven convection were mostly limited to a simplified case where the porous medium was assumed to be homogeneous. To account for the heterogeneity existing in the actual formations, we designed a series of experiments in controlled ways to introduce spatial variations of permeability. By measuring the mass transfer efficiency under different conditions, our experiments provide a new way to assess the convective mixing, triggered by gravitational instability, plays an important role in CO\(_2\) sequestration in saline aquifers. For this problem, the characteristics of the saturation event and the post saturation period over which a second onset of bifurcation occurs is investigated using discrete mode perturbation approach via high accuracy numerical simulation. We find that the critical time for the onset of nonlinear saturation scales linearly with the inverse of the Rayleigh number. The effect of initial perturbation as well as the critical wavelength at the onset of saturation is explored. In the post saturation period the flux decays as \(t^{-1}\). It is also observed that the second bifurcation is triggered beyond a critical point when the CO\(_2\) flux dips below the level associated with the corresponding unperturbed flow.

**2:36PM D1.00003** Nonlinear saturation and secondary bifurcation in gravitationally unstable boundary layers in porous medium, ZOHREH GHORBANI, AMIR RIAZ, Univ of Maryland-College Park — The convective mixing, triggered by gravitational instability, plays an important role in CO\(_2\) sequestration in saline aquifers. For this problem, the governing equations are solved at the porous scale level in both the fluid and solid phases while conserving the appropriate conjugate boundary condition at the solid-fluid interface. This allows us to calculate continuum-scale parameters such as the permeability and the stagnant thermal conductivity of the medium very accurately without using any empirical formulations. Also, the regular arrangement of the solid blocks allows us to calculate the intrinsic-averaged temperatures of each phase in the porous medium. This information is used to test the assumption of local thermal equilibrium between the fluid and solid phases. Our model is then used to probe the effect of contrasting thermal properties between the two phases. We observe that increasing the contrast in thermal conductivities leads to a departure from local thermal equilibrium between the two phases. This indeed causes a shift for the onset of convection in terms of critical Rayleigh number and a modification of the Nusselt-Rayleigh number correlation after the onset of convection.

**2:49PM D1.00004** Pore-scale study of Horton-Rogers-Lapwood convection in porous media: Effect of microscale thermophysical heterogeneity on the onset of convection, HAMID KARANI, CHRISTIAN HUBER, Georgia Inst of Tech — A direct-numerical-simulation is employed on a 2-dimensional Horton-Rogers-Lapwood convection problem in a regular porous medium. High resolution numerical simulation is performed by the thermal lattice-Boltzmann method. The governing equations are solved at the pore-scale level in both the fluid and solid phases while conserving the appropriate conjugate boundary condition at the solid-fluid interface. This allows us to calculate continuum-scale parameters such as the permeability and the stagnant thermal conductivity of the medium very accurately without using any empirical formulations. Also, the regular arrangement of the solid blocks allows us to calculate the intrinsic-averaged temperatures of each phase in the porous medium. This information is used to test the assumption of local thermal equilibrium between the fluid and solid phases. Our model is then used to probe the effect of contrasting thermal properties between the two phases. We observe that increasing the contrast in thermal conductivities leads to a departure from local thermal equilibrium between the two phases. This indeed causes a shift for the onset of convection in terms of critical Rayleigh number and a modification of the Nusselt-Rayleigh number correlation after the onset of convection.

**3:02PM D1.00005** Inclined porous medium convection at large Rayleigh number, BAOLE WEN, The University of Texas at Austin, University of New Hampshire, GREG CHINI, University of New Hampshire — DNS are performed to study pattern formation in and transport properties of high-Rayleigh-number (\(Ra\)) convection in a 2D porous layer inclined at an angle \(\phi\) to the horizontal. The results indicate that for \(0 < \phi < 25^\circ\), the flow exhibits a similar 3-region structure as is manifest in the horizontal case, except that as \(\phi\) is increased the time-mean spacing between neighboring interior ‘mega-plumes’ is also substantially increased. Nevertheless, for \(0 < \phi < 20^\circ\), the Nusselt number \(Nu\) is almost unchanged. However, when \(\phi > \phi_c\), where \(30^\circ < \phi_c < 32^\circ\), the columnar flow structure is completely broken down: the flow transitions to a large-scale traveling-wave convective roll state, and the heat transport is significantly reduced. To better understand the physics of high-\(Ra\) porous medium convection at small \(\phi\), a spatial Floquet analysis is performed, yielding predictions of the linear stability of numerically-computed, fully nonlinear steady convective states. Our results show that the background flow induced by the inclination of the layer intensifies the bulk instability during its subsequent nonlinear evolution, thereby favoring increased spacing between the interior plumes relative to the horizontal scenario.

**3:15PM D1.00006** On Permeability Dynamics in Carbonaceous Aquifers used in Energy Storage Applications\(^1\), DS BRADY, BS TILLEY, Worcester Polytechnic Institute, M UECKERT, T BAUMANN, Technische Universitaet Muenchen — Geothermal energy harvesting applications use deep groundwater aquifers to store harvested energy. The impact of this additional energy to the aquifer chemistry is crucial for long-term operation. Gaseous CO\(_2\) is added to the injected water to compensate potential precipitates of carbonates and to prevent structural changes to the aquifer. Both of these effects affect the local chemical equilibrium of the aquifer, and we consider an effective model through homogenization which captures the hydrochemistry, heat transfer, fluid flow and permeability dynamics of the aquifer as heated fluid is added to the aquifer (injection), and as it is later removed (production). The impact of these different physical mechanisms of the heat storage performance of the aquifer is discussed.

\(^1\)Support from the Bavarian State Ministry for the Economy is gratefully acknowledged
3:28PM D1.00007 Experimental study of pattern formation during carbon dioxide mineralization. GABOR SCHUSZTER, FABIAN BRAU, ANNE DE WIT, Universite Libre de Brussels — Injection of supercritical carbon dioxide in deep porous aquifers, where mineral carbonation takes place via chemical reactions, is one of the possible long-term storage of this greenhouse gas. This mineralization process is investigated experimentally under controlled conditions in a confined horizontal Hele-Shaw geometry where an aqueous solution of sodium carbonate is injected radially into a solution of calcium chloride. Precipitation of calcium carbonate in various finger, flower or tube-like patterns is observed in the mixing zone between the two solutions. These precipitation structures and their growth dynamics are studied quantitatively as a function of the parameters of the problem, which are the injection rate and the reactant concentrations. In particular, we show the existence of critical concentrations of reactants above which the amount of the calcium carbonate precipitate produced drops significantly.

3:41PM D1.00008 Modeling the Dynamics of Remobilized CO2 within the Geologic Subsurface1. ERIK HUBER, DONALD KOCH, ABRAMAH STROOCK, Cornell University — Long after CO2 is injected into a brine aquifer, most reservoir-scale fluid dynamic simulations predict large fractions of the original plume will become immobilized via capillary trapping and dispersed throughout the formation. We begin our analysis with a reservoir in this state and consider the effects caused by a depressurization of the domain (e.g. from a nearby production well or newly formed fracture between neighboring reservoirs). Using supercritical CO2 density data from NIST and an assumed knowledge of the minimum residual saturation of CO2, we demonstrate that even a large decrease in reservoir pressure is likely to only result in a small mass fraction of remobilized CO2. Once mobile, this volume of CO2 will rise in the reservoir and concentrate beneath the caprock of the domain. We show that a model of relative permeability that takes account of insights from percolation theory near the minimum CO2 saturation leads to much more rapid rise of remobilized CO2 than a traditional empirical correlation such as the Brooks-Corey model.

3:54PM D1.00009 Micro-PIV Study of Supercritical CO2-Water Interactions in Porous Micromodels. FARZAN KAZEMIFAR, GIANLUCA BLOIS, KENNETH T. CHISTENSEN, University of Notre Dame — Multiphase flow of immiscible fluids in porous media is encountered in numerous natural systems and engineering applications such as enhanced oil recovery (EOR) and CO2 sequestration among others. Geothermal sequestration of CO2 in saline aquifers has emerged as a viable option for reducing CO2 emissions, and thus it has been the subject of numerous studies in recent years. A key objective is improving the accuracy of numerical models used for field-scale simulations by incorporation/better representation of the pore-scale flow physics. This necessitates experimental data for developing, testing and validating such models. We have studied drainage and imbibition processes in a homogeneous, two-dimensional porous micromodel with CO2 and water at reservoir-relevant conditions. Microscopic particle image velocimetry (micro-PIV) technique was applied to obtain spatially- and temporally-resolved velocity vector fields in the aqueous phase. The results provide new insight into the flow processes at the pore scale.

4:07PM D1.00010 Micro-PIV measurements of multiphase flow of water and liquid CO2 in 2D homogeneous and heterogeneous porous micromodels. YAOFA LI, FARZAN KAZEMIFAR, GIANLUCA BLOIS, KENNETH CHRISTENSEN, University of Notre Dame — Geological sequestration of carbon dioxide (CO2) has been of great interest primarily for the reason of CO2 emission reduction and enhanced oil recovery. Yet, our fundamental understanding of the coupled flow dynamics of CO2 and water in geologic media still remains limited, especially at the pore scale. Therefore, in this work the pore-scale flow of water and liquid/supercritical CO2 are quantified in 2D homogeneous and heterogeneous porous micro-models under reservoir-relevant conditions. Fluorescent microscopy and the micro-PIV technique are employed to simultaneously visualize both phases and obtain the velocity field in the aqueous phase. The velocity measurements in the homogeneous micro-model illustrate active and passive flow pathways and circulation regions near the fluid-fluid interfaces induced by shear. Moreover, the results for heterogeneous micro-models are presented and compared with those for homogeneous micro-models, which give valuable insight into flow processes at the pore scale in natural rock.

Sunday, November 22, 2015 2:10PM - 4:20PM — Session D2 Suspensions: Migration and Mixing 101 - Roger Bonnecaze, University of Texas at Austin

2:10PM D2.00001 How to Magnetically Generate Flows in Dead-Ends with Dilute Suspensions of Iron Particles. ROGER BONNECAZE, MICHAEL CLEMENTS, University of Texas at Austin — Dilute suspensions of iron particles in the presence of a magnetic field can create flows in dead-ends of pores, channels and even blocked arteries to help dissolve clots. Observations show that added iron particles in a rotating magnetic field form rotating rods along the wall of the blocked channel, creating a convective flow. We present a proposed mechanism for this magnetically driven flow in the form of coupled particle-scale and channel-scale flow models. At the particle-scale, particles chain up to lengths balancing magnetic and hydrodynamic forces on the resulting rods. The weak gradient of the magnetic field causes the rods to accumulate on one side of the channel. The rods rotate due to the rotating magnetic field, provided the field strength is high enough, which creates a localized body couple in the flow that drives a macroscopic convective flow in the channel. Coupled transport equations for the particles and the suspension as a whole are presented. The model equations are solved asymptotically and numerically and compared to experimental observations. Design rules for implementation of this technique are presented to optimize the flow.

2:23PM D2.00002 Three-dimensional collision of general-shaped particles in a viscous fluid. MOHSEN DAGOOGHI, IMAN BORAZJANI, State Univ of NY - Buffalo — The hydrodynamic interactions between rigid particles in a viscous fluid in semi-dilute and dense suspensions require a collision strategy to detect and prevent near collision and overlapping between particles in numerical simulations. While various collision models have been developed for spherical particles, very limited models are currently available for complex-shaped particles. In earlier methods, a repulsive force is applied to the particles when their distance is less than a threshold value and, depending on the magnitude of this repulsive force, collision may not be prevented or particles may bounce unrealistically. We have developed a three-dimensional numerical technique for general-shaped particles that: (1) detects near collision of complex-shaped objects in contrast to straightforward detection of spherical particles; and (2) guarantees overlap prevention. Contrary to conventional methods, we used an iterative method to exert a sufficient force and/or moment on each particle to prevent particles from overlapping without causing them to bounce back as expected at low Stokes numbers.
2:36PM D2.00003 Transport of particle-laden viscoelastic suspensions: tuning particle behavior with elasticity and geometry, ALEXANDER BARBATI, Massachusetts Inst of Tech-MIT, AGATHE ROBISSON, ELIZABETH DUSSAN V., Schlumberger-Doll Research Center, GARETH MCKINLEY, Massachusetts Inst of Tech-MIT — The transport of particle-laden viscoelastic suspensions is routine in several industrial and natural systems. Many applications, such as hydraulic fracturing in the oilfield, require the successful (and occasionally simultaneous) flow and placement or rigid particles, commonly known as proppant. Hydraulically-generated fractures are routinely less than 6 particle diameters in width. We investigate the flow of viscoelastic particle-laden suspensions in microfabricated geometries mimicking hydraulically-generated fractures under a variety of dynamic conditions to illustrate the interaction between inertia, elasticity, and geometrical parameter of particle behavior during flow. We characterize the flow in these model geometries with a combination of streakline imaging, particle image velocimetry, and direct imaging of model proppant particles embedded in the flow. We accompany these small-scale measurements with macro-scale interrogation of fluid rheology by measuring material functions of the working fluid in under shear and extension. These material functions are used in concert with imposed flow conditions and imaging results to identify dominant transport mechanisms on the channel and particle scale, which indicate overall system behavior.

2:49PM D2.00004 Viscous resuspension in pressure driven confined flows of suspensions, ANAS MACHADO, HUGUES BODIGUEL, ANNIE COLIN, Univ of Bordeaux — Flows of non-Brownian semi-dilute suspensions are mainly governed by the spatial repartition of the particles. At low Reynolds numbers, it is indeed generally non-uniform due to the cross-stream migration towards low sheared regions of the flow. Though this phenomenon has been has the focus of many work for several decades, discrepancies still exist between experiments and modelling, and in particular for pressure driven confined flows which are the focus of this work. In order to quantify shear induced migration, we take advantage horizontal flow of buoyant particles in slits, where viscous resuspension is balanced by buoyancy. We study PMMA rigid spheres of 6 µm dispersed several liquids of various density, and impose pressure driven flows in slits of several tens of µm. Using advanced particle imaging velocimetry techniques and confocal microscopy, we measure systematically both volume fraction and velocity profiles. At low flow rates, the particle density is highly asymmetric due to buoyancy whereas it becomes symmetric at high flow rate, due to shear-induced migration. The transition occurs for a given Shields number which we characterize as a function of concentration and confinement. The results are analyzed and discussed in the framework.

3:02PM D2.00005 Particle dispersion in non-stationary and non-uniform suspension flows, AMANDA HOWARD, MARTIN MAXEY, Brown University, KYONGMIN YEO, IBM Watson Research Center — In a low Reynolds number pressure driven flow of neutrally buoyant, non-Brownian particles in suspension there is usually an irreversible dispersion of the particles and near the central core of the flow in a region of low concentration near the walls. Surface roughness and the resulting near-contact forces between particles have been shown to the leading source of irreversibility in suspension shear flows. We report on a series of numerical simulations of particle suspensions in a planar channel for developing flows under both steady and oscillating pressure gradients and for different non-uniform particle distributions. We observe a correlation between the particle pressure associated with contact forces and the development of particle fluxes. In low shear zones, there may be a high number of near-contacts or high coordination number but limited particle pressure. We relate the results to our recent study of suspension plugs where there is a sharp change in particle concentration across a front in the streamwise direction and in oscillatory flow gives a local flux towards the walls. We consider the results in the context of stress balance models and mechanisms generating net particle transport.

3:15PM D2.00006 Radial distribution of neutrally buoyant spherical particles suspended in Poiseuille flow, YUSUKE MORITA, TOMOAKI ITANO, MASAKO SUGIHARA-SEKI, Kansai Univ. — An experimental study of the inertial migration of neutrally buoyant spherical particles suspended in the Poiseuille flow through circular tubes has been conducted at Reynolds numbers (Re) from 100 to 1000. The distributions of particles at downstream cross-sections were measured and the probability density function (PDF) of particles was calculated as a function of the radial position. At relatively high Re, the PDF was found to have two peaks, corresponding to the so-called Segre-Silberberg annulus and the inner annulus, whereas at low Re only the Segre-Silberberg annulus was present. As the measurement sites got downstream, the fraction of the particles observed on the inner annulus decreased and its radial position moved outward towards the Segre-Silberberg annulus. These results suggest that, if the tubes were long enough, the inner annulus would disappear, so that only the Segre-Silberberg annulus would be present.

3:28PM D2.00007 The motion of a light particle in a rotating Stokes flow, TOM MULLIN, TANIA SAUMA PEREZ, YANG LI, Univ of Manchester — We present the results of experimental investigations into the motion of light spheres in a rotating horizontal drum filled with viscous fluid. Stokesian dynamics calculations indicate a single stable fixed point on the centreline of the annulus, with an unstable fixed point on the inner wall of the annulus. The stable fixed point is in the so-called Segre-Silberberg annulus and the inner annulus, whereas at low Re only the Segre-Silberberg annulus was present. As the measurement sites got downstream, the fraction of the particles observed on the inner annulus decreased and its radial position moved outward towards the Segre-Silberberg annulus. These results suggest that, if the tubes were long enough, the inner annulus would disappear, so that only the Segre-Silberberg annulus would be present.

3:41PM D2.00008 Mixing in sheared suspensions, MATHIEU SOUZY, NORA CHERIBA ABID, IUSTI, EMMANUEL VILLERMAUX, IRPHE, BLOEN METZGER, IUSTI — Mixing occurs spontaneously in sheared suspensions, even at low Reynolds number. Under flow, successive collisions between particles deviate the laminar streamlines, and thus induce disturbances in the fluid phase, which produce very efficient mixing. We measure fluid velocity fields by performing high spatial resolution PIV experiments within a sheared suspension, and present numerically advanced particle image velocimetry (PIV) methods. The deformation statistics are found to be well modeled by a Langevin equation with multiplicative noise, which can be coupled with diffusion to infer the probability density function of the concentration in the medium.

3:54PM D2.00009 Suspension Dynamics of Liquefied Lignocellulosic Biomass in Pipeflow using Echo Particle Image Velocimetry, NICHOLAS DEMARCHI, CHRISTOPHER WHITE, University of New Hampshire — Echo particle image velocimetry (EPIV) is used to acquire planar fields of velocity in pipeflow of liquefied biomass. The biomass used is acid washed corn stover liquefied by enzymatic hydrolysis. The liquefaction process produces a complex multiphasic fluid suspension with a microstructure consisting of insoluble solid particles dispersed within a continuous liquid phase. The solid particles are generally heavier than the liquid phase, non-spherical, and distributed over a wide range of aspect ratios and sizes. Batches of liquefied biomass are produced at incremental mass loadings doubling from 1.5% to 12%. The rheology, microstructure, and solid particle settling velocities of the liquefied biomass as a function of mass loading is first quantified. Next, EPIV is used to measure and quantify the flow dynamics of liquefied biomass suspensions under laminar pressure driven pipeflow conditions. Finally, information gathered from the experimental data is used to simulate particle settling rates and predict the particle physics under the same pipeflow conditions.
4:07PM D2.00010 Flow of a suspension over an obstacle: revisiting an old problem in a new context, JEFFREY MORRIS, Levich Institute of City College of New York, HAMED HADDADI, University of California Los Angeles, SHAHAB SHOJAEI-ZADEH, Rutgers University, EVIN CONNINGTON, Stevens Institute of Technology — The flow of a fluid over an obstacle, with the associated separation of streamlines and recirculating wake, is a classical fluid mechanical phenomena that has been instrumental in development of our understanding of the interaction of viscous and inertial effects in simple fluids. Visualizations of these behaviors serve as benchmark observations. However, replacing the pure fluid with even a simple multiphase material, a suspension immersed in a Newtonian fluid, poses new questions in understanding the physics in these phenomena. In experimental observation of a dilute suspension flow over bluff bodies in a microfluidic device, we have observed formation of a depleted zone in the recirculating wake region. Using numerical simulations, it has been deduced that rigid spherical particles with finite size released inside the wake region migrate towards the wake boundaries, forming a limit cycle. The tendency of particles to outward motion leads to formation of the depleted region. In the present work, we probe the limit cycle phenomena and other aspects of the suspension flow over obstacles, such as average particle velocities and velocity fluctuations and force on the obstacle using detailed lattice-Boltzmann simulations and microfluidic experiments.

Sunday, November 22, 2015 2:10PM - 4:20PM –
Session D3 Particle-Laden Flows: Density Effects 102 - Luca Brandt, Linne FLOW Center, KTH Mechanics

2:10PM D3.00001 Sedimentation of finite-size particles in quiescent and turbulent environments1, LUCA BRANDT, WALTER FORNARI, Linne FLOW Centre, KTH Mechanics, Stockholm, Sweden, FRANCESCO PICANO, Department of Industrial Engineering, University of Padova, 35131 Padua, Italy — Sedimentation of a dispersed solid phase is widely encountered in applications and environmental flows. We present Direct Numerical Simulations of sedimentation in quiescent and turbulent environments using an Immersed Boundary Method to study the behavior of finite-size particles in homogeneous isotropic turbulence. The particle radius is approximately 6 Komlogorov length-scales, the volume fraction \(0.5\%\) and \(1\%\) and the density ratio \(1.02\). The results show that the mean settling velocity is lower in an already turbulent flow than in a quiescent fluid. The reduction with respect to a single particle in a quiescent fluid is about \(12\%\) in dilute conditions. The probability density function of the particle velocity is almost Gaussian in a turbulent flow, whereas it displays large positive tails in quiescent fluid. These tails are associated to the intermittent fast sedimentation of particle pairs in drafting-kissing-tumbling motions. Using the concept of mean relative velocity we estimate the mean drag coefficient from empirical formulas and show that non stationary effects, related to vortex shedding, explain the increased reduction in mean settling velocity in a turbulent environment.

1This work was supported by the European Research Council Grant No. ERC-2013- CoG-616186, TRITOS

2:23PM D3.00002 DNS Study of Particle-Bed-Turbulence Interactions in an Oscillatory Wall-Bounded Flow1, CHAITANYA GHODKE, SOURABH APTE, Oregon State University — Particle-resolved direct numerical simulations are performed to investigate the effects of an oscillatory flow field over a rough-bed, corresponding to the experimental setup of Keiller & Sleath (1976) for transitional and turbulent flows over a range of Reynolds numbers (95-400) based on the Stokes-layer thickness. It is shown that the near-bed turbulence, distorts and breaks the streamwise horse-shoe structures, and reduces the large-scale anisotropy. A double-averaging of the flow field reveals spatial inhomogeneities at the roughness scale and alternate paths of energy transport in TKE budget. The unsteady nature of hydrodynamic forces on particles and their cross-correlations with measurable flow variables are also studied. Temporal correlations showed drag and lift to be positively correlated with a phase difference, which is approximately equal to the Taylor micro-scale related to drag/lift correlations. Spatio-temporal correlations between the flow field and particle force showed that the lift force is well correlated with the streamwise velocity fluctuations up to distances of the same order as the particle diameter, whereas pressure fluctuations are correlated and anti-correlated with the lift force in the front and aft regions of the particle.

1Supported by NSF # 1133363

2:36PM D3.00003 Similarity between particles and bubbles as micro-additives in turbulent channel flow1, YOICHI MITO, Kitami Institute of Technology — The acceleration of turbulent fluid flow in a vertical channel has been examined by using a direct numerical simulation to calculate the fluid velocities seen by the flows. The flows considered are the downward gas flow to which solid particles of density ratio of \(10^3\) are added and the upward liquid flow to which bubbles of density ratio of \(10^{-3}\) are added. Both additives, ranging in volume fraction up to \(2 \times 10^{-3}\), are represented as solid spheres. The Froude numbers are chosen so as to have similar effects in both flows by the use of the same volume fraction of the additives. The fluid-phase momentum balance, integrated over the domain, is used to examine the changes in drag, wall friction and averaged feedback force of the non-stationary flow models. The feedback force per volume fraction is unchanged in the bubble flow. It decreases with increasing volume fraction and inertia of particles in the particle flow. Similarities between the two disperse flows are seen at small times for small volume fractions. Drag is reduced by both additives. The amount of reduced drag decreases with time at large times in the bubble flow, due to the increases in the accumulation of bubbles above walls.

1This work was supported by JSPS KAKENHI Grant Number 26420097.

2:49PM D3.00004 DNS with Discrete Element Modeling of Suspended Sediment Particles in an Open Channel Flow1, PEDRAM PAKSEREHT, SOURABH APTE, Oregon State University, JUSTIN FINN, University of Liverpool, UK — Interactions of glass particles in water in a turbulent open channel flow over a smooth bed with gravity perpendicular to the mean flow is examined using direct numerical simulation (DNS) together with Lagrangian Discrete-Element-Model (DEM) for particles. The turbulent Reynolds number (\(Re_t\)) is 710 corresponding to the experimental observations of Righetti & Romano (JFM, 2004). Particles of size 100 μm, with volume loading \(3\%\), are simulated using four-way coupling with standard models for drag, added mass, lift, pressure, and inter-particle collision forces. The presence of particles affect the outer as well as inner region of the wall layer where particle inertia and concentration are higher. The DNS-DEM is able to capture the fluid-particle interactions in the outer layer accurately. However, in the inner layer, an increase in mean as well as rms fluid velocity, as observed in the experiments, is not predicted by the DNS-DEM model. It is conjectured that particles slide and roll on the bottom wall, creating slip-like condition. Predictions using different models for drag and lift forces, as well as strong torque coupling are explored and compared with experimental data.

1Funding: NSF project #1133363, Sediment-Bed-Turbulence Coupling in Oscillatory Flows.
3:02PM D3.00005 Entrainment in sediment laden flows

3:15PM D3.00006 Resuspension of a granular bed by thermal convection

3:28PM D3.00007 Study of interactions between sediment particles in sheet flow using CFD-DEM

3:41PM D3.00008 Simultaneous Measurement of Fluid and Particle Motion in Shear Induced Erosion

3:54PM D3.00009 Effect of particle size distribution on the hydrodynamics of dense CFB risers

4:07PM D3.00010 Particle-driven gravity currents in non-rectangular cross-section channels

Sunday, November 22, 2015 2:10PM - 4:20PM – Session D4 Compressible Flows: Shocks 103 - Lawrence Ukeiley Ukeiley, University of Florida
2:10PM D4.00001 Numerical and Experimental Investigation of Oblique Shock Wave Reflection from a Water Wedge, QIAN WAN, HONGJOO JEON, VERONICA ELIASSON, Univ of Southern California — Shock wave interaction with solid wedges at different inclination angles has been an area of much research in the past, but not many results have been obtained for shock wave reflection from liquid wedges. To find the transition angle from regular to irregular reflection of shock wave reflection over liquid wedges - both Newtonian and non-Newtonian liquids - we used a combination of experimental and numerical methods. In experiments, an inclined shock tube with adjustable inclination angle and a test section filled with the liquid of interest was used. Simulations were performed using a collection of CFD and CSD solvers to simulate the same situation as in the experiments. Results show that the transition angles for liquid wedges is different from smooth solid wedges, but agree fairly well if one assumes a certain surface roughness of the solid wedge.

2:23PM D4.00002 Measurements of Transient Phenomena in a Shock Tube using Pulse-Burst PIV, JUSTIN WAGNER, STEVEN BERESH, EDWARD DEMAURO, BRIAN PRUETT, PAUL FARIAS, Sandia National Laboratories — Time-resolved particle image velocimetry (TR-PIV) measurements are made in a shock tube using a pulse-burst laser. Two transient flowfields are investigated including the baseline flow in the empty shock tube and the wake growth downstream of a cylinder spanning the width of the test section. Boundary layer growth is observed following the passage of the incident shock in the baseline flow, while the core flow velocity increases with time. Comparison of this measured core flow acceleration to that predicted using classical unsteady boundary layer growth models shows good agreement in some instances. As a result of wall boundary layers, a significant amount of spatial non-uniformity remains in the flow following the passage of the end-wall reflected shock. In the transient wake growth measurements, the wake downstream of the cylinder is symmetric immediately following the passage of the incident shock. At later times, the wake transitions to von Karman vortex shedding. The TR-PIV data are bandpass filtered about the vortex shedding frequency and its harmonics to reveal additional details on the transient wake growth.

2:36PM D4.00003 Kinematical Compatibility Conditions for Vorticity Across Shock Waves, ROY BATY, Los Alamos National Laboratory — This work develops the general kinematical compatibility conditions for vorticity across arbitrary shock waves in compressible, inviscid fluids. The vorticity compatibility conditions are derived from the jump conditions across the shock surface using singular distributions defined on two-dimensional shock wave surfaces embedded in three-dimensional flow fields. The singular distributions are represented as generalized differential operators concentrated on moving shock wave surfaces. The derivation of the compatibility conditions for vorticity requires the application of second-order generalized derivatives and elementary tensor algebra. The well-known vorticity jump conditions across a shock wave are then shown to follow from the general kinematical compatibility conditions for vorticity by expressing the flow field velocity in vectorial components normal and tangential to a shock surface.

2:49PM D4.00004 Passive Shock Wave Attenuation by Liquid Sheets, NICHOLAS AMEN, VERONICA ELIASSON, University of Southern California — The use of liquid sheets to understand passive shock wave attenuation was investigated experimentally by impacting planar liquid sheets of varying thicknesses with planar shock waves. To create a sheet of liquid, a liquid container was designed to hold liquid in the desired shape with 5, 10, and 20 mm thickness in the test section of a shock tube. Planar shock waves with shock Mach numbers ranging from $M_s = 1.1$ to 1.5 were generated. In addition to varying the shock Mach number, different liquid media, water and a cornstarch suspension, were used. Pressure traces show that stronger incident shock waves result in higher reflected and transmitted pressures and that a thicker sheet reduces transmitted pressure but increases reflected pressure. Furthermore, for water versus cornstarch suspensions, a different thickness threshold was found to determine whether the transmitted wave is a shock wave or a pressure wave. Also, the breakup of the liquid sheets was fundamentally different between the two liquids.

1This work is supported by National Science Foundation (NSF grant #CBET-1437412)

3:02PM D4.00005 Mitigating Shock Waves Using Solid Obstacles with Semi-Circular Grooves, NICHOLAS AMEN, VERONICA ELIASSON, University of Southern California, ARTURO CAJAL, Universidad Nacional Autónoma de México (UNAM), VERONICA ELIASSON, University of Southern California — An experimental investigation was performed to assess the effectiveness of multiple obstacles with semi-circular grooves placed along a logarithmic spiral curve as a method to attenuate a shock wave. Previous research has shown that the use of multiple obstacles arranged in a log spiral configuration can attenuate a shock wave by reducing the energy of the wave. In this study, four different obstacle cross-sections in the same log spiral configuration were considered. All spirals had square cross-sections with zero, one, two, or three semi-circular grooves in each face. They were placed inside a shock tube where the incident, reflected, and transmitted shocks were visualized with high-speed schlieren imaging and the pressure histories were recorded with four pressure transducers. Each case was studied varying shock Mach numbers between $M = 1.1$ and $M = 1.5$. The pressure traces were used to establish incident, reflected, and transmitted shock strength and wave speed. Results for the zero groove case were compared to that of the one, two, and three groove cross-sections which show that increasing the number of grooves has a profound effect on the generation of the reflected wave and impacts the transmitted wave via vortex generation behind the spiral.

3:15PM D4.00006 Effect of initial perturbation amplitude on Richtmeyer-Meshkov flows induced by strong shocks, ZACHARY DELL, Carnegie Mellon University, ROBERT STELLINGWERF, Stellingwerf Consulting, SNEZHANA ABARZHI, Carnegie Mellon University — We study the effect initial perturbation on the Richtmeyer-Meshkov (RM) flows induced by strong shocks in fluids with contrasting densities. Smooth Particle Hydrodynamics simulations are employed. Broad range of shock strengths and density ratios is considered ($M = 3.5, 10$, and $Atwood = 0.6, 0.8, 0.95$). The amplitude of initial single mode sinusoidal perturbation of the interface varies from 0% to 100% of its wavelength. We analyze the initial growth-rate of the RMI immediately after the shock passage, when the perturbation amplitude increases linearly with time. We find that the initial growth-rate of RMI is a non-monotone function of the amplitude of the initial perturbation. This restrains the amount of energy that can be deposited by the shock at the interface. The maximum value of the initial growth-rate depends strongly and the corresponding value of the initial perturbation amplitude depends only slightly on the shock strength and density ratio. The maximum value of the initial growth-rate increases with the increase of the Atwood number for a fixed Mach number, and decreases with the increase of the Mach number for a fixed Atwood number. We argue that the non-monotonicity of RMI growth-rate is a result of a combination of geometric effect and the effect of secondary shocks.
3:28PM D4.00007 Numerical modeling of an experimental shock tube for traumatic brain injury studies. MICHAEL PHILLIPS, JONATHAN D. REGELE, Iowa State University — Unfortunately, Improvised Explosive Devices (IEDs) are encountered commonly by both civilians and military soldiers throughout the world. Over a decade of medical history suggests that traumatic brain injury (TBI) may result from exposure to the blast waves created by these explosions, even if the person does not experience any immediate injury or lose consciousness. Medical researchers study the exposure of mice and rats to blast waves created in specially designed shock tubes to understand the effect on brain tissue. A newly developed table-top shock tube with a short driver section has been developed for mice experiments to reduce the time necessary to administer the blast radiation and increase the amount of statistical information available. In this study, numerical simulations of this shock tube are performed to assess how the blast wave takes its shape. The pressure profiles obtained from the numerical results are compared with the pressure histories from the experimental pressure transducers. The results show differences in behavior from what was expected, but the blast wave may still be an effective means of studying TBI.

3:41PM D4.00008 Experimental Investigation of the Interaction of Blast Waves Generated by Exploding Wires using Background Oriented Schlieren. JONATHAN GROSS, VERONICA ELIASSON, University of Southern California — Work has been performed to experimentally characterize the interaction of a multiple blast waves. The blast waves were generated using an exploding wire system. This system can store up to 400 J of energy in a high voltage capacitor bank. By discharging the capacitors through wires of a diameter of 150 μm it was possible to produce blast waves with Mach numbers as high as 2.3 at a distance of 40 mm from the center of the blast. A parametric study was performed to measure the behavior of the shocks for a variety of wire thicknesses, voltages, and separation distances. Additionally a background oriented schlieren system was used to quantify the flowfield behind the shocks. The interaction of the shocks featured expected nonlinear phenomena such as the presence of Mach stems, and showed good agreement with results in the shock wave literature. This investigation lays the groundwork for subsequent research that will use exploding wires to experimentally reproduce conditions investigated numerically, in which the effects of multiple converging blast waves on a central target were investigated.

3:54PM D4.00009 Reflections Over Coupled Surfaces by Means of a High Resolution Setup. MEITAL GEVA, OMRI RAM, OREN SADOT, Pearlstone Center for Aeronautical Engineering Studies, Ben-Gurion University, BEN-GURION UNIVERSITY OF THE NEGEV, ISREAL TEAM — The reflection patterns over two coupled cylindrical surfaces are studied using a high spatial and high temporal resolution experimental setup. This fully automated setup enabled the repetition of experiments many times while retaining extremely high resolution. For the investigated moderate shock strengths, the repeatability was less than 0.01 in the Mach number. Each experiment produced a single image with a pixel size of 0.03 mm. All images were later sequentially merged generating a detailed description of a single reflection process. Unlike previous studies in which analysis was subject to human inconsistency, an automatic image processing procedure was used to locate the triple point in each image. The high resolution enabled the experimental detection of the early stages of Mach-reflection as were never demonstrated before. The experimental results were compared with numerical computation and a suitable uncertainty analysis was performed. The reflection over the first model enabled the transitions between MR→RR→MR. These successive transitions have proven the existence of a non-stationary hysteresis shock-wave reflection phenomenon. The reflection over the second model enabled the monitoring of the RR→MR transition and the evolution of a newly three shock configuration established on the Mach stem of the original reflection (MRMR). It was found that the MRMR→MRRR transition angles could be adjusted to match those obtained over a single cylinder.

4:07PM D4.00010 ABSTRACT WITHDRAWN –

Sunday, November 22, 2015 2:10PM - 4:20PM –
Session D5 Combustion I 104 - Javier Urzay, Stanford University

2:10PM D5.00001 Influence of equivalence ratio on the mechanism of pressure wave generation during knocking combustion. HIROSHI TERASHIMA, The University of Tokyo, MITSUO KOSHI, Yokohama National University — Knocking in spark-assisted engines is known as a severe pressure oscillation mainly caused by hot-spot autoignition in end-gas regions. In this study, knocking combustion of n-heptane/air mixtures modeled in a one-dimensional constant volume reactor is simulated with particular emphasis on the effects of equivalence ratio (0.6 to 2.0) on the mechanism of pressure wave generation. An efficient compressible flow solver with detailed chemical kinetics of n-heptane (373 species and 1071 reactions) is applied. The results demonstrate that the presence of negative temperature coefficient region significantly influence the knocking timing and knocking intensity, i.e., pressure wave amplitude in end-gas regions. The condition with equivalence ratios lower than 1.0 mostly leads to the reduction of the knocking intensity because of slower heat release rates of end-gas autoignition. On the other hand, the results with higher equivalence ratios of 1.2 to 2.0 indicate that a significant peak in the knocking intensity is produced at an equivalence ratio, which varies with initial temperature conditions. The final presentation will address the relationship between the knocking intensity and equivalence ratio with the discussion on detailed physics of pressure wave generation.

2:23PM D5.00002 Combustion properties in multi-particulate flows with direct numerical simulation. LONGHUI ZHANG, CHANGFU YOU, Department of Thermal Engineering, Tsinghua University, DEPARTMENT OF THERMAL ENGINEERING, TSINGHUA UNIVERSITY TEAM — Multiphase combustion is widely applied in industries. With high solid concentrations, the influence of particle interactions must be taken into account in the combustion models. Many literature have developed group combustion models with particles treated as point sources. However, for dense phase flow the particle size is in the same scale with the average particle spacing, and the point source assumption is not accurate enough. This work presents the fully resolved direct numerical simulation results of reacting particulate flows. The particles are considered as finite sized regions in flow fields. Therefore the influence of particle motion and distribution on combustion properties can be obtained. The moving, colliding and burning process of char particles in confined space is calculated. The flow and combustion characteristics under different conditions are observed. The char burning rate is compared to that of fixed char particles with uniform distribution. The result shows that the burning rate decreases when the particle distribution non-uniformity perpendicular to the main flow direction increases. A model of char group combustion rate is developed using non-uniformity coefficients.
2:36PM D5.00003 Evaporation and Combustion Characteristics of Multicomponent Fuels\textsuperscript{1}. PAVAN GOVINDARAJU, Stanford University, ALESSANDRO STAGNI, Politecnico Di Milano, MATTHIAS IHME, Stanford University — Current generation fuel mixtures are fuels of hundreds of complicated organic compounds and accurate modeling of their combustion characteristics provides fundamental physical insights which also help in the design of efficient combustors. This however requires accurate simulation of both evaporation and combustion processes, which, in case of such fuels, demands an approach based on calculating properties using only the information of functional groups present in the mixture. The presentation will elaborate on the assumptions and the framework utilized for evaporation and chemical mechanisms. We also present a comparison between various fuels used in the aviation industry as test cases while highlighting on their pros and cons. The focus of the talk will however be on the physical aspects captured using 1D simulations, i.e., preferential evaporation of each species, ignition parameters and emissions while justifying the numerical calculations with experimental data at each stage. Further work involving the coupling of flow with evaporation and combustion can be performed and we briefly discuss why a DNS is necessary to characterize the various combustion regimes.

\textsuperscript{1}Federal Aviation Administration

2:49PM D5.00004 LES of Mild Combustion using Pareto-efficient Combustion Adaptation. HAO WU, Stanford University, MICHAEL EVANS, University of Adelaide, MATTHIAS IHME, Stanford University — Moderate or Intense Low-Oxygen Dilution (MILD) combustion is a combustion regime that provides opportunities for improved thermal efficiency and reduced pollutant emissions. In this study, large-eddy simulation is used to investigate the ignition, mixing, and stabilization of a jet flame in this kinetics-controlled combustion regime. The combustion process is modeled by a Pareto-efficient combustion (PEC) formulation that optimally combines reaction-transport and chemistry combustion models. In this approach, a three-stream flamelet/progress variable model is used as a computationally efficient description of equilibrated flame regions, and a finite-rate chemistry representation is employed to accurately represent the ignition behavior and flame stabilization. Through comparisons with experiments and simulations with single-regime combustion models, it will be shown that this Pareto-efficient combustion submodel assignment accurately captures important dynamics in complex turbulent flame configurations.

3:02PM D5.00005 LES of combustion dynamics near blowout in a realistic gas-turbine combustor. LUCAS ESCALPEZ, MEDIH N. B. MA, PETER C. MA, JEFF O'BRIEN, SERENA CARBAJAL, MATTHIAS IHME, Stanford University — Driven by increasingly stringent emission regulations, modern gas turbines operate at lean conditions to reduce combustion chamber temperature and NO\textsubscript{x} emissions. However, as the combustor operates closer to the lean blow-out (LBO) limit, flame stabilization mechanisms are weakened, which increases the risk for complete flame blowout. To better understand the LBO-process, large-eddy simulations of the combustion dynamics near blowout are performed in a realistic two-phase flow combustor. An unstructured incompressible Navier-Stokes solver is used in combination with a Lagrangian dispersed phase formulation. Flame dynamics near and at LBO conditions are studied to identify the role of the liquid fuel composition, spray evaporation, and complex flow pattern on the LBO limit.

3:15PM D5.00006 A Jet-Stirred Apparatus for Turbulent Combustion Experiments\textsuperscript{1}. ABBASALI DAVANI, PAUL RONNEY, University of Southern California — A novel jet-stirred combustion chamber is designed to study turbulent premixed flames. In the new approach, multiple impinging turbulent jets are used to stir the mixture. It is well known that pair of countercflowing turbulent jets produces nearly a constant intensity (u') along the jet axes. In this study, different numbers of impinging jets in various configurations are used to produce isotropic turbulence intensity. FLUENT simulations have been conducted to assess the viability of the proposed chamber. In order to be able to compare different configurations, three different non-dimensional indices are introduced: Mean flow index; Homogeneity index, and Isotropy index. Using these indices one can compare various chambers including conventional Fan-stirred Reactors. Results show that a concentric inlet/outlet chamber (CAIO) with 8 inlets and 8 outlets with inlet velocity of 20 m/s and initial intensity of 15% produces near zero mean flow and 2.5 m/s turbulence intensity which is much more higher than reported values for Fan-stirred chamber.

\textsuperscript{1}This research was sponsored by National Science Foundation

3:28PM D5.00007 Simulation of High-Pressure Methane Hydrate Combustion\textsuperscript{1}. PAVEL POPOV, WILLIAM SIRIGNANO, University of California at Irvine, Mechanical and Aerospace Engineering — With its prevalence in ocean floor deposits, methane hydrate has recently attracted considerable attention in the combustion community. We present a new scheme for the simulation of methane hydrate combustion at high, near critical pressures. This process features a combination of solid, liquid and gas phases, wherein the solid methane hydrate melts into a bubbly liquid, which then evaporates into a gas phase; methane-air combustion occurs in the gas phase. In addition to its multiphase nature, this problem features the additional challenge of modelling the gas/liquid phase transition at near-critical pressures. A new computational procedure has been developed to simulate this problem, using a detailed chemical mechanism for the simulation of reaction in the gas phase, and featuring a volume-of-fluid (VOF) approach for the simulation of the liquid phase with gas bubbles — a low Stokes number is assumed. This procedure is applied to a laminar shear flow methane hydrate combustion problem. Particular attention is directed to the effects on simulation results of the high-pressure equation of state, liquid/gas phase transition modelling, and the bubbly liquid phase modelling. Simulation results are compared to experimental observations.

\textsuperscript{1}Supported by AFOSR grant FA9550-12-1-0156, AFOSR scientific manager: Dr. Mitat Birkan

3:41PM D5.00008 Analysis of Fuel Injection and Atomization of a Hybrid Air-Blas Atomizer\textsuperscript{1}. LUCAS ESCALPEZ, Stanford University, TIMO BÜCHHAGEN, SAMEER NAIK, JAY GORE, ROBERT LÜCHT, Purdue University, MATTHIAS IHME, Stanford University — Fuel injection and atomization are of direct importance to the design of injector systems in aviation gas turbine engines. Primary and secondary breakup processes have significant influence on the drop-size distribution, fuel deposition, and flame stabilization, thereby directly affecting fuel conversion, combustion stability, and emission formation. The lack of predictive modeling capabilities for the reliable characterization of primary and secondary breakup mechanisms is still one of the main issues in improving injector systems. In this study, an unstructured Volume-of-Fluid method was used in conjunction with a Lagrangian-spray framework to conduct high-fidelity simulations of the breakup and atomization processes in a realistic gas turbine hybrid air blast atomizer. Results for injection with JP-8 aviation fuel are presented and compared to available experimental data.

\textsuperscript{1}Financial support through the FAA National Jet Fuel Combustion Program is gratefully acknowledged.
3:54PM D5.00009 Multiscale Interactions and Backscatter in Premixed Combustion. PETER HAMILTON, COLIN TOWERY, University of Colorado, Boulder, JEFFREY O'BRIEN, Stanford University, ALEXEI POLUDNENKO, Naval Research Laboratory, JAVIER URZAY, Center for Turbulence Research, Stanford University, MATTHIAS IHME, Stanford University — Multiscale interactions and energy transfer between turbulence and flames are fundamental to understanding and modeling premixed turbulent reacting flows. To investigate such flows, direct numerical simulations of statistically planar turbulent premixed flames have been performed, and the dynamics of kinetic energy transfer are examined in both spectral and physical spaces. In the spectral analysis, two-dimensional kinetic energy spectra and triadic interactions are computed through the flame brush. It is found that there is suppression of turbulent small-scale motions in the combustion products, along with backscatter of energy for a range of scales near the thermal laminar flame width. In the physical-space analysis, a differential filter is applied to examine the transfer of kinetic energy between subgrid and resolved scales in the context of large eddy simulations. Subgrid-scale backscatter of kinetic energy driven by combustion is found to prevail over forward scatter throughout the flame brush. The spectral- and physical-space analyses thus both suggest an enhancement of reverse-cascade phenomena in the flame brush, which is possibly driven by accumulation of kinetic energy in the scales where combustion-induced heat release is preferentially deployed.

4:07PM D5.00010 A single-fluid multiphase formulation for diffuse-interface modeling of high-pressure liquid-fueled transcritical mixing layers. LLUIS JOFRE, JAVIER URZAY, ALI MANI, PARVIZ MOIN, Center for Turbulence Research, Stanford University — Liquid propellants are often used in propulsion systems. In subcritical conditions, atomization involves the rupture of the liquid volume through the competition between aerodynamic shearing and surface tension. In contrast, the classic atomization description becomes inadequate at supercritical conditions when the characteristic temperature and pressure of the gas environment are above the corresponding critical values. In that limit, the latent heat of vaporization vanishes and there is virtually no surface tension that prevents rupture of the liquid core and diffusive mixing with the gas environment. In particular, in high-pressure gas turbines the liquid fuel is seldom preheated to supercritical temperatures before injection, and the presence of both subcritical and supercritical conditions in the combustion chamber is warranted. Consideration of the liquid phase is therefore required in addition to the gas phase and the supercritical mixture. A single-fluid multiphase formulation of this problem is presented to investigate mixing and combustion in fuel-air transcritical mixing layers. The formulation makes use of diffuse-interface concepts facilitated by the relatively larger interface thicknesses at these high pressures.

1Funded by AFOSR.

Sunday, November 22, 2015 2:10PM - 4:20PM – Session D6 Granular Flows: Jamming and Cooling 105 - Matthieu Wyart, NYU

2:10PM D6.00001 Unified Theory of Inertial Granular Flows and Non-Brownian Suspensions. MATTHIEU WYART, nyu, ERIC DEGIULLI, NYU Physics, EDAN LERNER, university of amsterdam, GUSTAVO DURING, Pontificia Universidad Catlica de Chile, WYART GROUP TEAM — Rheological properties of dense flows of hard particles are singular as one approaches the jamming threshold where flow ceases, both for aerial granular flows dominated by inertia, and for over-damped suspensions. Concomitantly, the lengthscale characterizing velocity correlations appears to diverge at jamming. Here we introduce a theoretical framework that proposes a tentative, but potentially complete scaling description of stationary flows. Our analysis, which focuses on frictionless particles, applies to both suspensions and inertial flows of hard particles. We compare our predictions with the empirical literature, as well as with novel numerical data. Overall we find a very good agreement between theory and observations, except for frictional inertial flows whose scaling properties clearly differ from frictionless systems. For over-damped flows, more observations are needed to decide if friction is a relevant perturbation or not. Our analysis makes several new predictions on microscopic dynamical quantities that should be accessible experimentally.

2:23PM D6.00002 Effect of friction on shear jamming. DONG WANG, JONATHAN BARES, Duke University, JOSHUA DJUKSMAN, Wageningen University and Research Centre, JIE REN, Merck & Co., HU ZHENG, Hohai University; Duke University, ROBERT BEHRINGER, Duke University — Shear jamming of granular materials was first found for systems of frictional disks, with a static friction coefficient \( \mu \approx 0.6 \) (Bi et al. Nature (2011)). Jamming by shear is obtained by starting from a zero-stress state with a packing fraction \( \phi \) between \( \phi_j \) (isotropic jamming) and a lowest \( \phi_S \) for shear jamming. This phenomenon is associated with strong anisotropy in stress and the contact network in the form of force chains, which are stabilized and/or enhanced by the presence of friction. Whether shear jamming occurs for frictionless particles is under debate. The issue we address experimentally is how changing friction affects shear jamming. By applying a homogeneous simple shear, we study the effect of friction by using photoelastic disks either wrapped with Teflon to reduce friction or with fine teeth on the edge to increase friction. Jamming is still observed; however, the difference \( \phi_j - \phi_S \) is smaller with lower friction. We also observe larger fluctuations due to initial configurations both at the lowest and the highest friction systems studied. Ongoing work is to characterize response from different friction systems under shear with information at local scale.

1We acknowledge support from NSF-DMR1206351, NASA NNX15AD38G and W.M. Keck Foundation.

2:36PM D6.00003 ABSTRACT WITHDRAWN —

2:49PM D6.00004 Relaxation of densely packed gel particles under cyclic shearing. J.C. TSAI, Inst. of Physics, Academia Sinica, M.R. CHOU, P.C. HUANG, NTU, H.T. FEI. Inst. of Physics, Academia Sinica, J.R. HUANG, National Taiwan Normal University — We study experimentally the rheological response of fluid-immersed hydrogel particles. The particles are centimeter-sized and are driven by a roughened cone-shaped upper boundary, which imposes a cyclic shearing with a substantial stall period inserted between each reversal of its motion. The stall period reveals a characteristic timescale of relaxation belonging to these soft materials, in contrast to the build-up of stress that reflects a characteristic strain accumulated since each re-start of the shearing. We provide a coherent explanation on how the relaxation and the residual stress are related to observed steady-state rheology at different strain rates, and the use of a previously developed tomographical imaging technique allows us to look into the particle displacements during the relaxation.
3:02PM D6.00005 Yielding in a strongly aggregated colloidal gel: 2D simulations and theory, SAIKAT ROY, MAHESH TIRUMUKUDULU, Department of Chemical Engineering, Indian Institute of Technology Bombay, Powai, Mumbai - 400076, India — We investigated the micro-structural details and the mechanical response under uniaxial compression of the strongly aggregating gel starting from low to high packing fraction. The numerical simulations account for short-range inter-particle attractions, normal and tangential deformation at particle contacts, sliding and rolling friction, and preparation history. It is observed that in the absence of rolling resistance (RR), the average coordination number varies only slightly with compaction whereas it is significant in the presence of RR. The particle contact distribution is isotropic throughout the consolidation process. In both cases, the yield strain is constant with the volume fraction. The modulus values are very similar at different attraction, and with and without RR implying that the elastic modulus does not scale with attraction. The modulus was found to be a weak function of the preparation history. The increase in yield stress with volume fraction is a consequence of the increased elastic modulus of the network. However, the yield stress scales similarly both with and without RR. The power law exponent of 5.4 is in good agreement with previous simulation results. A micromechanical theory is also proposed to describe the stress versus strain relation for the gelled network.

3:15PM D6.00006 Experiments on Memory in a Sheared Soft Solid, NATHAN KEIM, DEVIN WIEKER, LUKE HOROWITZ, California Polytechnic State University, San Luis Obispo — We consider how a soft 2D jammed material may form memories of past deformation. Our experiments cyclically shear a material made of repulsive particles at an oil-water interface, observing the motion of many particles. Under repeated shearing, the system can evolve toward a “limit cycle” in which the same particle rearrangements recur on each cycle of shear; the set of rearrangements is specific to the strain amplitude. We discuss how the materials history-dependence may be viewed as a memory of the strain amplitude, and we report on progress in describing this behavior, including whether memories of multiple strains may coexist.

3:28PM D6.00007 Controlling the shear profile of highly strained granular materials, JONATHAN BARES, BOB BEHRINGER, Duke University — Bi et al. (Nature 2011) have shown that, if sheared, a granular material can jam even if its packing fraction (ϕ) is lower than the critical isotropic jamming point ϕ_J. They have introduced a new critical packing fraction value ϕ_S such that for ϕ_S < ϕ < ϕ_J, the system jams if sheared. Nevertheless, the value of ϕ_S as a function of the shear profile or the strain necessary to observe jamming remain poorly understood because of the experimental complexity to access high strain without the formation of shear bands. We present a novel 2D periodic shear apparatus made of 21 independent, aligned and mirrored glass rings. Each of ring can be moved independently which permits us to impose any desired shear profile. The circular geometry allows access to any strain value. The forces between grains are measured using reflective photoelasticity. This talk will present this novel apparatus and discuss the effect of the shear profile and shear amplitude on the jamming transition.

3:41PM D6.00008 Self organization and jamming in magnetic photoelastic particles, MEREDITH COX, Durham Academy Upper School, JONATHAN BARES, DONG WANG, ROBERT BEHRINGER, Duke University — Many experimental studies of simple particles in granular systems have been conducted, but the behavior of complex particles in such systems has not been addressed. There has been a growing interest in functionalized microparticles, and the study of these complex particles may reveal interesting analogues between micro- and macroparticles. We perform experiments to investigate magnetic particles in a 2D granular material close to the jamming transition. We incrementally compress photoelastic particles containing magnets and image the interparticle forces in each compression using a photoelastic technique. To track the orientation of individual particles, we draw UV-visible bars on each particle image. The results of the system under ultraviolet light. We repeat the experimental procedure using varying ratios of magnetic to nonmagnetic particles from 0% magnetic to 100% magnetic. By using custom software to resolve particle deformations, we extract particle contact forces and demonstrate that as the concentration of nonmagnetic particles grows, the rate of increase of pressure with strain also grows.

3:54PM D6.00009 Granular flow and clog in silo with moving outlet, KIWING TO, HSIANG-TING TAI, Institute of Physics, Academia Sinica, 128 Sec. 2 Academia Road, Nankang, Taipei, Taiwan — When grains flow out of a silo, the flow rate Q increases with the outlet size d. If d is too small, an arch may form and blocks the flow at the outlet. To recover from clogging, the arch have to be destroyed. Oscillating the outlet mechanically is one simple way of destroying the arch. In this paper, we report the effect of oscillating the outlet on the processes of clogging as well as recovery from clogging in two-dimensional silo equipped with movable outlet. We measure the flow rate Q(d) and find that, in the presence of outlet oscillation, Q(d) may remain finite even when d is only slightly larger than the grain diameter. Transition from continuous flow to intermittent flow occurs by decreasing d or by reducing the oscillation speed. In addition, at small d when the flow is intermittent, the flow time (duration of flow before clogged) follows Poisson distribution whereas the recovery time (duration of clog before flow recovery) follows power law distribution.

4:07PM D6.00010 Shear-induced dynamical phase-transitions in a granular system, JIE ZHANG, YINQIAO WANG, HONGYANG TANG, Shanghai Jiaotong University — In this study, we investigate the granular materials in a 2D Couette cell under shear using photo-elastic disks. For initial packing fractions ϕ smaller than ϕ_c = 82%, an initially stress-free state can be first shear jammed and then gradually relaxes to a steady state with strong stress fluctuations. Such a steady state will then make a stochastic transition to an unjammed state under the continuous shear. For packing fractions ϕ larger than ϕ_c, we observe no such transitions. The characteristic strain of the transition diverges as a function of |ϕ − ϕ_c|^{-2} which resembles a second-order dynamical phase transition. We interpret some portion of the results of such phase transitions using a mean-field model that was originally proposed to explain the discontinuous shear thickening and the shear jamming of the frictional granular materials. We are currently still investigating such an intriguing phenomenon in order to understand the detailed dynamics of the transition.

Sunday, November 22, 2015 2:10PM - 4:20PM – Session D7 Acoustics II: General

107 - Pablo Rendon, Universidad Nacional Autonoma de Mexico
tic levitator\textsuperscript{1}, PABLO LUIS RENDON, RICARDO R. BOULLOSA, Centro de Ciencias Aplicadas y Desarrollo Tecnologico, Universidad Nacional Autonoma de Mexico, CARLOS ECHEVERRIA, DAVID PORTA, Facultad de Ciencias, Universidad Nacional Autonoma de Mexico — We consider a model of a single axis acoustic levitator consisting of two cylinders immersed in air and directed along the same axis. The first cylinder has a flat termination and functions as a sound emitter, and the second cylinder, which is simply a reflector, has the side facing the first cylinder cut out by a spherical surface. By making the first cylinder vibrate at ultrasonic frequencies a standing wave is produced in the air between the cylinders which makes it possible, by means of the acoustic radiation pressure, to levitate one or several objects of different shapes, such as spheres or disks. We use schlieren imaging to observe the acoustic field resulting from the levitation of one or several objects, and compare these results to previous numerical approximations of the field obtained using a finite element method.

\textsuperscript{1}The authors acknowledge financial support from DGAPA-UNAM through project PAPIIT IN109214.

2:10PM D7.00001 Schlieren imaging of the standing wave field in an ultrasonic acoustic levitator\textsuperscript{1}, PABLO LUIS RENDON, RICARDO R. BOULLOSA, Centro de Ciencias Aplicadas y Desarrollo Tecnologico, Universidad Nacional Autonoma de Mexico, CARLOS ECHEVERRIA, DAVID PORTA, Facultad de Ciencias, Universidad Nacional Autonoma de Mexico — We consider a model of a single axis acoustic levitator consisting of two cylinders immersed in air and directed along the same axis. The first cylinder has a flat termination and functions as a sound emitter, and the second cylinder, which is simply a reflector, has the side facing the first cylinder cut out by a spherical surface. By making the first cylinder vibrate at ultrasonic frequencies a standing wave is produced in the air between the cylinders which makes it possible, by means of the acoustic radiation pressure, to levitate one or several objects of different shapes, such as spheres or disks. We use schlieren imaging to observe the acoustic field resulting from the levitation of one or several objects, and compare these results to previous numerical approximations of the field obtained using a finite element method.

\textsuperscript{1}The authors acknowledge financial support from DGAPA-UNAM through project PAPIIT IN109214.

2:33PM D7.00002 Large eddy simulation of trailing edge noise\textsuperscript{1}, JACOB KELLER, ZANE NITZKORSKI, KRISHNAN MAHESH, University of Minnesota — Noise generation is an important engineering constraint to many marine vehicles. A significant portion of the noise comes from propellers and rotors, specifically due to flow interactions at the trailing edge. Large eddy simulation is used to investigate the noise produced by a turbulent 45 degree beveled trailing edge and a NACA 0012 airfoil. A porous surface Ffowcs-Williams and Hawking acoustic analogy is combined with a dynamic endcap method to compute the sound. This methodology allows for the impact of incident flow noise versus the total noise to be assessed. LES results for the 45 degree beveled trailing edge are compared to experiment at $M = 0.1$ and $Re = 1.96 \times 10^7$. The effect of boundary layer thickness on sound production is investigated by computing using both the experimental boundary layer thickness and a thinner boundary layer. Direct numerical simulation results of the NACA 0012 are compared to available data at $M = 0.4$ and $Re = 5.0 \times 10^5$ for both the hydrodynamic field and the acoustic field. Sound intensities and directivities are investigated and compared. Finally, some of the physical mechanisms of far-field noise generation, common to the two configurations, are discussed.

\textsuperscript{1}Supported by Office of Naval research

2:36PM D7.00003 Sound produced by subcritical Reynolds number cylinder flow\textsuperscript{1}, ZANE NITZKORSKI, KRISHNAN MAHESH, Univ. of Minn - Minneapolis — Sound production from cylinders has been studied due to their capabilities in investigating bluff body flow noise. The effect of Reynolds number for circular cylinders for $Re = 3900, 10000,$ and $89000$ are investigated with the resulting impact on the noise generation process. The physics of noise production are investigated and a model for understanding the source and direction of noise propagation is presented. The acoustic solution is calculated from a novel porous Ffowcs-Williams and Hawking acoustic analogy which is described and allows for investigating the scattered and incident acoustic fields by separating volume contributions from the total noise. The volume source terms are correlated over multiple planes to obtain a convection velocity that is then used to determine a corrective convective flux at the FW-H porous surface. The hydrodynamic fields are validated and the computed sound is compared with experiments. The effect of spanwise coherence and its effect on the physics of sound production is discussed. For the highest Reynolds number case, a dynamic mode decomposition is applied and performed on the acoustic sources to demonstrate their spatial distribution and net effect.

\textsuperscript{1}Supported by Office of Naval research

2:49PM D7.00004 Eulerian Simulation of Acoustic Waves Over Long Range in Realistic Environments\textsuperscript{1}, SUBHASHINI CHITTA, JOHN STEINHOFF, Wave CPC, Inc. — In this paper, we describe a new method for computation of long-range acoustics. The approach is a hybrid of near and far-field methods, and is unique in its Eulerian treatment of the far-field propagation. The near-field generated by any existing method to project an acoustic solution onto a spherical surface that surrounds the source. The acoustic field on this source surface is then extended to an arbitrarily large distance in an inhomogeneous far-field. This would normally require an Eulerian solution of the wave equation. However, conventional Eulerian methods have prohibitive grid requirements. This problem is overcome by using a new method, “Wave Confinement” (WC) that propagates wave-identifying phase fronts as nonlinear solitary waves that live on grid indefinitely. This involves modification of wave equation by the addition of a nonlinear term without changing the basic conservation properties of the equation. These solitary waves can then be used to “carry” the essential integrals of the acoustic wave. For example, arrival time, centroid position and other properties that are invariant as the wave passes a grid point. Because of this property the grid can be made as coarse as necessary, with overall accuracy to resolve atmospheric/ground variations.

\textsuperscript{1}This work is being funded by the U.S. Army under a Small Business Innovation Research (SBIR) program (contract number: W911W6-12-C-0036). The authors would like to thank Dr. Frank Caradonna and Dr. Ben W. Sim for this support.

3:02PM D7.00005 Roles of a scatter on boundary-layer instability and acoustic radiation , MING DONG, Tianjin University, XUESONG WU, Imperial College — When a boundary-layer instability mode propagates through a region of rapid distortion, the ensuing scattering causes two consequences of physical interest. First, the amplitude of the instability mode may be suppressed or energized. Second, substantial sound wave can be radiated by the boundary-layer instability mode. This paper focuses on this issue by proposing a framework which is called Local Scattering Theory. In this framework, a transmission coefficient, defined as the ratio of the T-S wave amplitude downstream of the scatter to that upstream, is introduced to characterize the effect of a local scatter on boundary-layer instability and transition. The mathematical formulation is based on triple-deck formalism, but in order to accommodate the acoustic far field, the unsteady terms in the upper deck are retained. By computation, the impacts of a steady local suction on flow instability and acoustic radiation are studied. It is found that, (1) a suction slot would suppress the oncoming T-S wave; (2) the acoustic waves radiated by the scattering effect have similar directivities; (3) the intensity of the sound increases with the mass flux when the latter is not too large, and it also increases with the frequency monotonously.

3:15PM D7.00006 Acoustic Localization with Infrasonic Signals , ARNESHA THREATT, BRIAN ELBING, Oklahoma State University — Numerous geophysical and anthropogenic events emit infrasonic frequencies (<20 Hz), including volcanoes, hurricanes, wind turbines and tornadoes. These sounds, which cannot be heard by the human ear, can be detected from large distances (in excess of 100 miles) due to low frequency acoustic signals having a very low decay rate in the atmosphere. Thus infrasound could be useful for long-range, passive monitoring and detection of these events. An array of microphones separated by known distances can be used to locate a given source, which is known as acoustic localization. However, acoustic localization with infrasound is particularly challenging due to contamination from other signals, sensitivity to wind noise and producing a trusted source for system development. The objective of the current work is to create an infrasonic source using a propane torch wand or a subwoofer and locate the source using multiple infrasonic microphones. This presentation will present preliminary results from various microphone configurations used to locate the source.
3:28PM D7.00007 Nozzleless Spray Cooling Using Surface Acoustic Waves. Kar Man Ang, School of Engineering, Monash University, 47500 Bandar Sunway, Malaysia; Leslie Yeo, Micro/Nanophysics Research Laboratory, RMIT University, Melbourne, VIC 3001, Australia; James Friend, Department of Mechanical and Aerospace Engineering, University of California, San Diego, Ca 92093, United States; Yew Mun Hung, Ming Kwang Tan, School of Engineering, Monash University, 47500 Bandar Sunway, Selangor, Malaysia. Due to its reliability and portability, surface acoustic wave (SAW) atomization is an attractive approach for the generation of monodisperse microdroplets in microfluidics devices. Here, we present a nozzleless spray cooling technique via SAW atomization with key advantages of downward scalability by simply increasing the excitation frequency. With generation of micron size droplets through surface destabilization using SAW, the clogging issues commonly encountered by spraying nozzle can be neutralized. Using deionised water, cooling is improved when the atomization rate is increased and the position of the device is optimized such that the atomized droplets can be easily seeded into the upstream of the flow circulation. Cooling is further improved with the use of nanofluids; a suspension of nanoparticles in water. By increasing nanoparticle mass concentration from 1% to 3%, cooling is enhanced due to the deposition and formation of nanoparticle clusters on heated surface and eventually increase the surface area. However, further increase the concentration to 10% reduces the cooling efficiency due to drastic increase in viscosity $\mu$ that leads to lower atomization rate which scales as $n_0 \sim \mu^{-1/2}$.

3:41PM D7.00008 Tunable Acoustic Attenuation by Dilute Suspensions of Oblate-Spheroidal Ferromagnetic Particles Under an External Magnetic Field: An Experimental Study. Wuhan Yuan, Jerry Shan, Liping Liu, Rutgers University. The microstructure of suspensions of spheroidal ferromagnetic particles with subwavelength size can be controlled by an external field, making it possible to develop novel broadband acoustic materials with anistotropic and tunable acoustic properties. In this study we experimentally show that dilute suspensions of nickel microflakes exhibit a 20% to 30% change in attenuation coefficient at MHz frequencies upon changing the direction of an external magnetic field, at particle volume fractions of only 0.5%. Further investigations are conducted to study the mechanism behind this anisotropy. The effects of particle aligning and chaining are analyzed with the aid of optical transmission measurements. By making comparison to suspensions of spherical particles, we show that the ellipsoidal shape of the nickel microflakes plays an important role in tunable acoustic properties of these suspensions.

3:54PM D7.00009 Dynamic vortex interactions with flexible fibers and edges for prediction of owl noise suppression. Sarah Korykora, Justin Jaworski, Lehigh University. The compliant trailing-edge fringe of owls and the soft downy material on the upper wing surfaces are thought to enable their silent flight by weakening the interaction of boundary layer turbulence with these flexible structures. Previous analysis of vortex noise generation shows that flexible trailing edges have shown that the far-field acoustic power scaling can be weakened by up to the square of the Mach number relative to a rigid edge. However, it is unclear whether or not the wave-bearing feature or simply the flexible nature of the edge scatterer produces this noise suppression. To assess this distinction, a dynamic vortex interaction model is developed whereby the motion of a line vortex round a rigid but elastically-restrained wall-mounted fiber or trailing edge is determined numerically. Special attention is paid to the dynamic interaction between the flexible structure and vortex, which is accomplished via a conformal mapping relationship determined in closed form. Results from this analysis seek to develop a vortex sound model to discern the effect of flexible versus wave-bearing scatterers on turbulence noise suppression and help explain the mechanisms of silent owl flight.

4:07PM D7.00010 Development of a Transient Acoustic Boundary Element Method to Predict the Noise Signature of Swimming Fish. Nathan Wagenhofer, Lehigh University; Keith Moorhead, Justin Jaworski, Lehigh University. Animals have evolved flexible wings and fins to efficiently and quietly propel themselves through the air and water. The design of quiet and efficient bio-inspired propulsive concepts requires a rapid, unified computational framework that integrates three essential features: the fluid mechanics, the elastic structural response, and the noise generation. This study focuses on the development, validation, and demonstration of a transient, two-dimensional acoustic boundary element solver accelerated by a fast multipole algorithm. The resulting acoustic solver is used to characterize the acoustic signature produced by a vortex street advecting over a NACA 0012 airfoil, which is representative of vortex-body interactions that occur in schools of swimming fish. Both 2S and 2P canonical vortex streets generated using a range of Strouhal numbers $0.2 < \text{SI} < 0.4$ are investigated and the acoustic signature of the airfoil is quantified. This study provides the first estimate of the noise signature of a school of swimming fish.

Session D8 CFD: Turbulent Flows

2:10PM D8.00001 Data-driven RANS for prediction of wind turbine wakes. Giacomo Valerio Jungo, UT Dallas. The design of quiet and efficient bio-inspired propulsive concepts requires a rapid, unified computational framework that integrates three essential features: the fluid mechanics, the elastic structural response, and the noise generation. This study focuses on the development, validation, and demonstration of a transient, two-dimensional acoustic boundary element solver accelerated by a fast multipole algorithm. The resulting acoustic solver is used to characterize the acoustic signature produced by a vortex street advecting over a NACA 0012 airfoil, which is representative of vortex-body interactions that occur in schools of swimming fish. Both 2S and 2P canonical vortex streets generated using a range of Strouhal numbers $0.2 < \text{SI} < 0.4$ are investigated and the acoustic signature of the airfoil is quantified. This study provides the first estimate of the noise signature of a school of swimming fish.
2:36PM D8.00003 Air entrainment due to shear-flow free surface turbulence. XIANGMING YU, DICK K.P. YUE, KELLI HENDRICKSON, Massachusetts Institute of Technology, Cambridge, MA, 02139 — We perform direct numerical simulations to study air entrainment at the air-water interface in three dimensional sheared turbulent flows with two-phase coupled at the free surface, using a two-phase conservative Volume-Of-Fluid (cVOF) method. For a given Reynolds number the problem is governed by the Froude number (Fr), above a threshold value of which air entrainment (AE) is observed. We consider a range of Fr and study the dependence on Fr of the volume V of AE, and the underlying air entraining structures and mechanisms of the interface. We determine the scaling of V with Fr and identify two key mechanisms for AE characterized respectively by surface-parallel vorticity and wave breaking. The former is associated with rising lambda vortices and strong near-surface horizontal vorticity, while the latter can be quantified by the decrease in potential energy of the interface. We propose models parameterized on Fr and the local turbulent flow properties that predict the AE volume associated with each of these mechanisms.

2:49PM D8.00004 Minimum-dissipation models for large-eddy simulation1, HYUNJ JAN BAE, Stanford University, WYBE ROZEMA, University of Groningen, Netherlands, PARVIZ MOIN, Stanford University, ROEL VERSTAPPEN, University of Groningen, Netherlands — Minimum-dissipation eddy-viscosity models are a class of subgrid scale models for LES that give the minimum eddy dissipation required to dissipate the energy of subgrid scales. The QR minimum-dissipation model [Verstappen, J. Sci. Comp., 2011] gives good results in simulations of decaying grid turbulence carried out on an isotropic grid. In particular, due to the minimum dissipation property of the model, the predicted energy spectra are in very good agreement with the DNS results up to the cut-off wave number unlike other models. However, its results on anisotropic grids are often unsatisfactory because the model does not properly incorporate the grid anisotropy. We propose the anisotropic minimum-dissipation (AMD) model [Rozena et. al., submitted for publication, 2015], a minimum-dissipation model that generalizes the QR model to anisotropic grids. The AMD model is more cost effective than the dynamic Smagorinsky model, appropriately switches off in laminar and transitional flow on anisotropic grids, and its subgrid scale model is consistent with the theoretic subgrid tensor. Experiments show that the AMD model is as accurate as the dynamic Smagorinsky model and Vreman model in simulations of isotropic turbulence, temporal mixing layer, and turbulent channel flow.

1H. J. Bae acknowledges support from SGF. W. Rozema and R. Verstappen acknowledge sponsoring by NWO for the use of supercomputing facilities and the financial support to attend the CTR SP 2014.

3:02PM D8.00005 Error-measure for anisotropic grid-adaptation in turbulence-resolving simulations, SIAWASH TOOSI, JOHAN LARSSON, University of Maryland — Grid-adaptation requires an error-measure that identifies where the grid should be refined. In the case of turbulence-resolving simulations (DES, LES, DNS), a simple error-measure is the small-scale resolved energy, which scales with both the modeled subgrid-stresses and the numerical truncation errors in many situations. Since this is a scalar measure, it does not carry any information on the anisotropy of the optimal grid-refinement. The purpose of this work is to introduce a new error-measure for turbulence-resolving simulations that is capable of predicting nearly-optimal anisotropic grids. Turbulent channel flow at Reτ ≈ 300 is used to assess the performance of the proposed error-measure. The formulation is geometrically general, applicable to any type of unstructured grid.

3:15PM D8.00006 Vortex Particle-Mesh methods for large scale LES of aircraft wakes, PHILIPPE CHATELAINE, MATHIEU DUPONCHEEL, Univ Catholique de Louvain, YVES MARICHAL, WaPT, Wake Prediction Technologies, GREGOIRE WINCKELMANS, Univ Catholique de Louvain — Vortex methods solve the NS equations in vorticity-velocity formulation. The present Particle-Mesh variant exploits the advantages of a hybrid approach: advection is handled by the particles while the mesh allows the evaluation of the differential operators and the use of fast Poisson solvers (here a Fourier-based solver which allows for unbounded directions and inlet/outlet boundaries). A lifting line approach models the vorticity sources in the flow; its immersed treatment efficiently captures the development of vorticity from thin sheets into 3-D field. Large scale simulations of aircraft wakes (including encounter cases where a following aircraft flies into the wake) are presented, which also demonstrate the performance of the methodology: the adequate treatment of particle distortion, the high-order discretization, and the multiscale subgrid models allow to capture wake dynamics with minimal spurious dispersion and diffusion.

3:28PM D8.00007 Sensitivity Analysis of Chaotic Flow around Two-Dimensional Airfoil, PATRICK BLONIGAN, QIQI WANG, Massachusetts Inst of Tech-MIT, ERIC NIELSEN, NASA Langley Research Center, BORIS DISKIN, National Institute of Aerospace — Computational methods for sensitivity analysis are invaluable tools for fluid dynamics research and engineering design. These methods are used in many applications, including aerodynamic shape optimization and adaptive grid refinement. However, traditional sensitivity analysis methods, including the adjoint method, break down when applied to long-time averaged quantities in chaotic fluid flow fields, such as high-fidelity turbulence simulations. This break down is due to the “Butterfly Effect”; the high sensitivity of chaotic dynamical systems to the initial condition. A new sensitivity analysis method developed by the authors, Least Squares Shadowing (LSS), can compute useful and accurate gradients for quantities of interest in chaotic dynamical systems. LSS computes gradients using the “shadow trajectory”, a phase space trajectory (or solution) for which perturbations to the flow field do not grow exponentially in time. To efficiently compute many gradients for one objective function, we use an adjoint version of LSS. This talk will briefly outline Least Squares Shadowing and demonstrate it on chaotic flow around a Two-Dimensional airfoil.

3:41PM D8.00008 Slow transition of the Osborne Reynolds pipe flow: A direct numerical simulation study, XIAOHUA WU, Royal Military College of Canada, PARVIZ MOIN, Center for Turbulence Research, Stanford University, RONALD J. ADRIAN, School of EMTE, Arizona State University, JON R. BALTZER, Los Alamos National Laboratory — Osborne Reynolds’ pipe transition experiment marked the onset of fundamental turbulence research, yet the precise dynamics carrying the laminar state to fully-developed turbulence has been quite elusive. Our spatially-developing direct numerical simulation of this problem reveals interesting connections with theory and experiments. In particular, during transition the energy norms of localized, weakly finite inlet perturbations grow exponentially, rather than algebraically, with axial distance, in agreement with the edge-state based temporal results of Schneider et al (PRL, 034502, 2007). When inlet disturbance is the core region, helical vortex filaments evolve into large-scale reverse hairpin vortices. The interaction of these reverse hairpins among themselves or with the near-wall flow produces small-scale hairpin packets. When inlet disturbance is near the wall, optimally positioned quasi-spanwise structure is stretched into a Lambda vortex, which grows into a turbulent spot of concentrated small-scale hairpin vortices. Waves of hairpin-like structures were observed by Mullin (Ann. Rev. Fluid Mech., Vol.43, 2011) in their experiment with very weak blowing and suction. This vortex dynamics is broadly analogous to that in the boundary layer bypass transition and in the secondary instability and breakdown stage of natural transition. Further details of our simulation are reported in Wu et al (PNAS, 1509451112, 2015).
2:10PM D9.00001 Taylor bubbles at high viscosity ratios: experiments and numerical simulations

2:23PM D9.00002 CFD-informed unified closure relation for the rise velocity of Taylor bubbles in pipes

2:36PM D9.00003 Detached eddy simulations of Taylor bubbles rising in stagnant liquid columns

2:49PM D9.00004 Numerical modeling of turbulent swirling flow in a multi-inlet vortex nanoprecipitation reactor using dynamic DDES

3:02PM D9.00005 Development of multiphase Navier-Stokes simulation capability for turbulent gas flow over laminar liquid for Cartesian grids
3:15PM D9.00006 New techniques for meshless flow simulation generalizing moving least squares1. NATHANIEL TRASK, MARTIN MAXEY, Brown University — While the Lagrangian nature of SPH offers unique flexibility in application problems, practitioners are forced to choose between compatibility in div/grad operators or low accuracy limiting the scope of the method. In this work, two new discretization frameworks are introduced that extend concepts from finite difference methods to a meshless context: one generalizing the high-order convergence of compact finite differences and another generalizing the enhanced stability of staggered marker-and-cell schemes. The discretizations are based on a novel polynomial reconstruction process that allows arbitrary order polynomial accuracy for both the differential operators and general boundary conditions while maintaining stability and computational efficiency. We demonstrate how the method fits neatly into the ISPH framework and offers a new degree of fidelity and accuracy in Lagrangian particle methods.

1Supported by the Collaboratory on Mathematics for Mesoscopic Modeling of Materials (CM4), DOE Award DE-SC0009247

3:28PM D9.00007 Methods to Prescribe Particle Motion to Minimize Quadrature Error in Meshfree Methods. JEREMY TEMPLETON, LINDSAY ERICKSON, KARLA MORRIS, DAVID POLIAKOFF, Sandia National Laboratories — Meshfree methods are an attractive approach for simulating material systems undergoing large-scale deformation, such as spray break up, free surface flows, and droplets. Particles, which can be easily moved, are used as nodes and/or quadrature points rather than a relying on a fixed mesh. Most methods move particles according to the local fluid velocity that allows for the convection terms in the Navier-Stokes equations to be easily accounted for. However, this is a trade-off against numerical accuracy as the flow can often move particles to configurations with high quadrature error, and artificial compressibility is often required to prevent particles from forming undesirable regions of high and low concentrations. In this work, we consider the other side of the trade-off: moving particles based on reducing numerical error. Methods derived from molecular dynamics show that particles can be moved to minimize a surrogate for the solution error, resulting in substantially more accurate simulations at a fixed cost. Sandia National Laboratories is a multiprogram laboratory operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the United States Department of Energy’s National Nuclear Security Administration under contract DE-AC04-94AL85000.

3:41PM D9.00008 A balanced-force conservative volume-of-fluid method for simulating two-phase flows on unstructured grids1. CHRISTOPHER IVEY, PARVIZ MOIN, Center for Turbulence Research, Stanford University — A balanced-forced conservative volume-of-fluid method for simulating two-phase flows on unstructured grids is presented. The two-phase Navier-Stokes equations are solved using a median-dual-partitioned colocated node-centered finite-volume discretization and a specialized fractional-step method. Conservative mass and momentum convection fluxes are calculated using a novel volume-of-fluid method. Accurate interface normals and curvatures are calculated on the non-convex median-dual mesh using the recently proposed embedded height-function technique. Spurious currents are minimized using a balanced-force algorithm and the continuum-surface force description of surface tension. The results of several two- and three-dimensional benchmark test cases on various unstructured meshes demonstrate the effectiveness of the proposed proposed two-phase flow solver.

1Supported by the DOE Computational Science Graduate Fellowship (grant number DE-FG02-97ER25308) and the Stanford Graduate Fellowship

3:54PM D9.00009 High-Order Discontinuous Galerkin Level Set Method for Interface Tracking and Re-Distancing on Unstructured Meshes1. PATRICK GREENE, ROBERT NOURGALIEV, SAM SCHOFIELD, Lawrence Livermore National Laboratory — A new sharp high-order interface tracking method for multi-material flow problems on unstructured meshes is presented. The method combines the marker-tracking algorithm with a discontinuous Galerkin (DG) level set method to implicitly track interfaces. DG projection is used to provide a mapping from the Lagrangian marker field to the Eulerian level set field. For the level set re-distancing, we developed a novel marching method that takes advantage of the unique features of the DG representation of the level set. The method efficiently marches outward from the zero level set with values in the new cells being computed solely from cell neighbors. Results are presented for a number of different interface geometries including ones with sharp corners and multiple hierarchical level sets. The method can robustly handle the level set discontinuities without explicit utilization of solution limiters. Results show that the expected high order (3rd and higher) of convergence for the DG representation of the level set is obtained for smooth solutions on unstructured meshes. High-order re-distancing on irregular meshes is a must for applications were the interfacial curvature is important for underlying physics, such as surface tension, wetting and detonation shock dynamics.

1This work was performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract DE-AC52-07NA27344. Information management release number LLNL-ABS-675636.

4:07PM D9.00010 High-order accurate multi-phase simulations: building blocks and what's tricky about them1. FLORIAN KUMMER, TU Darmstadt / Chair of Fluid Dynamics — We are going to present a high-order numerical method for multi-phase flow problems, which employs a sharp interface representation by a level-set and an extended discontinuous Galerkin (XDG) discretization for the flow properties. The shape of the XDG basis functions is dynamically adapted to the position of the fluid interface, so that the spatial approximation space can represent jumps in pressure and kinks in velocity accurately. By this approach, the ‘h^p-convergence’ property of the classical discontinuous Galerkin (DG) method can be preserved for the low-regularity, discontinuous solutions, such as those appearing in multi-phase flows. Within the past years, several building blocks of such a method were presented: this includes numerical integration on cut-cells, the spatial discretization by the XDG method, precise evaluation of curvature and level-set algorithms tailored to the special requirements of XDG-methods. The presentation covers a short review on these building-block and their integration into a full multi-phase solver. A special emphasis is put on the discussion of the several pitfalls one may expire in the formulation of such a solver.

1German Research Foundation
2:10PM D10.00001 Topological transitions in unidirectional flow of nematic liquid crystal. LINDA CUMMINGS, New Jersey Institute of Technology, THOMAS ANDERSON, Caltech, ENSELA MEMA, LOU KONDIC, New Jersey Institute of Technology — Recent experiments by Sengupta et al. (Phys. Rev. Lett. 2013) revealed interesting transitions that can occur in flow of nematic liquid crystal under carefully controlled conditions within a long microfluidic channel of rectangular cross-section, with homotropically anchoring at the walls. At low flow rates the director field of the nematic adopts a configuration that is dominated by the surface anchoring, being nearly parallel to the channel height direction over most of the cross-section; but at high flow rates there is a transition to a flow-dominated state, where the director configuration at the channel centerline is aligned with the flow (perpendicular to the channel height direction). We analyze simple channel-flow solutions to the Leslie-Ericksen model for nematics. We demonstrate that two solutions exist, at all flow rates, but that there is a transition between the elastic free energies of these solutions: the anchoring-dominated solution has the lowest energy at low flow rates, and the flow-dominated solution has lowest energy at high flow rates.

1 NSF DMS 1211713

2:23PM D10.00002 Quasiparallel flow of a binary gas mixture: the Stefan tube revisited. S.J.S. MORRIS, Mechanical Engineering, University of California, Berkeley — Placed in the bottom of a vertical tube open at the top, volatile liquid (species 1) evaporates at a rate set by diffusion of vapour through the carrier gas (species 2). In the textbook solution, due to J. Stefan, species 2 is assumed to be stationary, but numerical solutions of the governing equations show that species 2, in fact, recirculates (Mills and Chang 1933; and references therein). But although Stefan’s solution is based on an incorrect assumption, the same numerical solutions show that it predicts the evaporation rate to within a few percent (Mills and Chang, below eq.12). Assuming the ratio $L/a$ of the tube to be constant, we use a variational approach to give an elementary solution determining the velocity profiles for each species, including the effect of slip. It is shown that, in the limit as $L/a \to \infty$, the Stefan solution correctly determines the total evaporation rate; this conclusion is independent of the precise form of the boundary condition placed on the species velocities at the tube wall.

2:36PM D10.00003 Boundary integral simulations of dissolving drops in segmented two-phase flows. ARUN RAMCHANDRAN, THOMAS LEARY, University of Toronto — Recent years have seen an upsurge in the literature reporting the microfluidic measurement of the kinetics of ‘fast’ gas-liquid reactions by recording the shrinkage of bubbles in segmented flows of these gas-liquid combinations in microfluidic channels. A critical aspect of the data analysis in these experiments is the knowledge of how dissolution influences the velocity field in the liquid slug, and hence, the mass transport characteristics. Unfortunately, there is no literature on this connection for dissolving bubbles. In this paper we present boundary integral simulations. The effects of the dissolution rate on the film thickness and the inter-drop separation are examined as a function of the capillary number and the viscosity ratio. The results demonstrate that dissolution can enhance the degree of mixing appreciably from one slug to the next. A curious result is that the film thickness and the droplet separation distance can change significantly beyond a critical capillary number, producing flow patterns completely different from those known for the unsegmented case. These results will guide the selection of operating regimes that enable convenient interpretation of data from experiments to deduce kinetic constants.

2:49PM D10.00004 Elastic deformations in a Hele-Shaw cell driven by local non-homogeneities of fluid properties. SHIMON RUBIN, AMIR GAT, MORAN BERCOVICI, Technion Israel Institute of Technology — We consider a Hele-Shaw chamber with an elastic top plate, and study the effect of spatial variations in fluid properties on deformations of the plate. Specifically, we present analytical solutions for the pressure and depth-averaged flow field for axially-symmetric variations in slip velocity, viscosity, slip length, and channel height. We then focus on electroosmotic flow, which may be a practical method for obtaining gradients in slip velocity via non-uniform zeta-potential patterning of the surface. We derive an equation which relates elastic deformations of a Kirchhoff-Love plate with shear gradients in zeta potential, and obtain an analytical solution for the zeta potential distribution which gives rise to a local Gaussian deformation. Owing to the fact that any surface can be represented by superposition of Gaussians, we are thus able to determine the zeta potential necessary for creation of arbitrary deformations.

3:02PM D10.00005 Blood Perfusion in Microfluidic Models of Pulmonary Capillary Networks: Role of Geometry and Hematocrit. HAGIT STAUBER, Technion-IIT, DAN WAISMAN, Department of Neonatology Carmel Medical Center Faculty of Medicine Technion IIT, JOSUE SZNITMAN, Technion-IIT, TECHNION-IIT TEAM, DEPARTMENT OF NEONATOLOGY CARMELO MEDICAL CENTER AND FACULTY OF MEDICINE — TECHNION IIT COLLABORATION, † Microfluidic platforms are increasingly used to study blood microflows at true physiological scale due to their ability to overcome manufacturing obstacle of complex anatomical morphologies, such as the organ-specific architectures of the microcirculation. In the present work, we utilize microfluidic platforms to devise in vitro models of the underlying pulmonary capillary networks (PCN), where capillary lengths and diameters are similar to the size of RBCs (∼5-10 µm). To better understand flow characteristics and dispersion of red blood cells (RBCs) in PCNs, we have designed microfluidic models of alveolar capillary beds inspired by the seminal “sheet flow” model of Fung and Sobin (1969). Our microfluidic PCNs feature confined arrays of staggered pillars with diameters of ∼5, 7, and 10 µm, mimicking the dense structure of pulmonary capillary meshes. The devices are perfused with suspensions of RBCs at varying hematocrit levels under different flow rates. Whole-field velocity patterns using micro-PIV and single-cell tracking using PTV are obtained with fluorescently-labelled RBCs and discussed. Our experiments deliver a real-scale quantitative description of RBC perfusion characteristics across the pulmonary capillary microcirculation.

3:15PM D10.00006 Imbibition of “Open Capillary”: Fundamentals and Applications. MARIE TANI, Department of Physics, Faculty of Science, Ochanomizu University, RYUJI KAWANO, Division of Biotechnology and Life Science, Tokyo University of Agriculture and Technology, KOKI KAMIYA, Artificial Cell Membrane Systems Group, Kanagawa Academy of Science and Technology (KAST), KO OKUMURA, Department of Physics, Faculty of Science, Ochanomizu University — Control or transportation of small amount of liquid is one of the most important issues in various contexts including medical science or pharmaceutical industries to fuel delivery. We studied imbibition of “open capillary” both experimentally and theoretically, and found simple scaling laws for both statics and dynamics of the imbibition, similarly as that of imbibition of capillary tubes. Furthermore, we revealed the existence of “precursor film,” which developed ahead of the imbibing front, and the dynamics of it is described well by another scaling law for capillary rise in a corner [1]. Then, to show capabilities of open capillaries, we demonstrated two experiments by fabricating micro mixing devices to achieve (1) simultaneous multi-color change of the Bromothymol blue (BTB) solution and (2) expression of the green florescent protein (GFP) [2]. [1] A. Ponomarenko, D. Quere and C. Clanet, J. Fluid Mech., 666, 146 (2011). [2] M. Tani, R. Kawano, K. Kamiya and K. Okumura, Sci. Rep. 5, 10263 (2015).

1 This research was partly supported by ImPACT Program of Council for Science, Technology and Innovation (Cabinet Office, Government of Japan). M. T. is supported by the Japan Society for the Promotion of Science Research Fellowships for Young Scientists.
3:28PM D10.00007 Experimental investigation of non-Newtonian/Newtonian liquid-liquid flow in microchannel, EYNAGELIA-PANAGIOTA ROUMPEA, WEHELIYE WEHELIYE, MAXIME CHINAUD, PANAGIOTA ANGELI, Department of Chemical Engineering, University College London, Torrington Place, London, WC1E 7JE, LYES KAHOUADJI-COLLABORATION1, OMAR. K. MATAR-COLLABORATION2 — Flow of an organic phase and an aqueous non-Newtonian solution was investigated experimentally in a quartz microchannel with ID. 200 µm. The aqueous phase was a glycerol solution where 1000 and 2000 ppm of xanthan gum was added while the organic phase was silicone oil with 155 and 5 cSt viscosity. The two phases were brought together in a T-junction and their flowrates varied from 0.3 to 6 ml/hr. High speed imaging was used to study the characteristics of the plugs and the effect of the liquid properties on the flow patterns while a two-colour micro-PIV technique was used to investigate velocity profiles and circulation patterns within the plugs. The experimental results revealed that plug length was affected by both flowrate and viscosity. In all cases investigated, a film of the continuous phase always surrounded the plugs and its thickness was compared with existing literature models. Circulation patterns inside plugs were obtained by subtracting the plug velocity and found to be depended on the plug length and the amount of xanthan gum in the aqueous phase. Finally, the dimensionless circulation time was calculated and plotted as a function of the plug length.

1Department of Chemical Engineering South Kensington Campus Imperial College London SW7 2AZ
2Department of Chemical Engineering South Kensington Campus Imperial College London SW7 2AZ

3:34PM D10.00008 Destabilization of highly viscous fluid threads in complex microgeometries, THOMAS CUBAUD, Stony Brook University — High-viscosity multiphase flows in microchannels encompass a broad range of fluid phenomena, including self-lubrication and viscous buckling instabilities. Here, a series of experiments is conducted to study the dynamic response of miscible fluid threads to a change in carrier flow velocity due to varying microgeometries. The structural stability of core-annular flows is systematically investigated in simple and complex microchannels, such as square, bifurcating, and corrugated channels, from low to high flow rates of injection and for a variety of fluid viscosities. Focus is on flow regimes of practical interest for the improvement of mixing and separation processes between fluids having large viscosity contrasts at the small scale.

This work is supported by NSF (CBET-1150389)

3:35PM D10.00009 The motion of long drops in rectangular capillaries at low capillary numbers, HARRIS WONG, SAI RAO, Louisiana State University — The immiscible liquid-liquid drop flow in rectangular capillaries has found extensive industrial applications. However, the flow patterns and pressure-flow rate relations are not well understood. We study the steady motion of a long drop of length LW (L>>1) in a rectangular microchannel of width W and height BW (B>>1). The drop is moving at a velocity U such that the capillary number Ca=mU/L <<1, where m is the viscosity of the carrier liquid and L is the interfacial tension. The drop is non-wetting so that a carrier liquid film separates the drop from the channel wall. We find that the carrier liquid either pushes the drop (plug flow) or bypasses the drop through corner channels (corner flow). When LCa1/3 >>1, the plug flow dominates, whereas the corner flow dominates when LCa1/3 <<1. The plug flow and the corner flow are coupled through the corner interface. Hence, when the corner flow dominates, the carrier liquid bypasses the drop and drops the drop fluid faster than the drop velocity. To conserve mass, the drop fluid circulates from the front to the back of the drop along the center region. The pressure-flow rate relation is linear when LCa1/3 <<1 or >1, and is nonlinear when LCa1/3 ∼ 1. The coupled flow is studied for B = 1, 1.2, 1.5, and 2, and for viscosity ratio R = 0.001 to 100, where R is the ratio of drop viscosity to carrier liquid viscosity.

4:07PM D10.00010 Broadband light based optoelectric tweezers, AVANISH MISHRA, KATHERINE CLAYTON, STEVE WERELEY, Purdue University — Trapping, sorting and transport of particles are fundamental operations in microfluidic platforms. However, very few methods exist that can dynamically trap and manipulate particles with high spatial resolution and accuracy. Recently, a new set of methods have emerged that can trap and sort particles by optically controlling electrokinetic effects. Rapid Electrokinetic Patterning (REP) is such an emerging optoelectric technique. It utilizes a laser activated electrothermal (ET) vortex and particle-electrode interactions for trapping particles. Trapped particles can be translated by optically steering the laser or by moving the trapping chamber. Previously demonstrated applications of REP have utilized a 1064 nm infrared laser, integrated in an inverted microscope, to create the necessary temperature rise for producing the ET flow. Use of an external laser for REP trapping is expensive and time intensive to integrate, making it difficult to design a portable REP system. Using experiments and simulations, we show that a non-coherent incandescent broadband light source can be used for REP trapping and manipulation. This allows for a microscope with a broadband lamp to be used for REP trapping without integrating an external laser.

Sunday, November 22, 2015 2:10PM - 4:20PM — Session D11 Convection and Buoyancy-Driven Flows: Analytic Techniques

2:10PM D11.00001 How does the diffusion fish swim?, GUNNAR PENG, DAMTP, University of Cambridge

2:23PM D11.00002 Optimizing exit times, JEAN-LUC THIFFEAULT, University of Wisconsin, FLORENCE MARCOTTE, Ecole Normale Superieure, CHARLES DOERING, University of Michigan, WILLIAM YOUNG, Scripps Institution of Oceanography — A heat exchanger can be modeled as a closed domain containing an incompressible fluid. The fluid has some temperature distribution obeying the advection-diffusion equation, with zero temperature boundary conditions at the walls. The goal is then to start from some initial positive temperature rise for producing the ET flow. Use of an external laser for REP trapping is expensive and time intensive to integrate, making it difficult to design a portable REP system. Using experiments and simulations, we show that a non-coherent incandescent broadband light source can be used for REP trapping and manipulation. This allows for a microscope with a broadband lamp to be used for REP trapping without integrating an external laser.

This work was completed at the 2015 WHOI GFD Program, which is supported by the National Science Foundation and the Office of Naval Research.

2:33PM D11.00003 Destabilization of highly viscous fluid threads in complex microgeometries, THOMAS CUBAUD, Stony Brook University — High-viscosity multiphase flows in microchannels encompass a broad range of fluid phenomena, including self-lubrication and viscous buckling instabilities. Here, a series of experiments is conducted to study the dynamic response of miscible fluid threads to a change in carrier flow velocity due to varying microgeometries. The structural stability of core-annular flows is systematically investigated in simple and complex microchannels, such as square, bifurcating, and corrugated channels, from low to high flow rates of injection and for a variety of fluid viscosities. Focus is on flow regimes of practical interest for the improvement of mixing and separation processes between fluids having large viscosity contrasts at the small scale.

This work is supported by NSF (CBET-1150389)
2:36PM D11.00003 ABSTRACT WITHDRAWN —

2:49PM D11.00004 Non-Boussinesq Rayleigh-Benard linear stability¹, THIERRY AL-BOUSSIÈRE, YANICK RICARD, Laboratoire de Géologie de Lyon, université de Lyon, CNRS, ENS de Lyon, France — The simplest Rayleigh-Benard configuration consists in a horizontal fluid layer maintained at a higher temperature on the under side, with no shear stress on its boundary. In the Boussinesq approximation, Rayleigh obtained an analytic value, $2\pi^{5/4}/\Lambda$, for the critical stability threshold of a dimensionless parameter which now bears his name. There are two ways to go away from the Boussinesq approximation: when there is a significant temperature difference across the layer compared to the average thermodynamic temperature, or when gravity creates a significant compression of the fluid near the bottom. We have determined an approximate analytical expression for the critical Rayleigh number depending on the two non-Boussinesq causes. We have also determined the critical threshold for the intermediate model called the 'anelastic liquid approximation' in which the adiabatic temperature gradient is taken into account, while density fluctuations are assumed to be solely due to temperature fluctuations. It is found that a small product $\alpha T$ (thermal expansion coefficient times temperature) does not make the anelastic liquid approximation any better, for a Gruneisen parameter close to unity.

¹TA acknowledges support from the PNP program of INSU (CNRS)

3:02PM D11.00005 Buoyancy effects in a wall jet over a heated horizontal plate, RAMON FERNANDEZ-FERIA, FRANCISCO CASTILLO-CARRASCO, Universidad de Malaga (Spain) — A similarity solution of the boundary layer equations for a jet on a heated horizontal surface taking into account the coupling of the temperature and velocity fields by buoyancy is described. It exists for any positive value of $\Lambda = Gr/Re^2$, characterizing this coupling between natural and forced convection over the horizontal plate, i.e., only when the plate temperature is larger than the ambient one. The flow structure is qualitatively very different from the well known Glauber's similarity solution for a wall jet without buoyancy effects ($\Lambda = 0$): basically coincides for both a radially spreading jet and a two-dimensional jet, and the maximum of the horizontal velocity increases as the jet spreads over the surface, with the power $1/5$. The similarity solution is checked by solving numerically the boundary layer equations for a jet with uniform velocity and temperature emerging from a slot of height $\delta$ and radius $r_0$ (in the radial case). An approximate, analytical similarity solution near the jet exit is also found that helps to start the numerical integration. The similarity solution is reached for any set of the non-dimensional parameters governing the problem provided that the plate is heated ($\Lambda > 0$).

3:15PM D11.00006 Convective dissolution in partially miscible systems: classification of the effect of reactions, V. LOODTS, C. THOMAS, L. RONGY, A. DE WIT, Université libre de Bruxelles (ULB), Nonlinear Physical Chemistry Unit, 1050 Brussels, Belgium — Dissolution-driven convection in partially miscible systems has regained much interest in the context of CO₂ sequestration. A buoyantly unstable density stratification can build up upon dissolution of a species into the host fluid phase, thereby developing convection. Chemical reactions can impact such convection as they affect concentrations and thus the density of the host phase. We theoretically classify the effects of reactions on the convective instability as a function of the contributions to density and diffusion coefficients of the chemical species involved. To do so, we compute the reaction-diffusion density profiles in the host phase and assess their stability with regard to buoyancy-driven convection by a linear stability analysis. The buoyancy-driven instability grows faster when the product of the reaction contributes sufficiently more to density than the initially dissolved reactant. We illustrate this by experimental results showing that reactions accelerate the development of buoyancy-driven fingering during the convective dissolution of CO₂ into aqueous solutions of alkali hydroxides.

3:28PM D11.00007 Nonlinear convection in unbounded vertical channels, RISHAD SHAH-MUROV, LAYACHI HADJI, University of Alabama — We investigate the linear and weakly nonlinear solutions to a convection problem that was first studied by Ostroumov in 1947. The problem is to find the steady-state solutions of the equations governing convective motion in an infinite vertical fluid layer that is heated from below. Ostroumov’s linear stability analysis yields instability threshold conditions that are characterized by zero wavenumber for the Fourier mode in the vertical direction and by eigenfunctions that are independent of the vertical coordinate. Thus, any undertaking at determining the supercritical nonlinear solutions and their stability through a small amplitude expansion fails. This failure is attributed to the fact that the nonlinear interaction of the linear modes vanish identically. In this paper, we put forth exact and stable similarity type solutions to the Ostroumov problem. These solutions are characterized by the same linear threshold conditions as Ostroumov’s solutions. Moreover, we are able to extend the analysis to the supercritical regime through a small amplitude analysis to obtain steady two-dimensional solutions for a small range of Prandtl numbers. These solutions are found to be stable to general two-dimensional, time-dependent disturbances.

3:41PM D11.00008 Pattern selection in ternary mushy layers, PETER GUBA, Comenius University, DANIEL ANDERSON, George Mason University — We consider finite-amplitude convection in a mushy layer during the primary solidification of a ternary alloy. A previous linear theory identified, for the case of vanishing latent heat, solute rejection and background solidification, a direct mode of convective instability when all the individual stratifying agencies (thermal and two solutal) were statically stabilizing. The physical mechanism behind this instability was attributed to the local-phase-change effect on the net solute balance through the liquid-phase solutal diffusivity. A weakly nonlinear development of this instability is investigated in detail. We examine the stability of two-dimensional roll, and three-dimensional square and hexagonal convection patterns. The amplitude evolution equations governing roll/square and roll/hexagon convection show the same. We find that rolls, squares or hexagons can be nonlinearly stable, depending on the relative importance of a number of physical effects as reflected in the coefficients of the amplitude equations. The results for a special case are found to isolate a purely double-diffusive phase-change mechanisms of pattern selection. Subcritical behaviour is identified inside the domain of individual static stability.

3:54PM D11.00009 Using Covariant Lyapunov Vectors to Build a Physical Understanding of Spatiotemporal Chaos in Rayleigh-Bénard Convection¹, MU XI, MARK PAUL, Virginia Tech — We explore the high-dimensional spatiotemporal chaos of Rayleigh-Bénard convection using covariant Lyapunov vectors. We integrate the three-dimensional and time dependent Boussinesq equations for a convection layer for very long-times and for a range of Rayleigh numbers. We simultaneously integrate many copies of the tangent space equations in order to compute the covariant Lyapunov vectors. We explore chaotic dynamics with a fractal dimension of nearly 50 and we compute over 150 covariant Lyapunov vectors. Using the Lyapunov vectors we quantify the hyperbolicity of the dynamics, the degree of Oseledec splitting, and explore the temporal, spatial, and spectral dynamics of the Lyapunov vectors. Our results indicate that the dynamics undergoes a hyperbolic to non-hyperbolic transition as the Rayleigh number is increased. Our results yield that all of the Lyapunov vectors computed have near tangencies with neighboring vectors. A closer look at the vectors suggests that the dynamics are composed of physical modes that are connected with tangled spurious modes that extend to all of the covariant Lyapunov vectors we have computed.

¹This research was funded by NSF grant no. DMS-1125234. The computations were done using resources from the Virginia Tech ADVANCED Research Computing center.
destabilizing effect of elasticity at moderate Reynolds numbers. The amplitude of this forcing decreases at higher Weissenberg number and also with increasing polymer concentration. The results demonstrate the transition at moderate Reynolds number, the FENE-P constitutive equations, and both the polymer concentration and Weissenberg number are varied in order to assess their effect on and transition to turbulence in viscoelastic Couette flow are studied using direct numerical simulations (DNS). Viscoelasticity is modeled using viscoelastic flow models such as FENE-P (used in this study). This presentation provides an overview of the structure of EIT in 2D channel flows for Reynolds numbers ranging from $Re = 10$ to 100 and for 3D simulations up to $Re_{tau} = 300$. For flows below the Newtonian critical Reynolds number, EIT increases the drag. For higher Reynolds numbers, EIT is summarised to be the energetic bound of Maximum Drag Reduction (MDR), the asymptotic state of the instability in polymer solutions. This phenomenon of EIT at low Reynolds numbers ($Re < 600$) highlights a backward energy transfer from the small scale polymer dynamics to larger flow scales. Similar dynamics is identified at higher Reynolds numbers, which could explain why polymer flows never become fully laminar.

The authors acknowledge computational resources from CÉCI (F.R.S.-FNRS grant No.2.5020.11), the PRACE infrastructure, and the Vermont Advanced Computing Core.

2:23PM D12.00002 The maximum drag reduction asymptote, GEORGE H. CHOUEIRI, BJORN HOF, Institute of Science and Technology Austria — Addition of long chain polymers is one of the most efficient ways to reduce the drag of turbulent flows. Already very low concentration of polymers can lead to a substantial drag and upon further increase of the concentration the drag reduces until it reaches an empirically found limit, the so called maximum drag reduction (MDR) asymptote, which is independent of the type of polymer used. We here carry out a detailed experimental study of the approach to this asymptote for pipe flow. Particular attention is paid to the recently observed state of elasto-inertial turbulence (EIT) which has been reported to occur in polymer solutions at sufficiently high shear. Our results show that upon the approach to MDR Newtonian turbulence becomes marginalized (hibernation) and eventually completely disappears and is replaced by EIT. In particular, spectra of high Reynolds number MDR flows are compared to flows at high shear rates in small diameter tubes where EIT is found at $Re < 100$.

The research leading to these results has received funding from the People Programme (Marie Curie Actions) of the European Union’s Seventh Framework Programme (FP7/2007-2013) under REA grant agreement n [291734].

2:36PM D12.00003 Spatial-Temporal dynamics of Newtonian and viscoelastic turbulence, SUNG-NING WANG, MICHAEL GRAHAM, University of Wisconsin - Madison — Introducing a trace amount of polymer into liquid turbulent flows can result in substantial reductions in friction drag. This phenomenon has been widely used in fluid transport, such as the Alaska crude oil pipeline. However, the mechanism is not well understood. We conduct direct numerical simulations of Newtonian and viscoelastic turbulence in large domains, in which the flow shows different characteristics in different regions. In some areas the drag is low and vortex motions are quiescent, while in other areas the drag is higher and the motions are more active. To identify these regions, we apply a statistical method, k-means clustering, which partitions the observations into k clusters by assigning each observation to its nearest centroid. The resulting partition maximizes the between-cluster variance. In the simulations, the observations are the instantaneous wall shear rate. Regions with different levels of drag are automatically identified by the partitioning algorithm. We find that the velocity profiles of the centroids exhibit characteristics similar to the individual coherent structures observed in minimal domain simulations. In addition, as viscoelasticity increases, polymer stretch becomes strongly correlated with wall shear stress.

This work was supported by NSF grant CBET-1510291.

2:49PM D12.00004 Scaling of energy amplification in the weak and strong elastic limits of viscoelastic shear flows, ISMAIL HAMEDUDDIN, TAMER ZAKI, DENNICE GAYME, The Johns Hopkins University — We investigate energy amplification in viscoelastic parallel shear flows in terms of the steady-state variance maintained in the velocity and polymer stresses when either quantity is excited with white noise. We derive analytical expressions that show how this amplification scales with both Reynolds (Re) and Weissenberg (Wi) numbers. The analysis focuses on the streamwise-constant fields in the limits of high and low velocity. By introducing stochastic forcing in both the velocity and the polymer stress dynamics, we show that at low elasticity the scaling retains a form similar to the well-known $O(Re^3)$ relationship but with an added elastic correction. At high elasticity, however, the scaling is $O(Wi^3)$ with a viscous correction. Our results demonstrate that energy amplification in a viscoelastic flow can be considerable even at low Re, correlating well with recent observations of elastic turbulence in creeping flows. We also note that forcing in the polymer stress dynamics can contribute significantly to the energy amplification.

3:02PM D12.00005 Streak instability in viscoelastic Couette flow, LUCA BIANCOFIORE, Imperial College London, LUCA BRANDT, KTH - Stockholm, TAMER ZAKI, John Hopkins University — The secondary instability of streaks and transition to turbulence in viscoelastic Couette flow are studied using direct numerical simulations (DNS). Viscoelasticity is modeled using the FENE-P constitutive equations, and both the polymer concentration and Weissenberg number are varied in order to assess their effect on transition at moderate Reynolds number, $Re = 400$. The base streaks are obtained from nonlinear simulations of the Couette flow response to a streamwise vortex, and can be classified as quasi-Newtonian streaks according to the terminology introduced by Page & Zaki (2014). At every streak amplitude of interest, harmonic forcing is introduced to trigger the secondary instability and breakdown to turbulence. The critical amplitude of this forcing decreases at higher Weissenberg number and also with increasing polymer concentration. The results demonstrate the destabilizing effect of elasticity at moderate Reynolds numbers.
The flowfield is treated as inviscid but rotational. Linear solutions are obtained using a nonuniform coordinate transformation that converts the free surface boundary condition into a modified Bessel equation. Velocity components are expanded in modified Bessel functions of the first kind of order zero.

Temporal, there is an increase in low frequency fluctuations (and decrease in high frequency velocity fluctuations).

4:07PM D12.00010 Energy transfer and drag reduction in elasto-inertial turbulence laden with elongated contravariant and covariant polymers, KIYOSI HORIUTI, Dept. Mechano-Aerospace Engineering, Tokyo Institute of Technology, Tokyo, Japan — We study elongation and energy-transfer process of polymers released in the homogeneous nonaffinely shear flow. By connecting mesoscopic Brownian description of elastic dumbbells to macroscopic description for the solvent (DNS). The dumbbells are allowed to be advected either affinely with the macroscopically-imposed deformation (contravariant) or completely non-affinely (covariant). We consider the elasto-inertial regime in which the relaxation time of polymer is in the order of the eddy turnover time. Highly-elongated contravariant polymers remove more energy from the large scales than they can dissipate and transfer the excess energy back into the solvent as in P.C. Valente et al. (2014). By deriving the approximate solution of the constitutive equation for the polymer stress (Horiuti et al. 2013), we identified the term responsible for causing this transfer. The skewness of the strain-rate tensor $(\mathcal{S}_{ik} \mathcal{S}_{jk})$ in the elasto-inertial production term transfers the elastic energy of the polymer from the smallest scales of the solvent and increases the dissipation. In the covariant polymers, this trend is reversed and leads to enhancement of drag reduction, in accordance with the hypothesis that stretched polymers may behave like rods and exhibit rigidity (de Gennes 1986).

3:15PM D12.00006 Elastic Turbulence in Parallel Shear Flows at Low Re$^1$, BOYANG QIN, PAULO ARRATIA, Univ of Pennsylvania — In this talk, the flow of a viscoelastic fluid is experimentally investigated using particle velocity methods in a microfluidic device. The device is a long and straight microchannel that is 100-μm wide and deep; the channel has a short 3-mm region that contains an linear array of cylinders (perturbation region) followed by a 3-cm long and straight region (parallel shear region). We find that, both in the wake of the cylinders and far downstream in the parallel shear region, the flow is excited over a broad range of frequencies and wavelengths. These velocity fluctuations are consistent with the main features that characterize elastic turbulence at low Re. In the wake of the cylinder, we find that the decay in velocity temporal and spatial spectra is approximately -2.7 and -3.0, respectively. These fluctuations persist far downstream in the parallel shear flow region, but with a different power law for the spatial spectrum. -2.0. Our velocimetry measurements indicate that, as the flow moves from the perturbation to the parallel shear region, there is substantial decrease in large length scale fluctuations. Temporally, there is an increase in low frequency fluctuations (and decrease in high frequency velocity fluctuations).

$^1$This work is supported by NSF-CBET-1336171.

3:28PM D12.00007 Transition to asymmetry in pipe flow of shear-thinning fluids: a linear instability?, DAVID DENNIS, CHAOFAN WEN, ROBERT POOLE, University of Liverpool — Previous studies of shear-thinning fluids in pipe flow discovered that, although the time-averaged velocity profile was axisymmetric when the flow was laminar or fully turbulent, contrary to common perception it was asymmetric in the laminar-turbulent transition regime. We reveal that in fact the asymmetry is responsible for returning symmetry to the flow (i.e. the opposite to what was previously believed), which explains why the fully turbulent case is axisymmetric. The experiment was performed using an aqueous solution of xanthan gum (0.15%), an essentially inelastic shear-thinning polymer solution. Stereoscopic particle image velocimetry was used to measure the 3C velocity vectors over the entire circular cross-section of the pipe. 220 pipe diameters downstream of the inlet. The deviation from the axisymmetric laminar state is observed to develop in the form of a supercritical bifurcation with square-root dependence on Reynolds number. The asymmetry is non-hysteretic and reversible, not only having a favoured location, but a preferred route between axisymmetry and asymmetry, which it adheres to regardless of the direction of the transition.

3:41PM D12.00008 Investigation of the required length for fully developed pipe flow with drag-reducing polymer solutions, YASAMAN FARSIANI, BRIAN ELBING, Oklahoma State University — Adding trace amounts of long chain polymers into a liquid flow is known to reduce skin friction drag by up to 80%. While polymer drag reduction (PDR) has been successfully implemented in internal flows, diffusion and degradation have limited its external flow applications. A weakness in many previous PDR studies is that there was no characterization of the polymer being injected into the turbulent boundary layer, which can be accomplished by testing a sample in a pressure-drop tube. An implicit assumption in polymer characterization is that the flow is fully developed at the differential pressure measurement. While available data in the literature shows that the entry length to achieve fully developed flow increases with polymeric solutions, it is unclear how long is required to achieve fully developed flow for non-Newtonian turbulent flows. In the present study, the pressure-drop is measured across a 1.05 meter length section of a 1.04 cm inner diameter pipe. Differential pressure is measured with a pressure transducer for different entry lengths, flow and polymer solution properties. This presentation will present preliminary data on the required entrance length as well as characterization of polymer solution an estimate of the mean molecular weight.

3:54PM D12.00009 Natural transition to turbulence in polymeric channel flow, SANG JIN LEE, Imperial College London, TAMER ZAKI, Johns Hopkins University — Natural transition in viscoelastic channel flow is investigated using direct numerical simulations (DNS), where the polymer is modeled using the FENE-P constitutive equations. The computations capture the amplification of the primary two-dimensional Tollmien-Schlichting (TS) waves, their secondary instability and ultimately the onset of turbulence. Various Weissenberg numbers (Wi) are simulated in order to assess the influence of elasticity. As Wi is increased, the primary TS waves initially become more linearly unstable, but are subsequently stabilized at higher Wi. This trend suggests that elasticity can either promote or delay transition to turbulence, and the DNS substantiate this prediction. In order to isolate the effect of the polymer on the secondary instability process, simulations are performed for a set of elastic parameters where the primary TS wave has the same linear growth rate as the Newtonian configuration. As a result, while the linear disturbance amplification is similar in the viscoelastic and Newtonian flows, the nonlinear saturated state of the TS waves differs in the two cases, as well as their secondary instability and breakdown to turbulence. The changes in the transition process are examined by analyzing the disturbance energy budget and spectra.

4:07PM D12.00010 Energy transfer and drag reduction in elasto-inertial turbulence laden with elongated contravariant and covariant polymers, KIYOSI HORIUTI, Dept. Mechano-Aerospace Engineering, Tokyo Institute of Technology, Tokyo, Japan — We study elongation and energy-transfer process of polymers released in the homogeneous nonaffinely shear turbulence by connecting mesoscopic Brownian description of elastic dumbbells to macroscopic description for the solvent (DNS). The dumbbells are allowed to be advected either affinely with the macroscopically-imposed deformation (contravariant) or completely non-affinely (covariant). We consider the elasto-inertial regime in which the relaxation time of polymer is in the order of the eddy turnover time. Highly-elongated contravariant polymers remove more energy from the large scales than they can dissipate and transfer the excess energy back into the solvent as in P.C. Valente et al. (2014). By deriving the approximate solution of the constitutive equation for the polymer stress (Horiuti et al. 2013), we identified the term responsible for causing this transfer. The skewness of the strain-rate tensor $(\mathcal{S}_{ik} \mathcal{S}_{jk})$ in the elasto-inertial production term transfers the elastic energy of the polymer from the smallest scales of the solvent and increases the dissipation. In the covariant polymers, this trend is reversed and leads to enhancement of drag reduction, in accordance with the hypothesis that stretched polymers may behave like rods and exhibit rigidity (de Gennes 1986).
2:23PM D13.00002 Extreme Wave Impact on a Flexible Plate, ALIZA ABRAHAM, ALEXANDRA TECHET, MIT — Digital image correlation (DIC) and particle image velocimetry (PIV) are combined to characterize the flow-structure interaction of a breaking wave impacting a flexible vertically mounted plate. DIC is used with the beam bending equation to determine the stresses on the plate and PIV is used to describe the flow of the wave. In this experiment, a simulated dam break in which water is rapidly released from a reservoir generates the wave, which impinges on a cantilevered stainless steel plate downstream. Pressure sensors mounted on the plate are used to gather further information about the forces acting on it. A series of waves of different heights and breaking locations are tested, controlled by the volume of water in the tank and the volume of water in the dam break reservoir. The deflection of the plate varies depending on the point of breaking and the height of the wave. These results shed light on the effect of breaking wave impacts on offshore structures and ship hulls.

2:36PM D13.00003 Onset of wind-wave generation on a viscous liquid, ANNA PAQUIER, MARC RABAUD, FREDERIC MOISY, Laboratory FAST, Orsay, France — In a new experimental set-up, we investigate the onset of wave generation over a viscous liquid. We access the spatio-temporal structure of the surface deformations using Free Surface Synthetic Schlieren. Alternating wind speed, surface elevation, and monochromatic waves organizes the complex three-dimensional structures, with propagating waves spatially in the downstream direction. This spatial growth is found to be exponential with the fetch (distance along the tank) at small fetch. The spatial growth rate increases linearly with wind speed, from which the onset for wind generation can be determined accurately. At higher wind velocity or fetch, nonlinear effects are observed, resulting in an increase of the wavelength and phase velocity, and to more disordered wave patterns.

2:49PM D13.00004 Experimental demonstration of epsilon-near-zero water waves focusing, TOMASZ BOBINSKI, ANTONIN EDDI, PHILIPPE PETITJEANS, PMMH-ESPCI, AGNES MAUREL, Institut Langevin, VINCENT PAGNEUX, Laboratoire d’Acoustique de l’Université du Maine — We demonstrate experimentally the epsilon-near-zero (ENZ) analogue for water waves in the nonlinear regime. In the context of electromagnetic waves, ENZ media are known to realize super lensing effect, because they are associated to very large wavelength. A lens made of such material with, say, circular edge shape, produces focused waves at the center of the circle (focal point of the lens). In the context of water waves, we demonstrate the analog of these media by tuning the bathymetry of the bottom sea owing the analogy between electromagnetic waves and water waves in the shallow water regime. Experimentally, we obtain uniform phase of the water wave at the edge of the semicircular lens, resulting in the expected lensing effect. By using time space resolved measurement of the two-dimensional field of surface elevation, we are able to separate the linear components of the wave and the harmonics generated by nonlinearities. Interestingly, we observe a cascade of highly focused harmonics. These harmonic components are analyzed in term of free-waves and bound-waves.

3:02PM D13.00005 A seabed-mounted diode for unidirectional water-wave propagation, LOUIS-ALEXANDRE COUSTON, MOHAMMAD-REZA ALAM, Department of Mechanical Engineering, University of California, Berkeley — The effect of a series of seabed patches of small-amplitude bars with increasing obliquity on a monochromatic oceanic wavetrain is shown to be analogous to the effect of a diode on the current in an electronic circuit. The incoming water wavetrain is deflected at a 90-degree angle in one direction, while barely changing its route in the other. In the reverse direction, i.e., in the direction where wave propagation is blocked, the incoming wavetrain (with bearing angle bi=0 degrees) satisfies a Bragg resonance condition over each one of the seabed patches, ceding its energy to a series of transmitted waves, with increasing bearing angle (i.e., btj -> bt1 > bi with btj the angle of the j-th transmitted wave). The resonances continue over the patches until btj=90 degrees. In the forward direction, i.e., in the direction where wave propagation is allowed, the incident wave satisfies a Bragg condition only with the last patch such that its deflection remains small. The minimum patch lengths leading to full deflection are obtained within potential flow theory using multiple-scale analysis, and in the analytical results are validated and extended with High-Order Spectral simulations. Some of the difficulties expected with laboratory experiments are given.

3:15PM D13.00006 ABSTRACT WITHDRAWN

3:28PM D13.00007 Reconstruction of arbitrary surface wave fields by refraction global method in a wave tank, HEYNERT GARCIA, ANDREI LUDU, Embry-Riddle Aeronautical University — We use a new photographic procedure and design to construct reliable system for measurement and analysis of various surface water waves in a wave tank, including rogue and tsunami-like waves. The grid placed at the bottom of the tank (3 feet maximum depth) is deformed by the surface waves and recorded on one or two cameras placed above the water. The measurement of the height and slope of the surface waves is determined by inverse refraction calculations plus the calibration information at four grouped points from capacitive level gauges.

3:41PM D13.00008 Numerical simulation of the resonantly excited capillary-gravity waves, HIDESHI HANAZAKI, MOTOHORI HIRATA, SHINYA OKINO, Kyoto University — Capillary gravity waves excited by an obstacle are investigated by a direct numerical simulation. In the flow without capillary effects, it is well known that large-amplitude upstream advancing solitary waves are generated periodically under the resonant condition, i.e., when the phase velocity of the long surface waves and the mean flow velocity agrees. With capillary effects, solutions of the Euler equations show the generation of very short waves further upstream of the solitary waves and also in the depression region downstream of the obstacle. The overall characteristics of these waves agree with the solutions of the forced fifth-order KdV equation, while the weakly nonlinear theory generally overestimates the wavelength of the short waves.

3:54PM D13.00009 Variational modelling of nonlinear water waves, ANNA KALOGIROU, ONNO BOKHOVE, University of Leeds — Mathematical modelling of water waves is demonstrated by investigating variational methods. A potential flow water wave model is derived using variational techniques and extended to include explicit time-dependence, leading to non-autonomous dynamics. As a first example, we consider the problem of a soliton splash in a long wave channel with a contraction at its end, resulting after a sluice gate is removed at a finite time. The removal of the sluice gate is included in the variational principle through a time-dependent gravitational potential. A second example involving non-autonomous dynamics concerns the motion of a free surface in a vertical Hele-Shaw cell. Explicit time-dependence now enters the model through a linear damping term due to the effect of wall friction and a term representing the motion of an artificially driven wave pump. In both cases, the model is solved numerically using a Galerkin FEM and the numerical results are compared to wave structures observed in experiments. The water wave model is also adapted to accommodate nonlinear ship dynamics. The novelty is this case is the coupling between the water wave dynamics, the ship dynamics and wave line dynamics on the ship. For simplicity, we consider a simple ship structure consisting of V-shaped cross-sections.
4:07PM D13.00010 Instability of propagating axial symmetric waves generated by a vertically oscillating sphere. MENG SHEN, YUMING LIU, Mechanical Engineering Department, Massachusetts Institute of Technology — We study the instability of propagating axial symmetric waves in a basin that are generated by a vertically oscillating sphere. Laboratory experiments indicate that when the oscillation amplitude exceeds a threshold value, the axial symmetric propagating waves abruptly transfigure into non-axial symmetric waves. Fully-nonlinear time-domain numerical simulation of wave-body interaction is applied to describe the nonlinear temporal and spatial evolution dynamics of the propagating waves. Transition matrix method is employed to analyze the stability of the periodic wave-body interaction system leading to the instability of the wave-body system and investigate the critical condition for the occurrence of the instability. We quantify the growth rate and dominant modes of unstable disturbances and study their dependence on physical parameters including body motion frequency and amplitude, body geometry, surface tension and basin size. Moreover, the long-time evolution dynamics of the unstable wave-body system including wave patterns and responsive body forces are also investigated.

Sunday, November 22, 2015 2:10PM - 4:20PM – Session D14 Industrial Applications II 202 - Kumaran Kannaiyan, Texas AM University at Qatar

2:10PM D14.00001 Effect of Fuel Additives on Spray Performance of Alternative Jet Fuels1, KUMARAN KANNAIYAN, REZA SADR, Texas A&M University at Qatar — Role of alternative fuels on reducing the combustion pollutants is gaining momentum in both land and air transport. Recent studies have shown that addition of nanoscale metal particles as fuel additives to liquid fuels have a positive effect not only on their combustion performance but also in reducing the pollutant formation. However, most of those studies are still in the early stages of investigation with the addition of nanoparticles at low weight percentages. Such an addition can affect the hydrodynamic and thermo-physical properties of the fuel. In this study, the near nozzle spray performance of gas-to-liquid jet fuel with and without the addition of alumina nanoparticles are investigated at macro- and microscopic levels using optical diagnostic techniques. At macroscopic level, the addition of nanoparticles is seen to enhance the sheet breakup process when compared to that of the base fuel. Furthermore, the microscopic spray characteristics such as droplet size and velocity are also found to be affected. Although the addition of nanoscale metal particles at low weight percentages does not affect the bulk fluid properties, the atomization process is found to be affected in the near nozzle region.

1Funded by Qatar National Research Fund

2:33PM D14.00002 Design of a Laboratory-scale Marine Hydrokinetic device, UROS MARKOVIC, MARIA BENINATI, Bucknell University, MICHAEL KRANE, The Pennsylvania State University — This study focused on the design of a small-scale marine hydrokinetic turbine, centered on a precision brake to facilitate rotational speed control, torque and power measurements. The design of size and power capacity suitable for laboratory-scale experiments generally operates at vanishingly small efficiency, making accurate power measurements difficult. A small magnetic particle brake was attached to the shaft of a two-bladed model marine turbine (0.1 m rotor diameter). Preliminary testing of the device was performed to calibrate torque measurement by the magnetic brake. Further testing was conducted in the hydraulic flume facility (9.8 m long, 1.2 m wide and 0.4 m deep) at Bucknell University, to measure turbine torque and power to establish the range of rotational speed control.

2:36PM D14.00003 Resource Evaluation and Energy Production Estimate for a Tidal Energy Conversion Installation using Acoustic Flow Measurements1, IAN GAGNON, KEN BALDWIN, MARTIN WOSNIK, University of New Hampshire — The Living Bridge project plans to install a tidal turbine at Memorial Bridge in the Piscataqua River at Portsmouth, NH. A spatio-temporal tidal energy resource assessment was performed using long term bottom-deployed Acoustic Doppler Current Profilers ADCP. Two locations were evaluated: at the planned deployment location and mid-channel. The goal was to determine the amount of available kinetic energy that can be converted into usable electrical energy on the bridge. Changes in available kinetic energy with ebb/flood and spring/neap tidal cycles and electrical energy demand were analyzed. A system model is used to calculate the net energy savings using various tidal generator and battery bank configurations. Differences in the tidal characteristics between the two measurement locations are highlighted. Different resource evaluation methodologies were also analyzed, e.g., using a representative ADCP bin vs. a more refined, turbine-geometry-specific methodology, and using static bin height vs. bin height that move w.r.t. the free surface throughout the tidal cycle (representative of a bottom-fixed or floating turbine deployment, respectively). ADCP operating frequencies and bin sizes affect the standard deviation of measurements, and measurement uncertainties are evaluated.

1Supported by NSF-IIP grant 1430260

2:49PM D14.00004 Numerical Simulation of Tethered Underwater Kites for Power Generation, AMIRMADHI GHASEMI, DAVID OLINGER, Worcester Polytech Inst., GRETRAR TRYGGVASON, University of Notre Dame — An emerging renewable energy technology, tethered underwater kites (TUSK), which is used to extract hydrokinetic energy from ocean and tidal waves, is considered. TUSK systems, a variant of a rigid-winged “kite,” or glider, floating on an ocean current which is connected to tethers to a floating buoy on the ocean surface. The TUSK kite is a current speed enhancement device since the kite can move in high-speed, cross-current motion at 4-6 times the current velocity, thus producing more power than conventional marine turbines. A computational simulation is developed to simulate the dynamic motion of the kite in an underwater kite and extendable tether. A two-step projection method within a finite volume formulation, along with an Open MP acceleration method is employed to solve the equations. An immersed boundary method is incorporated to model the fluid-structure interaction of the rigid kite (with NACA 0012 airfoil shape in 2D and NACA 0021 airfoil shape in 3D simulations) and the fluid flow. PID control methods are used to adjust the kite angle of attack during power (kite reel-out) and retraction (reel-in) phases. Two baseline simulations (for kite motions in two and three dimensions) are studied, and system power output, flow field vorticity, tether tension, and hydrodynamic coefficients (lift and drag) for the kite are determined. The simulated power output shows good agreement with established theoretical results for a kite moving in two-dimensions.

3:02PM D14.00005 Why do Cross-Flow Turbines Stall? , ROBERT CAVAGNARO, BENJAMIN STROM, BRIAN POLAYGE, University of Washington — Hydrokinetic turbines are prone to instability and stall near their peak operating points under torque control. Understanding the physics of turbine stall may help to mitigate this undesirable occurrence and improve the robustness of torque controllers. A laboratory-scale two-bladed cross-flow turbine operating at a chord-based Reynolds number $\approx 3 \times 10^5$ is shown to stall at a critical tip-speed ratio. Experiments are conducting bringing the turbine to this critical speed in a recirculating current flume by increasing resistive torque and allowing the rotor to rapidly decelerate while monitoring inflow velocity, torque, and drag. The turbine stalls probabilistically with a distribution generated from hundreds of such events. A machine learning algorithm identifies stall events and indicates the effectiveness of available measurements or combinations of measurements as predictors. Bubble flow visualization and PIV are utilized to observe fluid conditions during stall events including the formation, separation, and advection of leading-edge vortices involved in the stall process.
3:15PM D14.00006 Experimental Evaluation of a Method for Turbocharging Four-Stroke, Single Cylinder, Internal Combustion Engines, Michael Buchman, MIT graduate student, AMOS Winter V., MIT Professor — Turbocharging an engine increases specific power, improves fuel economy, reduces emissions, and lowers cost compared to a naturally aspirated engine of the same power output. These advantages make turbocharging commonplace for multi-cylinder engines. Single cylinder engines are not commonly turbocharged due to the phase lag between the exhaust stroke, which powers the turbocharger, and the intake stroke, when air is pumped into the engine. Our proposed method of turbocharging single cylinder engines is to add an “air capacitor” to the intake manifold, an additional volume that acts as a buffer to store compressed air between the exhaust and intake strokes, and smooth out the pressure pulses from the turbocharger. This talk presents experimental results from a single cylinder, turbocharged diesel engine fit with various sized air capacitors. Power output from the engine was measured using a dynamometer made from a generator, with the electrical power dissipated with resistive heating elements. We found that intake air density increases with capacitor size as theoretically predicted, ranging from 40 to 60 percent depending on heat transfer. Our experiment was able to produce 29 percent more power compared to using natural aspiration. These results validated that an air capacitor and turbocharger may be a simple, cost effective means of increasing the power density of single cylinder engines.

3:28PM D14.00007 Numerical study of crude oil fouling in a Taylor-Couette-type reactor, Misha Crastes, Lydia Lagkaditi, Jonathan Ball, Junfeng Yang, Francesco Coletti, Sandro Macchetto, Omar Matar, Imperial College London — We consider the non-isothermal flow of crude-oil mixtures in a Taylor-Couette-type reactor; this flow is accompanied by the deposition of soft-solid wall-layers, commonly referred to as “fouling”, driven by chemical reactions and phase separation. Three-dimensional CFD simulations are carried out to resolve the flow and temperature fields, as well as the volume fraction of the foulant phase. The simulations also account for the effect of evolving deposit rheology. The CFD predictions are validated against published results for isothermal flow, in the absence of fouling, in terms of the characteristics of the vortical structures that accompany the flow. In the presence of fouling, we examine the spatial distribution of the wall stresses as a function of the Reynolds and Taylor numbers, and demonstrate that wall regions exposed to higher (lower) shear stresses tend to form thinner (thicker) fouling layers. The simulation results capture the trends observed experimentally.

1Skolkovo Foundation through the UNIHEAT Project

3:41PM D14.00008 A new framework to increase the efficiency of large-scale solar power plants, SHAHROUZ ALIMOHAMMADI, JAN P. KLEISSL, UC San Diego — A new framework to estimate the spatio-temporal behavior of solar power is introduced, which predicts the statistical behavior of power output at utility scale Photo-Voltaic (PV) power plants. The framework is based on spatio-temporal Gaussian Processes Regression (Kriging) models, which incorporates satellite data with the UCSD version of the Weather and Research Forecasting model. This framework is designed to improve the efficiency of the large-scale solar power plants. The results are also validated from measurements of the local pyranometer sensors, and some improvements in different scenarios are observed.

3:54PM D14.00009 ABSTRACT WITHDRAWN —

4:07PM D14.00010 Numerical Investigation of the effect of adiabatic section location on thermal performance of a heat pipe network with the application in thermal energy storage systems, MAHBOOBE MAHDAVI, SAEED TIARI, SONGGANG QIU, Department of Mechanical Engineering, Temple University, 1947 N. 12th Street, Philadelphia, PA 19122, USA — Latent heat thermal energy storage systems benefits from high energy density and isothermal storing process. However, the low thermal conductivity of the phase change material leads to prolong the melting or solidification time. Using a passive device such as heat pipes is required to enhance the heat transfer and to improve the efficiency of the system. In the present work, the performance of a heat pipe network specifically designed for a thermal energy storage system is studied numerically. The network includes a primary heat pipe, which transfers heat received from solar receiver to the heat engine. The excess heat is simultaneously delivered to charge the phase change material via secondary heat pipes. The primary heat pipe composed of a disk shape evaporator, an adiabatic section and a disk shape condenser. The adiabatic section can be either located at the center or positioned outward to the surrounding of the condenser. The effect of adiabatic section position on thermal performance of the system is investigated. It was concluded that displacing the adiabatic section outwardly dramatically increases the average temperatures of the condensers and reduces the thermal resistance of heat pipes.

Sunday, November 22, 2015 2:10PM - 4:20PM — Session D15 General Fluid Dynamics: Viscous Flows 203 - Longhua Zhao, Case Western Reserve University

2:10PM D15.00001 Volume effect for particles transported in highly viscous fluids, LONGHUA ZHAO, Case Western Reserve University — For various shapes and sizes, particles are treated differently in the fluid system. The level complexity of models needs to be determined to capture the sufficient physical phenomena and such a decision is based on the assessment of the volume effect. This study is to measure the effect of a non-zero volume spherical particle transported in highly viscous flows. Utilizing the regularized Stokeslet method, particles are studied in three different approaches: a passive fluid tracer at the center of the particle, a sphere with Faxen’s correction, and a detailed non-zero volume particle. We compare the discrepancies in velocity field, Lagrangian trajectory and force exerted on the particle with three approaches and provide qualitative information for future studies.

2:23PM D15.00002 Slow viscous flow of two particles in a cylindrical tube, XIN YAO, TECK NENG WONG, MARCOS, Nanyang Technological University — The slow viscous flow around two particles in a cylindrical tube is obtained theoretically. We employ the Lamb’s general solution based on spherical harmonics and cylindrical harmonics to solve the flow field around the particles and the flow within the tube, respectively. We compute the drag and torque coefficients of the particles which are dependent on the distance among the cylinder wall and the two particles. The hydrodynamic forces are also a function of particle velocities and background velocity. Our results are in agreement with the existing theory of a single particle traveling in the tube when the distance between the two particles increases. We found that particle-particle interactions can be neglected when the separation distance is three times larger than the sum of particles radii. Furthermore, such analysis can give us insights to understand the mechanisms of collision and aggregation of particles.
Deformation of an Elastic beam due to Viscous Flow in an Embedded Channel Network, YOAV MATIA, AMIR GAT, Technion - Israel Institute of Technology — Elastic deformation due to embedded fluidic networks is currently studied in the context of soft-actuators and soft-robotic applications. In this work, we analyze the time dependent interaction between elastic deformation of a slender beam and viscous flow within a long serpentine channel, embedded in the elastic structure. The channel is positioned asymmetrically with regard to the midplane of the elastic beam, and thus pressure within the channel creates a local moment deforming the beam. We focus on creeping flows and small deformations of the elastic beam and obtain, in leading order, a convection–diffusion equation governing the pressure-field within the serpentine channel. The beam time-dependent deformation is then obtained as a function of the pressure-field and the geometry of the embedded network. This relation enables the design of complex time-dependent deformation patterns of beams with embedded channel networks. Our theoretical results were illustrated and verified by numerical computations.

Transient Dynamics of Elastic Hele-Shaw Cell due to External Forces with Application to Impact Mitigation, ARIE TULCHINSKY, AMIR GAT, Technion - Israel Institute of Technology, Faculty of Mechanical Engineering — We study the transient dynamics of a viscous liquid contained in a narrow gap between a rigid plate and an elastic plate. The elastic plate is under the influence of an externally applied time-varying force acting perpendicular to its surface. We model the flow in the narrow gap via the lubrication approximation, and the plate by the Kirchhoff-Love plate theory. The viscous-elastic interaction yields a governing 6th-order linear partial differential equation. We obtain a semi-similarity solution for the case of an external point force acting on the elastic plate. The pressure and deformation field during and after the application of the external force are derived and presented by closed form expressions. We examine a uniform external pressure acting on the elastic plate over a finite region and during a finite time period similar to the viscous-elastic interaction time-scale. The interaction between elasticity and viscosity is shown to reduce by order of magnitude the pressure within the Hele-Shaw cell compared with the externally applied pressure, thus suggesting such configurations may be used for impact mitigation.

Non-linear dynamics of annular creeping flow enclosed by an elastic membrane, SHAI ELBAZ, AMIR GAT, Technion Israel Institute of Technology — This study deals with the fluid-structure-interaction problem of longitudinal annular flow on a varying cross-section centre-body enclosed by an elastic membrane. The gap between the centre-body and membrane wall may be initially filled with a thin fluid layer or devoid of it. We employ elastic shell theory and the lubrication approximation and obtain a forced nonlinear diffusion equation governing the problem. In the case of an advancing liquid front in an initially empty annular gap, the forced convection equation degenerates into a forced linear Schrödinger equation for which several closed-form solutions can be obtained. Based on self-similarity we define propagation laws for the fluid-elastic interaction which in turn provide the basis for numerical investigation of compound solutions such as pulse trains and other waveforms. The presented interaction between viscosity and elasticity may be applied to fields such as soft-robotics and micro-scale or larger swimmers by allowing for the time-dependent control of a compliant boundary.

A Note on the breathing mode of an elastic sphere in Newtonian and complex fluids, VAHE GALSTYAN, Columbia University, ON SHUN PAK, Santa Clara University, HOWARD STONE, Princeton University — Experiments on the acoustic vibrations of elastic nanostructures in fluid media have been used to study the mechanical properties of materials. The medium surrounding the nanostructure is typically modeled as a Newtonian fluid. A recent experiment however suggested that high-frequency longitudinal vibration of bipyramidal nanoparticles could trigger a viscoelastic response in water-glycerol mixtures [Pelton et al., “Viscoelastic flows in simple liquids generated by vibrating nanostructures,” Phys. Rev. Lett. 111, 244502 (2013)]. Motivated by these experimental studies, we first revisit a classical continuum mechanics problem of the purely radial vibration of an elastic sphere in a compressible viscous fluid and then extend our analysis to a viscoelastic medium using the Maxwell fluid model. Although in the case of longitudinal vibration of bipyramidal nanoparticles, the effects of fluid compressibility were shown to be negligible, we demonstrate that it plays a significant role in the breathing mode of an elastic sphere. On the other hand, despite the different vibration modes, the breathing mode of a sphere triggers a viscoelastic response in water-glycerol mixtures similar to that triggered by the longitudinal vibration of bipyramidal nanoparticles.

Dragging cylinders in slow viscous flows, ELENA LUCA, DARREN CROWDY, Imperial College London — The so-called “dragging problem” in slow viscous fluids is an important basic flow with many applications. In two dimensions, it is easy to solve for a cylinder in free space. The presence of walls changes this; the solutions exist, but are not easy to find without purely numerical methods. This talk describes new “transform methods” that produce convenient, semi-analytical solutions to dragging problems for cylinders in various geometries. We apply the techniques to low-Reynolds-number swimming where dragging problem solutions can be combined with the reciprocal theorem to compute swimmer dynamics in confined domains.

Slender Ribbon Theory, LYNDON KOENS, ERIC LAUGA, Univ of Cambridge — Ribbons are long narrow strips possessing three distinct material length scales (thickness, width, and length) which allow them to produce unique shapes unobtainable by wires or filaments. Significant effort has gone into determining the structural shapes of ribbons but less is known about their behavior in viscous fluids. Here we determine asymptotically the leading-order hydrodynamic behavior of a slender ribbon in Stokes flows. The derivation, reminiscent of slender-body theory for filaments, assumes that the length of the ribbon is much larger than its width, which itself is much larger than its thickness. The final result is an integral equation for the force density on a mathematical surface located inside the ribbon. Our derivation agrees very well with the known hydrodynamics of long flat ellipsoids, and successfully captures the swimming behavior of artificial microscopic swimmers recently explored experimentally. Our asymptotic results provide the fundamental framework necessary to predict the behavior of slender ribbons at low Reynolds numbers in a variety of biological and engineering problems.

Temperature Variations in Lubricating Films Induced by Viscous Dissipation1, FARSHAD MOZAFFARI, RALPH METCALFE, Univ of Houston — We have studied temperature distributions of lubricating films. The study has applications in tribology where temperature-reduced viscosity decreases load carrying capacity of bearings, or degrades elastomeric seals. The viscosity-temperature dependency is modeled according to ASTM D341-09. We have modeled the film temperature distribution by our finite element program. The program is made up of three modules: the first one solves the general form of Reynolds equation for the film pressure and velocity gradients. The other two solve the energy equation for the film and its solid boundary temperature distributions. The modules are numerically coupled and iteratively converged to the solutions. We have shown that the temperature distribution in the film is strongly coupled with the thermal response at the boundary. In addition, only thermal diffusion and not film thickness is dominant. Moreover, thermal diffusion in the lateral directions, as well as all the convection terms, are negligible. The approximation reduces the energy equation to an ordinary differential equation, which significantly simplifies the modeling of temperature- viscosity effects in thin films.

1Supported by Kalsi Engineering, Inc.
Tip Vortex Characteristics. VERA KLIMCHENKO, University of Maryland — The current study investigates the effects the serrated tip rotor had strong tip vortices with the same core radius as the baseline case. Flow visualization was used to isolate the time-varying components of the flow. The vorticity of the phase-averaged time-varying field was calculated, and the tip vortices were identified using a vortex identification method.

Sunday, November 22, 2015 2:10PM - 4:20PM – Session D16 Aerodynamics: Control 204 - Samik Bhattacharya, Queen’s University

2:10PM D16.00001 The effect of acceleration on the growth and shedding of laminar separation bubbles. SAMIK BHATTACHARYA, DAVID RIVAL, Queen's University — It has been observed that when a laminar boundary layer separates, the shear layer undergoes transition to turbulence and subsequently reattaches to form a laminar separation bubble (LSB). In this work, a SD7003 airfoil, held at an angle of attack of 8 degree, is towed with different acceleration profiles starting from rest. The separation region is then analyzed with time-resolved, planar PIV at short convective times during the initial acceleration phase. The aim of this work is to characterize the variation in size and shedding frequency of the laminar separation bubble with increasing acceleration. We show that the formation and shedding process in the LSB depends on the rate of vorticity-containing mass transported by the separated shear layer. Consequent to any changes in the structure of the shear layer affect the formation of the LSB downstream. Finally, attempts are also made to characterize the shedding frequency of the bubble with increasing acceleration. Here the unsteadiness of the LSB is found to be closely linked to the degree of boundary-layer acceleration on the airfoil surface.

2:23PM D16.00002 A new plasma-driven pulsed jet actuator for flow control. JEAN-PAUL BONNET, retired, CNRS Institut Pprime Poitiers, GWENAEL ACHEEL, ANTON LEBEDEV, NICOLAS BENARD, ERIC MOREAU, CNRS Institut Pprime Poitiers, ELECTRO-FLUIDO GROUP TEAM — Active flow control requires actuators with enough authority and high frequency response. Synthetic jets can have high frequency response but are rather limited in terms of authority providing the exit velocity is limited. Pressurized (flowing) jets have a very high potential in terms of authority particularly for high velocity flow control purposes. However, for most purposes, high frequency modulation (of order of several kHz) is required in order to excite most unstable modes and to operate in closed mode. Rapid mechanical valves are limited in terms of frequency (up to typically a few hundred of Hz). We develop a new generation of plasma-driven pulsation of flowing jet. The principle is to increase the temperature at the sonic throat through a plasma discharge located at the throat. The flow rate being proportional to the square root of the temperature for a perfect gas, for the same settling chamber pressure, the actuator flow rate can be varied. The frequency is then no limited by any mechanical constraint. A demonstrator has been tested with a 1mm sonic throat. The electric discharge is created by a 10 kV voltage applied between the anode and the throat acting as the cathode. Within these conditions, a 30% modulation of the flow rate can be obtained.

2:36PM D16.00003 Large-Eddy Simulations of Plasma Flow Control on a GOE735 Wind Turbine Airfoil . ALEXANDER CZULAK, Robert Bosch LLC, JENNIFER FRANCK, Brown University — Active flow control using plasma actuation was studied for the GOE735 airfoil and compared to non-actuated baseline cases using numerical simulations. This investigation considers two-dimensional simulations at a Reynolds number of 1,000 using direct numerical simulation (DNS) as well as three-dimensional simulations at a Reynolds number of 50,000 and 100,000 using large-eddy simulation (LES). Plasma actuation is applied in terms of a source term within the boundary layer close to the airfoil surface. Angles of attack of 0°, 5°, and 15° were considered, and control is shown to be effective at increasing the lift coefficient, decreasing the drag coefficient and reducing the root mean squared deviation of both lift and drag. An analysis of the flow physics reveals that the actuated cases delay the point of separation, reduce the wake width and diminish the size and strength of the shed vortices. For this particular airfoil, there are significant differences in Reynolds number in terms of the baseline flow, control effectiveness and performance factors such as lift and drag.

2:49PM D16.00004 Effects of actuation waveform shape on the performance of pitching and heaving panels1. DANIEL FLORYAN, TYLER VAN BUREN, CLARENCE W. ROWLEY, Princeton University, ALEXANDER J. SMITS, Princeton University and Monash University — Experiments on panels pitching and heaving in a water channel are reported. The panels are rigid, of rectangular planform shape, and the flow is nominally two-dimensional. Through the use of Jacobi elliptic functions, we are able to actuate the panels in non-sinusoidal and asymmetric motions; we investigate how such motions affect the propulsive performance of the panels. Direct force measurements are taken using a six component force/torque sensor, and certain cases are supplemented with two-dimensional particle image velocimetry (PIV) taken at the mid-span of the panel; efficiency measurements are also reported.

1Supported under ONR MURI Grant N00014-14-1-0533.

3:02PM D16.00005 Experimental Study of the Effects of Blade Treatments on the Tip Vortex Characteristics. VERA KLIMCHENKO, University of Maryland — The current study investigates the characteristics of a wind turbine blade tip vortex. Three blade tip shapes including a blunted edge, leading edge comb, and a winglet were designed and tested in a low speed wind tunnel. The rotor with a blunted edge was considered to be a baseline case corresponding to an untreated blade tip. The leading-edge comb rotor was designed with leading edge tubercles extending from the tip of the blade inward, 6 percent of rotor diameter. The winglet located at the tip of the winglet rotor had a cant angle of 45 degrees. The wind turbine operated at a tip speed ratio of 5 and a tip Reynolds number of 14,000. The tip treatments were intended to weaken the tip vortices by encouraging dissipation (leading edge comb) or promoting the formation of weaker vortices (winglet). Time-resolved and phase-averaged PIV was used to measure the velocity field behind the rotor. The time-averaged velocity field was subtracted from the phase-averaged velocity field to isolate the time-varying components of the flow. The vorticity of the phase-averaged time-varying field was calculated, and the tip vortices were identified using a vortex identification method. Vortex characteristics such as core radius and vortex strength were calculated and compared for the three rotors. The analysis of the vorticity showed that the winglet rotor had weaker tip vortices with a larger core radius, while the serrated tip rotor had stronger tip vortices with the same core radius as the baseline case.
3:15PM D16.00006 Adjoint-based optimal control of an airfoil in gusting flows
JEESOOKO CHOI, TIM COLONIUS, Caltech, CALIFORNIA INSTITUTE OF TECHNOLOGY TEAM — In this study, we apply optimal control to an airfoil in gusting flow to investigate the possibility of extracting energy. The gradients of an objective function are obtained via the adjoint method and used to minimize the cost. The immersed boundary projection is used to solve the forward problem, and the Siadini adjoint approach is used to derive the gradients of the cost function. The solution is validated using 3D conjugate flow around a cylinder and a full-scale wind tunnel experiment with members of the Korean national team and confirms the results obtained from the experiment on the model.

3:28PM D16.00007 Aerodynamics of ski jumping flight and its control: I. Experiments
JEESOOKO CHOI, TIM COLONIUS, Caltech, CALIFORNIA INSTITUTE OF TECHNOLOGY TEAM — In this study, we apply optimal control to an airfoil in gusting flows to investigate the possibility of extracting energy. The gradients of an objective function are obtained via the adjoint method and used to minimize the cost. The immersed boundary projection method is used to solve the forward problem, and the Siadini adjoint approach is used to derive the gradients of the cost function. The solution is validated using 3D conjugate flow around a cylinder and a full-scale wind tunnel experiment with members of the Korean national team and confirm the results obtained from the experiment on the model.

3:41PM D16.00008 Aerodynamics of ski jumping flight and its control: II. Simulations
JEESOOKO CHOI, TIM COLONIUS, Caltech, CALIFORNIA INSTITUTE OF TECHNOLOGY TEAM — In this study, we apply optimal control to an airfoil in gusting flows to investigate the possibility of extracting energy. The gradients of an objective function are obtained via the adjoint method and used to minimize the cost. The immersed boundary projection method is used to solve the forward problem, and the Siadini adjoint approach is used to derive the gradients of the cost function. The solution is validated using 3D conjugate flow around a cylinder and a full-scale wind tunnel experiment with members of the Korean national team and confirm the results obtained from the experiment on the model.

3:54PM D16.00009 Parametric Study of Synthetic-Jet-Based Flow Control on a Vertical Tail Model
JEESOOKO CHOI, TIM COLONIUS, Caltech, CALIFORNIA INSTITUTE OF TECHNOLOGY TEAM — In this study, we apply optimal control to an airfoil in gusting flows to investigate the possibility of extracting energy. The gradients of an objective function are obtained via the adjoint method and used to minimize the cost. The immersed boundary projection method is used to solve the forward problem, and the Siadini adjoint approach is used to derive the gradients of the cost function. The solution is validated using 3D conjugate flow around a cylinder and a full-scale wind tunnel experiment with members of the Korean national team and confirm the results obtained from the experiment on the model.

4:07PM D16.00010 Numerical Investigation of Bending-Body Projectile Aerodynamics for Maneuver Control
JEESOOKO CHOI, TIM COLONIUS, Caltech, CALIFORNIA INSTITUTE OF TECHNOLOGY TEAM — In this study, we apply optimal control to an airfoil in gusting flows to investigate the possibility of extracting energy. The gradients of an objective function are obtained via the adjoint method and used to minimize the cost. The immersed boundary projection method is used to solve the forward problem, and the Siadini adjoint approach is used to derive the gradients of the cost function. The solution is validated using 3D conjugate flow around a cylinder and a full-scale wind tunnel experiment with members of the Korean national team and confirm the results obtained from the experiment on the model.

Session D17 Flow Control: Drag Reduction I

Sunday, November 22, 2015 2:10PM - 4:20PM

2:10PM D17.00001 Experimental study of flow in a channel with a periodically heated wall
JEESOOKO CHOI, TIM COLONIUS, Caltech, CALIFORNIA INSTITUTE OF TECHNOLOGY TEAM — In this study, we apply optimal control to an airfoil in gusting flows to investigate the possibility of extracting energy. The gradients of an objective function are obtained via the adjoint method and used to minimize the cost. The immersed boundary projection method is used to solve the forward problem, and the Siadini adjoint approach is used to derive the gradients of the cost function. The solution is validated using 3D conjugate flow around a cylinder and a full-scale wind tunnel experiment with members of the Korean national team and confirm the results obtained from the experiment on the model.
2:23PM D17.00002 On the Analysis of Flows in Vibrating Channels
SAHAB ZANDI, University of Western Ontario, ALIREZA MOHAMMADI, Princeton University, JERZY MACIEJ FLORYAN, University of Western Ontario — Pressure losses in channels with vibrating walls have been analyzed. Surface vibrations were assumed to have the form of travelling waves. The waves can have arbitrary profiles. The spectrally accurate immersed boundary conditions (IBC) method based on the Fourier expansions in the flow direction and the Chebyshev expansions in the transverse direction has been developed. The results show dependence of the pressure losses on the phase speed of the waves, with the waves propagating in the downstream direction reducing the pressure gradient required to maintain a fixed flow rate. A drag increase is observed when the waves propagate with a phase speed similar to the flow velocity. Analytical solution demonstrates that the drag changes result from the nonlinear interactions and vary proportionally to $A^2$ for small enough $A$, where $A$ stands for the wave amplitude.

1This work has been carried out with support from the Natural Sciences and Engineering Research Council (NSERC) of Canada.

2:36PM D17.00003 Turbulent boundary layer control at moderate Reynolds numbers by means of uniform blowing/suction
YUKINORI KAMETANI, The University of Tokyo, KOJI FUKAGATA, Keio University, RAMIS ORLU, PHILIPP SCHLATTER, Kungliga Tekniska Hogskolan KTH — The effect of uniform blowing or suction from the wall on a spatially developing turbulent boundary layer has been studied in order to use them ultimately for flow control on the surface of high-speed vehicles. In the present study, a series of large eddy simulations is performed to investigate the effects of uniform blowing/suction on the surface friction drag as well as the scale of turbulent structures at moderate Reynolds numbers up to $Re_\theta = 2500$, based on free-stream velocity, $U_\infty$, and momentum thickness, $\theta$. The amplitude of blowing or suction is fixed to 0.1% of $U_\infty$ with different streamwise ranges of the control region. While the Reynolds shear and normal stresses and their spectral energy distributions are increased by blowing and decreased by suction, in particular, in the outer region, the FIK identity reveals that drag reduction (DR) or enhancement (DE) are mainly linked to changes in the spatial development of the mean wall-normal convection term rather than the contribution from the Reynolds shear stress. Despite the weak amplitude of the control, over 10% of DR and DE are achieved by blowing and suction, respectively. In case of blowing, the mean DR rate increases as the blowing region extends because the local reduction rate grows in the streamwise direction.

1Grant-in-Aid for Scientific Research (C) (No. 25420129), Grant-in-Aid for JSPS Fellow (No. 24-3450), the Knut an Alice Wallenberg Foundation

2:49PM D17.00004 Reactive Control of Boundary Layer Streaks Induced by Freestream Turbulence Using Plasma Actuators
KEVIN GOUDER, Imperial College, AHMED NAGUIB, Michigan State University, PHILIPPE LAVOIE, University of Toronto, JONATHAN MORRISON, Imperial College — Over the past few years we have carried out a systematic series of investigations aimed at evaluating the capability of a plasma-actuator-based feedforward-feedback control system to weaken streaks induced “synthetically” in a Blasius boundary layer via dynamic roughness elements. This work has been motivated by the delay of bypass boundary layer transition in which the streaks form stochastically beneath a freestream with turbulence of intensity of more than approximately 1%. In the present work, we carry forward the knowhow from our previous research in a first attempt to control such naturally occurring streaks. The experimental setup consists of a turbulence-generating grid upstream of a flat plate with a sharp leading edge. At the freestream velocity of the experiment, turbulent spot formation is observed to start at a streamwise location of $x \approx 350$ mm from the leading edge. The control system is implemented within a streamwise domain stretching from $x = 150$ mm to 300mm, where the streaks exhibit linear growth. At the upstream and downstream end of the domain a feedforward and a feedback wall-shear-stress sensors are utilized. The output from the sensors is fed to appropriately designed controllers which drive two plasma actuators providing positive and negative wall-normal forcing to oppose naturally occurring high- and low-speed streaks respectively. The results provide an assessment of the viability of the control approach to weaken the boundary layer streaks and to delay transition.

3:02PM D17.00005 Drag reduction through wave-current interactions with a marine hydrofoil
IGNAZIO MARIA VIOLA, SUSAN TULLY, DAVID INGRAM, Univ of Edinburgh — A hydrofoil exposed to oscillating flow experiences a reduction in drag due to the Knoller-Betz effect. This is experimentally identifiable by an increasing inverted von Karmann wake and a corresponding thrust force on the foil. The rate of drag reduction, dependent on plunge amplitude and frequency, reduces with unsteady flow phenomena at higher reduced frequencies. For experimental ease, investigations of this effect have relied on actively plunging/pitching a foil within a steady current. However, one potential application is to drag reduction in high-speed ships adopting submerged foils. In this case the foil is travelling through wave-current induced oscillatory flow, resulting in an additional dynamic variation of hydrostatic pressure across the foil; a phenomena not fully addressed in previous experiments. Here we investigate the effects of this pressure gradient on drag reduction for a stationary foil in combined waves and current, through a combination of force measurements and particle image velocimetry.

3:15PM D17.00006 Rotational Stabilization of Cylinder Wakes Using Linear Feedback Control
JEFF BORRGAARD, SERKAN GUERCIN, LIZETTE ZIETSMAN, Virginia Tech — We demonstrate the feasibility of linear feedback control to stabilize vortex shedding behind twin cylinders using the cylinder rotations. Our approach is to linearize the flow about a desired steady-state flow, use a simulation-based linear model to create a low-dimensional model of the input-output system with input-independent error bounds, then use this reduced model to design the feedback control law. We then consider the practical issue of limited state measurements by building a nonlinear compensator that is computed from the same linear reduced-order model constructed through an extended Kalman filter with a proper orthogonal decomposition (POD) model. Closed-loop simulations of the Navier-Stokes equations coupled with controls generated through flow measurements demonstrate the effectiveness of this control strategy.

1Supported in part by the National Science Foundation

3:28PM D17.00007 Blunt-body drag reduction through base cavity shape optimization
MANUEL LORITE-DIEZ, JOSE IGNACIO JIMENEZ-GONZALEZ, CANDIDO GUTIERREZ-MONTES, CARLOS MARTINEZ-BAZAN, Univ de Jaen — We present a numerical study on the drag reduction of a turbulent incompressible flow around two different blunt bodies, of height $H$ and length $L$, at a Reynolds number $Re = \rho U_\infty L / \mu = 2000$, where $U_\infty$ is the turbulent incompressible free-stream velocity, $\rho$ is their density and $\mu$ their viscosity. The study is based on the optimization of the geometry of a cavity placed at the rear part of the body with the aim of increasing the base pressure. Thus, we have used an optimization algorithm, which implements the adjoint method, to compute the two-dimensional incompressible turbulent steady flow sensitivity field of axial forces on both bodies, and consequently modify the shape of the cavity to reduce the induced drag force. In addition, we have performed three dimensional numerical simulations using an IDDES model in order to analyze the drag reduction effect of the optimized cavities at higher Reynolds numbers. The results show average drag reductions of 17% and 25% for $Re = 2000$, as well as more regularized and less chaotic wake flows in both bodies.

Supports by the Spanish MINECO, Junta de Andalucía and EU Funds under projects DPI2014-59292-C3-3-P and P11-TEP7495
3:41PM D17.00008 Power loss minimizing blowing and suction profiles for drag reduction on a circular cylinder, PRITAM GIRI, RATNESH SHUKLA, Indian Institute of Science, Bangalore — Active and passive flow control strategies that facilitate drag reduction at low energetic costs are of considerable fundamental and practical relevance. Here, we investigate the efficacy of a zero net mass transpiration blowing and suction flow control strategy based on intake and expulsion of fluid from the boundary of a circular cylinder placed in a uniform cross flow of a viscous incompressible fluid. We find this control strategy to be most effective when the blowing and suction profile is such that the fluid intake and expulsion occur over upstream and downstream portions of the circular cylinder, respectively. With increasingly strong intake and expulsion, the vorticity production at the cylinder surface diminishes significantly and the unsteady vortex shedding is suppressed entirely. We find that for sufficiently strong blowing and suction strengths the net power consumption attains a minimum for a significantly reduced net drag force. At a Reynolds number of 1000 the drag is reduced by a factor of over 15 from its base value for a stationary cylinder with zero mass transpiration. We show that a self-propelling state with zero drag force is achieved for a configuration that corresponds to an irrotational flow with vanishing tangential but finite normal surface velocity.

3:54PM D17.00009 Feedback Control of a Square-Back Ahmed Body Flow for Form-Drag Reduction, OLGA EVSTAFYEVA, AIMEE MORGANS, None — Road transport accounts for roughly 22% of CO₂ emissions worldwide, and at highway speeds two thirds of usable energy is consumed overcoming aerodynamic drag. For square-back vehicles, aerodynamic drag is dominated by form drag, originating from pressure difference between the front and the back face (base) of the vehicle. This study explores feedback control to increase mean base pressure and thus reduce the form-drag of 3D Ahmed body flows at low (laminar) and medium (transitioning to turbulence) Reynolds numbers. Using Large Eddy Simulations as a test-bed, a linear control strategy to attenuate base-pressure force fluctuations is investigated. Body-mounted sensing and actuation is used: sensing of the base pressure force fluctuations, and actuation of a zero-mean slot jet just ahead of the base. The dynamic linearity of the response to actuation is tested and a feedback controller then designed using frequency domain harmonic forcing system identification data. Recent advances in understanding of the Ahmed body wake dynamics such as top-to-bottom and left-to-right bi-stable behaviour, are considered in the feedback control implementation.

4:07PM D17.00010 Simulation, Modeling and Feedback Control of the flow around a Square-Back Bluff Body, LAURENT DALLA LONGA, AIMEE MORGANS, Imperial College London, IMPERIAL COLLEGE LONDON - FLOW CONTROL TEAM — Because of capacity, aesthetic and comfort requirements, most road vehicles are not streamlined but blunt bluff bodies. The flow exhibits a large wake recirculation area leading to high pressure drag, which at highway speeds, represents the main source of energy loss. In this work, Large Eddy Simulations of the flow past a square-back bluff body with interacting shear layers are performed with the aim of reducing aerodynamic drag. A linear feedback control strategy is applied to increase the back face pressure and therefore obtain drag reduction. Synthetic jets located along the perimeter of the back face are used for actuation while body mounted sensors record the base pressure force fluctuations. The validation of the flow response to actuation is performed using this flow system identification strategy, which is argued to be both linearly and dynamically linear. Based on the identified frequency response, a feedback controller is designed in the frequency domain which aims to either attenuate or amplify base pressure fluctuations by shaping of the sensitivity transfer function. This is first done for a D-shaped body. Current work extends this strategy to a simplified lorry geometry on which experiments were carried out recently.

Sunday, November 22, 2015 2:10PM - 4:20PM Session D18 Vortex Dynamics: Flow Induced Vibrations and Interactions

2:10PM D18.00001 Three-dimensional flow visualization of a flexible cylinder wake subject to VIV, JASON M. DAHL, Ocean Engineering Department, University of Rhode Island, EMMA THOMAS, Physics Department, University of Massachusetts Amherst, ERSEGEN D. GEDIKLI, Ocean Engineering Department, University of Rhode Island — The vortex-induced vibration of a low aspect ratio, low mode number, flexible cylinder is investigated in a recirculating flow channel under uniform inflow conditions. The cylinder had an aspect ratio of 40 and mass ratio of 3.76. The motion of the cylinder is tracked visually, using two high-speed cameras and the intersection of a laser sheet with the cylinder surface, capturing the cross-sectional response of cylinder at various locations along the span. Concurrent with the motion capture system, Particle Image Velocimetry is used to capture the velocity field in the wake of the cylinder at the same locations. The periodic nature of vibrations along the span of the cylinder is used to phase average the motion and wake of the cylinder, allowing for a phase averaged 3-D reconstruction of the cylinder wake. The 3-D reconstruction consists of stereoscopic PIV planes with measurements obtained at an arc along the wake at equal intervals at various speeds showing the excitation of the first mode of the cylinder in the cross-flow direction and the transition to the excitation of the second mode of the cylinder in the in-line direction. This technique is shown to capture 3-D variation of vortex-shedding in the wake of the flexible cylinder.

2:23PM D18.00002 VIV of a Flexible Cylinder: Three-dimensional Response Reconstruction from Limited Localized Measurement Points, BANAFSHEH SEYED-AGHAZADEH, YAHYA MODARRES-SADEGHI, University of Massachusetts, Amherst — Vortex-induced vibration (VIV) of a low mass ratio flexible cylinder (m*<1), is studied experimentally. The flexible tension-dominated cylinder was held fixed at both ends and was immersed in the uniform incoming flow. Dynamic response of the system was studied in the reduced velocity range of U* = 2.9 – 14.5 and the Reynolds number range of Re = 313 – 1145. Continuous response measurements were conducted at a reduced frequency of 0.11 and using the cross theorem modified using Modal Assurance Criterion (MAC). This reconstruction technique made it possible to properly reconstruct a continuous response along the length of the cylinder, even when the measurement points were localized in a small region of the cylinder. Mono- and multi-frequency excitation responses as well as transition from low mode numbers to higher ones were studied. Also, flow forces acting on the cylinder were calculated and they showed a consistent relation between the regions where the cylinder was being excited by the flow (CLv>0) and the clockwise figure-eight trajectories of oscillations in which the phase difference between the two end points was in the range of φ<π/2.

2:36PM D18.00003 Regimes of flow induced vibration for tandem, tethered cylinders, GARY NAVE, MARK STREMLER, Virginia Tech — In the wake of a bluff body, there are a number of dynamic response regimes that exist for a trailing bluff body depending on spacing, structural restoring forces, and the mass-damping parameter m*C. For tandem cylinders with low values of m*C, two such regimes of motion are Gap Flow Switching and Wake Induced Vibration. In this study, we consider the dynamics of a single degree-of-freedom rigid cylinder in the wake of another in these regimes for a variety of center-to-center cylinder spacings (3-5 diameters) and Reynolds numbers (4,000-11,000). The system consists of a trailing cylinder constrained to a circular arc around a fixed leading cylinder, which, for small angle displacements, bears a close resemblance to the transversely oscillating cylinders found more commonly in existing literature. From experiments on this system, we compare and contrast the dynamic response within these two regimes. Our results show sustained oscillations in the absence of a structural restoring force in all cases, providing experimental support for the wake stiffness assumption, which is based on the mean lift toward the center line of flow.
Wake–induced vibrations in Tandem Cylinders

RAVI CHAITHANYA MYSYA, RAJEEV KUMAR JAIMAN, National University of Singapore — The upstream cylinder is fixed in the tandem cylinders arrangement. The downstream cylinder is placed at a distance of four diameters from the upstream cylinder in the free stream direction and is mounted on a spring. The dynamic response of the downstream cylinder is studied at Reynolds number of 10,000. The transverse displacement amplitude of the downstream cylinder is larger compared to that of single cylinder in the post-lock-in region. The transverse dynamic response of the downstream cylinder in the post-lock-in region is characterized by a dominant low frequency component compared to shed frequency, which is nearer to the structural natural frequency. The interaction of upstream wake with the downstream cylinder is carefully analyzed to understand the introduction of low frequency component in the transverse load along with the shed frequency. We found that the stagnation point moves in proportion to the velocity of the cylinder and is in-phase with the velocity. The low frequency component in the stagnation point movement on the downstream cylinder is sustained by the interaction of upstream wake. The frequencies in the movement of the stagnation point is reflected in the transverse load resulting in large deformation of the cylinder.

Work supported by National Science Foundation under Grant No. CBET #1033117.

Flow-induced oscillations of a freely vibrating circular cylinder in the vicinity of a stationary wall.

RAJEEV JAIMAN, DANIEL THAM, LI ZHONG, PARDHA GURUGUBELLI, National University of Singapore — We present a numerical study on vortex-induced vibration (VIV) of a freely vibrating two degree-of-freedom circular cylinder oscillation significantly affects the vortex shedding and consequently aerodynamic forces and damping. Experimental data show that aerodynamic damping is directly proportional to oscillation amplitude but this relationship is non-linear for the aerodynamic force. These experimental data have been used to revisit VIV scaling. Comparison between our experimental data and experimental data of past studies in Griffin plot under the new scaling have showed very good agreement.

Flow-induced oscillations of a prism with triangular cross-section placed in water.

DANIEL CARLSON, BANAFSHE SEYED-AGHAZADEH, YAHYA MODARRES-SADEGH, Univ of Mass - Amherst — Flow-induced oscillations of a prism with a triangular cross-section was studied experimentally. The cylinder had one-degree-of-freedom to oscillate in the crossflow direction. The response of the cylinder in terms of the amplitudes of oscillations as well as the flow forces were studied at varying angles of attack in the range of $\alpha=0^\circ$--$60^\circ$ and a reduced velocity range of $U^*=4$--$22$. Depending on the angle of attack and the reduced velocity, the cylinder experienced either VIV or galloping. For small angles of attack $\alpha<30^\circ$, the cylinder did not oscillate while for larger angles of $\alpha=30^\circ$ and $35^\circ$, the cylinder underwent VIV in a range of reduced velocities ($U^*=7$--$14.5$) and galloping at higher reduced velocities ($U^*=19.5$--$22$). The conducted dye flow visualization as well as the measured flow forces confirmed the existence of lock-in as well as galloping-type response. For larger angles of attack of $\alpha>35^\circ$, the amplitude of oscillations increased monotonically with increasing reduced velocity and the cylinder underwent galloping. Several different vortex shedding patterns were observed in the wake of the cylinder at different angles of attack and flow velocities. New, high-frequency shedding patterns with their corresponding high harmonic shedding frequencies in the flow force FFTs were observed in the regions where galloping occurred.

This work is partially supported by the NSF-sponsored IGERT: Offshore Wind Energy Engineering, Environmental Science, and Policy (Grant Number 1068804).
4:07PM D18.00010 Two-dimensional wakes of oscillating and tandem cylinders at low Reynolds number Y, WENCHAO YANG, MARK STREMLER, Virginia Polytechnic Institute and State University — Transverse flow past an oscillating bluff body or multiple stationary bodies can produce wakes with complicated spatio-temporal structure. Previous work by others has characterized the wake structure as a function of system parameters. These are typically 2D characterizations, despite the fact that instabilities often cause such wakes to become fully 3D. We use a flowing soap film system to investigate the connections and differences between (quasi) 2D wakes and 3D wakes generated behind oscillating and tandem cylinders. Wake structure is identified through flow visualization. Inspired by the work of Williamson and collaborators, we investigate the wake structure behind a circular cylinder forced to oscillate transverse to the flow. We map the boundaries of the different wake modes with variations in the amplitude and frequency of oscillation, and we discuss how our quasi-2D results compare with 3D results from the literature. We also consider the wake interaction of two stationary cylinders arranged in tandem. Existing literature disagrees on the critical cylinder spacing that gives changes in the wake mode. We examine this point and discuss the connections and distinctions between our quasi-2D experiments, 2D simulations, and results from the literature.

Sunday, November 22, 2015 2:10PM - 4:20PM — Session D19 Boundary Layers: Structure and Turbulence I — 207 — Stephen Garrett, University of Leicester

2:10PM D19.00001 Large scale structures in transitional pipe flow LEO HELLSTRÖM, Princeton University, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton, ALEXANDER SMITS, Princeton University, Monash University — We present a dual-plane snapshot POD analysis of transitional pipe flow at a Reynolds number of 5400, based on the pipe diameter. The time-resolved high-speed PIV data were simultaneously acquired in two planes, a cross-stream plane (2D-3C) and a streamwise plane (2D-2C) on the pipe centerline. The two light sheets were orthogonally polarized, allowing particles situated in each plane to be viewed independently. In the snapshot POD analysis, the modal energy is based on the cross-stream plane, while the POD modes are calculated using the dual-plane data. We present results on the emergence and decay of the energetic large scale motions during transition to turbulence, and compare these motions to those observed in fully developed turbulent flow.

2:23PM D19.00002 Competing stability modes in vortex structure formation STEPHEN GARRETT, J. PAUL GOSTELOW, ALDO RONA, W. ANDREW McMULLAN, Department of Engineering, University of Leicester, UK — Nose cones and turbine blades have rotating components and represent very practical geometries for which the behavior of vortex structures is not completely understood. These two different physical cases demonstrate a common theme of competition between mode and vortex types. The literature concerning boundary-layer transition over rotating cones presents clear evidence of an alternative instability mode leading to counter-rotating vortex pairs, consistent with a centrifugal instability. This is in contrast to co-rotating vortices present over rotating disks that oscillate transverse to the flow. We map the boundaries of the different wake modes with variations in the amplitude and frequency of oscillation, and we discuss how our quasi-2D results compare with 3D results from the literature. We also consider the wake interaction of two stationary cylinders arranged in tandem. Existing literature disagrees on the critical cylinder spacing that gives changes in the wake mode. We examine this point and discuss the connections and distinctions between our quasi-2D experiments, 2D simulations, and results from the literature.

2:36PM D19.00003 The relation between skin friction fluctuations and turbulent fluctuating velocities in turbulent boundary layers CARLOS DIAZ DANIEL, SYLVAIN LAIZET, JOHN CHRISTOS VASSILICOS, Imperial College London — The Townsend-Perry hypothesis of wall-attached eddies relates the friction velocity at the wall to velocity fluctuations at a position y from the wall, resulting in a wavenumber range where the streamwise fluctuating velocity spectrum scales as $E(k) \sim k^{-1}$ and the corresponding structure function scales as $u^2 = k^{-2}$ in the corresponding length-scale range. However, this model does not take into account the fluctuations of the skin friction velocity, which are in fact strongly intermittent. A DNS of zero-pressure gradient turbulent boundary layer suggests a 10 to 15 degree angle from the lag of the peak in the cross-correlations between the fluctuations of the shear stress and streamwise fluctuating velocities at different heights in the boundary layer. Using this result, it is possible to refine the definition of the attached eddy range of scales, and our DNS suggests that, in this range, the second order structure function depends on filtered skin friction fluctuations in a way which is about the same at different distances from the wall and different local Reynolds numbers.

2:49PM D19.00004 Study of the near field wake of trips generating an artificially thick turbulent boundary layers EDUARDO RODRIGUEZ LOPEZ, PAUL J.K. BRUCE, OLIVER R.H. BUXTON, Imperial College London — The properties of an artificially thick turbulent boundary layer are influenced by its formation mechanism. Previous work has shown that wake or wall-driven mechanisms dominate boundary layer development depending on the trips aspect ratio. The current study characterizes these two formation mechanisms through the use of high-speed PIV in the near wake of obstacles arrays on a flat plate in a wind tunnel. The time resolved velocity field is studied using Optimal Mode Decomposition (OMD) generating a low order model which captures the representative motions. Results corroborate the original hypothesis and show that these mechanisms are divided in two families: (i) High aspect ratio trips (cylinders) generate vortices with a wall-normal axis which do not transfer information between the wall and the wake of the obstacle. In this case, the boundary layer growth is wall-driven entraining the low-momentum highly turbulent flow above it. (ii) Low aspect ratio trips generate spanwise vorticity increasing the influence of the obstacles wake in the wall region (wake-driven mechanism). A high level of correlation with the velocity fluctuations at the wall is maintained in case (ii) for the whole wake while in case (i) the correlation vanishes for heights smaller than half obstacle.
3:02PM D19.00005 Large scale motions of thermal transport in a turbulent channel.
SURANGA DHARMARATHNE, Texas Tech University, USA, MUTAR TUTKUN, Institute for Energy Technology, Norway, GUILLERMO ARAYA, Texas Tech University, USA, STEFANO LEONARDI, University of Texas at Dallas, USA, LUCIANO CASTILLO, Texas Tech University, USA — The importance of large-scale structures (LMS) on thermal transport in a turbulent channel flow at friction number of 394 is investigated. Two-point correlation analysis reveals that LSM which significantly contribute to turbulence kinetic energy and scalar transport is a remnant of a hairpin packet. Low-order mode representation of the original fields using proper orthogonal decomposition (POD) unveils that the most dominant mode that transports \( \langle u' \rangle \) is 3-4 channel half-heights long and such structure which contribute to scalar transport is 2-4 channel half-heights long. Consequently, the study discloses that LMSs are effective in transporting both streamwise component of turbulence kinetic energy and scalar variances.

3:15PM D19.00006 Evolution of vortex-surface fields in the K-type temporal transition in channel flow. YAO QIN ZHAO, YUE YANG, SHIYI CHEN, State Key Laboratory for Turbulence and Complex Systems, College of Engineering, Peking University — The vortex-surface field (VSF), a Lagrangian-based method (Yang and Pullin., J. Fluid Mech., 685, 2011), is used to study the evolution of vortex structures in the K-type temporal transition in channel flow. Iso-surfaces of an evolving VSF can represent vortex surfaces composed of vortex lines in evolution. Since the VSF was only used in simple flows with periodic boundary conditions, the validity of different wall boundary conditions for the VSF transport equation is first discussed, and then the Neumann boundary condition is applied in the implementation. The initial VSF is uniquely determined by proposed criteria, and its iso-surfaces are streamline-spanwise planes. Compared with the evolution of material surfaces with the same initial scalar field, the VSF evolution can capture the topological changes of vortex structures that are induced by the interaction between different hairpin-like vortices. It is noted that the vortex reconnection is a critical mechanism for the breakdown of vortical structures in the late transitional stage, and is challenging to be characterized via Eulerian-based methods.

3:28PM D19.00007 Unified concepts in internal and external wall-turbulence1. YONG SEOK KWON, CHENG CHIN, NICHOLAS HUTCHINS, JASON MONTY, Univ of Melbourne — Recently, Kwon, Hutchins and Monty (J. Fluid Mech., submitted) reported that the oscillation of turbulent/non-turbulent interface (TNTI) in a turbulent boundary layer can contaminate the fluctuating velocity component under the traditional Reynolds decomposition, which overestimates the extent of spatial coherence of velocity fluctuations. In order to overcome this issue, they proposed a new velocity decomposition method which removes the influence of the TNTI oscillation from the velocity fluctuations. Extension of their decomposition method to internal flows via the analogy between the free-stream and quiescent core (Kwon et al., J. Fluid Mech., vol. 751, 2014, pp. 228-254) reveals that both the scale and structure of turbulence are, in fact, similar in internal and external flows even up to the edge of the outer region. Based on this result, a new conceptual model for internal flows, that is similar to external flows, is proposed.

1The authors wish to acknowledge the financial support of the Australian Research Council and the Defence Science and Technology Organisation.

3:41PM D19.00008 Multi-scale geometric analysis of evolving Lagrangian structures in the compressible transitional boundary layer at \( Ma = 0.7 \). WENJIE ZHENG, YUE YANG, SHIYI CHEN, State Key Laboratory for Turbulence and Complex Systems, College of Engineering, Peking University, Beijing 100871, China — Evolutionary geometry of flow structures in a compressible transitional boundary layer at \( Ma = 0.7 \) is investigated from a Lagrangian perspective. The Lagrangian structures in the transition are extracted from the Lagrangian scalar field by a moving window filter, and then their geometry is characterized by the multi-scale and multi-directional geometric analysis (Yang and Pullin, J. Fluid Mech., 674, 2011), including the averaged inclination and sweep angles at different scales ranging from one half of the boundary layer thickness to several viscous length scales \( \delta \). The results show that averaged angles are almost unaltered for different scales before the transition. As the transition occurs, averaged inclination angles increase and sweep angles decrease rapidly with increasing reference time. Furthermore, the orientation changes more significantly for structures with small scales than large scales. In the late stage of transition, the averaged inclination angle of small-scale structures with the length scale \( \sim O(10) \delta \) is 42°, and the averaged sweep angle in the logarithm law region is approximately 30°.

1This work is supported in part by NSFC (No. 11472015) and the Thousand Young Talent Program of China.

3:54PM D19.00009 Spatial organisation of large scale structures in turbulent boundary layers. FELIX EICH, NICO REUTHER, MATTHEW BROSS, CHRISTIAN KAEHLER, Bundeswehr University, Munich — The experimental investigation of the spatial organization of large scale structures in a turbulent boundary layer at large Reynolds numbers is a difficult task due to the size of the turbulent structures and their mutual distance in streamwise and spanwise directions. However, by aligning various PIV systems side by side it was possible to resolve all relevant scales simultaneously in the Reynolds number range \( Re = 1800-13355 \). This range of Reynolds numbers was selected to fully characterise the relevant scaling of these large structures. The measurements were performed in wall-normal planes at several selected wall distances, to examine the variation of the average width and spanwise distance between the large scale flow structures. The acquired vector fields were analysed by means of two-point correlations. Form these correlations the average width and streamwise extent of large-scale structures was determined. Using conditional correlations, it was possible to separate and characterise the high and low speed structures. The statistical multipoint analysis shows that there are distinct variations between high and low speed structures as well as between the large-scale structures at different wall distances.

4:07PM D19.00010 Stress Boundary layer Development in Planar flow of Viscoelastic Fluids. NARIMAN ASHRAFI, MEYSAM MOHAMADALI, Young Researchers and Elites Club, Science and Research Branch, Islamic Azad University, Tehran, Iran — Two-dimensional steady planar creeping flow of the nonlinear viscoelastic Upper Convected Maxwell (UCM) fluid along a flat plate is analyzed for high Weissenberg numbers, Wi. The viscoelastic boundary layer, formed in a thin region closer to the wall in which the relaxation times are recovered. By means of similarity transformations the non-linear momentum and constitutive equations in each layer transform into a system of highly nonlinear coupled ordinary differential equations. The proper similarity variable is found that asymptotically matches each two adjacent layers. The numerical simulation shows that at the outer layer, the velocity profile changes linearly with the similarity variable meaning that no velocity boundary layer is developed. In general, the boundary layer is formed in all three stress components in different fashions. The stress boundary layer divides the flow into two separate regions of viscoelastic and elastic flows, in addition to the top outer flow. The viscoelastic region is completely bounded in two directions (x and y) for horizontal normal stress, \( \tau_{xy} \), and shear stress, \( \tau_{xy} \). Finally, it is observed that the stress boundary layer for vertical stress, \( \tau_{yy} \), is formed only in x direction.

Sunday, November 22, 2015 2:10PM - 4:20PM –
Session D20 Turbulence: Compressible Flows 208 - Daniel Livescu, Los Alamos National Laboratory
In many natural and engineering systems, turbulence is found to interact with shock waves. Thus, canonical interactions between isotropic turbulence and a normal shock have been studied extensively, theoretically and numerically, though theories assume the shock to be a discontinuity and most simulations have used shock-capturing schemes which may miss details of the structure of the shock, especially for weak shocks in relatively strong turbulence. We present results on this regime from shock-resolving direct numerical simulations at a range of Reynolds and Mach numbers. Our focus is on the shock structure and the effect on turbulence downstream of the shock. We study the distribution of velocity gradients, in particular dilatation across the shock and compare with theory available. We characterize turbulent shock jumps which are found to depart from the laminar theory as they depend not only on the mean Mach number but also on the Reynolds and turbulent Mach number. Changes experienced by thermodynamic variables across the shock will also be discussed.

1The authors gratefully acknowledge the support of AFOSR

2:10PM D20.00001 Subgrid-scale backscatter after the shock-turbulence interaction

2:23PM D20.00002 Turbulence generation through intense localized sources of energy1, AGUSTIN MAQUI, DIEGO DONZIS, Texas A&M University — Mechanisms to generate turbulence in controlled conditions have been studied for nearly a century. Most common methods include passive and active grids with a focus on incompressible turbulence. However, little attention has been given to compressible flows, and even less to hypersonic flows, where phenomena such as thermal non-equilibrium can be present. Using intense energy from lasers, extreme molecule velocities can be generated from photo-dissociation. This creates strong localized changes in both the hydrodynamics and thermodynamics of the flow, which may perturb the flow in a way similar to an active grid to generate turbulence in hypersonic flows. A large database of direct numerical simulations (DNS) are used to study the feasibility of such an approach. An extensive analysis of single and two point statistics, as well as spectral dynamics is used to characterize the evolution of the flow towards realistic turbulence. Local measures of enstrophy and dissipation are studied to diagnose the main mechanisms for energy exchange. As commonly done in compressible flows, dilatational and solenoidal components are separated to understand the effect of acoustics on the development of turbulence. Further results for cases that assimilate laboratory conditions will be discussed.

1The authors gratefully acknowledge the support of AFOSR

2:36PM D20.00003 Shock-resolving direct numerical simulations of strong turbulence interacting with a normal shock wave1, CHANG-HSIN CHEN, DIEGO DONZIS, Texas A&M University — In many natural and engineering systems, turbulence is found to interact with shock waves. Thus, canonical interactions between isotropic turbulence and a normal shock have been studied extensively, theoretically and numerically, though theories assume the shock to be a discontinuity and most simulations have used shock-capturing schemes which may miss details of the structure of the shock, especially for weak shocks in relatively strong turbulence. We present results on this regime from shock-resolving direct numerical simulations at a range of Reynolds and Mach numbers. Our focus is on the shock structure and the effect on turbulence downstream of the shock. We study the distribution of velocity gradients, in particular dilatation across the shock and compare with theory available. We characterize turbulent shock jumps which are found to depart from the laminar theory as they depend not only on the mean Mach number but also on the Reynolds and turbulent Mach number. Changes experienced by thermodynamic variables across the shock will also be discussed.

1The authors gratefully acknowledge the support of AFOSR

2:49PM D20.00004 Lagrangian Statistics of Velocity-gradient in Compressible Turbulence, MOHAMMAD DANISH, SAWAN SUMAN, BALAJI SRINIVASAN, Indian Inst of Tech-New Delhi — The Lagrangian-based analysis of various flow quantities, in particular the velocity-gradient tensor, has been a cornerstone in the study of turbulence. The physics of many important turbulence processes such as cascading, scalar mixing, material element deformation etc. can be explained in terms of the dynamics of velocity-gradient tensor itself. In this context, we present the Lagrangian statistics of the invariants of the velocity-gradient tensor over a wide range of Mach and Reynolds numbers in compressible turbulence. For this purpose, we track a large number of fluid particles in well resolved direct Navier-Stokes (DNS) simulations of decaying compressible turbulence. We show that these statistics significantly depend on the existence of shocklets. Specifically, we observe that the presence of shocklets tends to increase the decorrelation rate of Lagrangian autocorrelations of velocity-gradient tensor.

3:02PM D20.00005 Bulk viscosity effect on freely decaying compressible homogeneous isotropic turbulence, SHAWU PAN, ERIC JOHNSEN, University of Michigan, Ann Arbor — Despite growing interests in compressible turbulence, the effect of bulk viscosity has been long ignored. For certain gases, the bulk viscosity may be 1000 times greater than the shear viscosity and thus modify energy transfer and dissipation mechanisms. In this study, we use direct numerical simulations to investigate the role of bulk viscosity on decaying isotropic compressible turbulence. Our results show that bulk viscosity exhibits a negligible decrease on enstrophy, but moderate and significant increases on the turbulent kinetic energy and Taylor-scale Reynolds number, respectively. A Helmholtz decomposition of the velocity field indicates that the bulk viscosity has a negligible effect on the solenoidal part, but exhibits a cross-scale effect on the dilatational component.

3:15PM D20.00006 A Mesoscopic Model for the Description of Small-scale Inhomogeneity in Turbulent Flows with Thermal Nonequilibrium, VENKAT RAMAN, ROMAIN FIEVET, University of Michigan, PETER CLARKE, PHILIP VARGHESE, The University of Texas at Austin — Turbulent mixing of nonequilibrium flows exhibit certain anomalous properties as compared to equilibrium flows. In high-speed flows where the vibrational or rotational relaxation occurs over time and length-scales comparable to flow-through times, there a unique non-homogeneity introduced at the small-scales: The molecules are spatially homogeneous but their internal motions still exhibit inhomogeneity. This differential relaxation rate poses hurdles in the direct numerical simulation of turbulent nonequilibrium flows. In this work, a mesoscopic description of this homogeneity is introduced. Using a probabilistic model, the inhomogeneous vibrational modes are tracked as a function of space and time. Further, direct numerical solutions of the extended Boltzmann equation are used to develop mesoscopic closure models for this stochastic description. Simulations of representative scramjet isolators are used to illustrate the applicability and relevance of this mathematical description.
3:28PM D20.00007 Compressible turbulent mixing: Effects of compressibility and Schmidt number, QIONGLIN NI, Department of Physics, University of Rome Tor Vergata — Effects of compressibility and Schmidt number on passive scalar in compressible turbulence were studied. On the effect of compressibility, the scalar spectrum followed the $k^{-5/3}$ inertial-range scaling and suffered negligible influence from compressibility. The transfer of scalar flux was reduced by the transition from incompressible to compressible flows, however, was enhanced by the growth of Mach number. The intermittency parameter was increased by the growth of Mach number, and was decreased by the growth of the compressive mode of driven forcing. The dependency of the mixing timescale on compressibility showed that for the driven forcing, the compressive mode was less efficient in enhancing scalar mixing. On the effect of Schmidt number ($Sc$), in the inertial-convective range the scalar spectrum obeyed the $k^{-5/3}$ scaling. For $Sc>1$, a $k^{-1}$ power law appeared in the viscous-convective range, while for $Sc<<1$, a $k^{-17/3}$ power law was identified in the inertial-diffusive range. The transfer of scalar flux grew over $Sc$. In the $Sc>1$ flow the scalar field rolled up and mixed sufficiently, while in the $Sc<<1$ flow that only had the large-scale, cloudlike structures. In $Sc>1$ and $Sc<<1$ flows, the spectral densities of scalar advection and dissipation followed the $k^{-5/3}$ scaling, indicating that in compressible turbulence the processes of advection and dissipation might deferring to the Kolmogorov picture. Finally, the comparison with incompressible results showed that the scalar in compressible turbulence lacked a conspicuous bump structure in its spectrum, and was more intermittent in the dissipative range.

3:41PM D20.00008 On the effect of finite-time correlations on the turbulent mixing in smooth chaotic compressible velocity fields, SIIM AINSAAR, Institute of Physics, University of Tartu, JAAN KALDA, Institute of Cybernetics, Tallinn University of Technology — For incompressible flows, most theoretical studies about turbulent mixing have used the Kraichnan model where the velocity field has zero correlation time. Most of their predictions are derived through (the ratios of) two sets of parameters: Lyapunov exponents (LEs), and their “diffusivities” (defined as the asymptotic values of $\text{Var}(\Lambda)$; $\Lambda$ is a finite-time LE for time $t$). However, for compressible flows, there is a serious mismatch between the theoretical predictions for these parameters, and both simulations and experiments. We present a simple theoretical model that derives the LEs and their “diffusivities” from basic statistics of the velocity gradient tensor $\nabla v$. For finite correlation times, there is a breakdown of universality: the ratios of these parameters do not depend only on the flow compressibility and the correlation time, but also on the determinant of $\nabla v$ — a parameter discussed very sparsely, so far. Our model is in a good agreement with previously unexplained studies regarding the role of finite time correlations [G. Boffetta et al, 2004]. Our mapping from the statistics of $\nabla v$ to the LEs and their “diffusivities” extends a wide range of existing analytical “Kraichnanian” results to real time-correlated flows.

3:54PM D20.00009 Dynamics of Strongly Compressible Turbulence, COLIN TOWERY, Univ of Colorado - Boulder, ALEXEI POLUDNENKO, Naval Research Laboratory, PETER HAMLINGTON, Univ of Colorado - Boulder — Strongly compressible turbulence is very unusual in comparison to “normal,” weakly compressible and incompressible turbulence, which is relatively well understood. Strongly compressible turbulence is characterized by large variations in the thermodynamic state of the fluid in space and time, including excited acoustic modes, strong, localized shock and rarefaction structures, and rapid heating due to viscous dissipation. The exact nature of these thermo-fluid dynamics has yet to be discerned, which greatly limits the ability of current computational engineering models to successfully treat these problems. New direct numerical simulation (DNS) results of strongly compressible isotropic turbulence will be presented along with a framework for characterizing and evaluating compressible turbulence dynamics and a connection will be made between the present diagnostic analysis and the validation of engineering turbulence models.

4:07PM D20.00010 LES prediction and analysis of the aero-optical environment around a 3-D turret, EDWIN MATHEWS, KAN WANG, MENG WANG, ERIC JUMPER, University of Notre Dame — Using wall-modeled large-eddy simulation, a Mach 0.4 flow over a hemisphere-on-cylinder turret at the experimental Reynolds number of $Re_D = 2.3 \times 10^6$ is simulated to study the aero-optical distortions caused by turbulent density fluctuations. The optical distortions are calculated at over 250 viewing angles during the simulation to thoroughly investigate the optical environment around the turret. Flow field and optical results show good comparisons with experimental measurements. A large database of three-dimensional velocity and density fields is generated for study of the connection between global flow dynamics and local optical distortions. Proper orthogonal decomposition and dynamic mode decomposition are applied to both the distorted wavefronts and the flow-field database. A method of reconstructing the optical wavefronts from the density field modes is investigated. Relations between prominent flow features and wavefront components including tip/tilt and higher-order effects will be discussed.

1Supported by HEL-JTO through AFSOR Grant FA9550-13-1-0001

Sunday, November 22, 2015 2:10PM - 4:20PM — Session D21 Turbulence: Mixing
209 - Stavros Tavoularis, University of Ottawa

2:10PM D21.00001 Turbulence structure and scalar diffusion in uniformly sheared flow distorted by a grid, STAVROS TAVOULARIS, JOVAN NEDIC, University of Ottawa — Uniformly sheared flow, generated in a wind-tunnel by a shear generator, was let to develop a self-similar, strongly anisotropic turbulence structure and then it was disturbed by grids having square meshes with spacings larger than, comparable to and smaller than the spacing of the shear generator; a “fractal” grid was also used. The multi-scale, non-equilibrium turbulence structure downstream of each grid was documented and differences from the structures of the undisturbed shear flow and grid turbulence were identified. In addition, heat was injected passively from a line source located downstream of the grid and the growth of the heated plume under different conditions was examined.

1Supported by NSERC
2:23PM D21.00002 Turbulent diffusion from a heated line source in non-equilibrium grid turbulence\textsuperscript{1}, JOVAN NEDIC, STAVROS TAVOULARIS, University of Ottawa — We have investigated turbulent diffusion of heat injected passively from a line source in equilibrium and non-equilibrium grid-generated turbulence, which are, respectively, flows in which the value of the non-dimensional rate of kinetic energy dissipation is constant or changes with streamwise distance from the grid. We used three grids with uniform square meshes and one fractal square grid (FSG), all of the same solidity, to generate non-equilibrium and equilibrium turbulence in a wind-tunnel. The regular grids have mesh sizes that are comparable to the first (RG160), second (RG80) and fourth (RG18) iterations of the fractal grid. The heated line source was inserted on the centre-plane of the grids at either of two downstream locations or an upstream one and it spanned the entire width of the wind-tunnel. We found that RG80 produced the greatest heat diffusion, followed by FSG, RG80 and RG18, in this order. The apparent turbulent diffusivity produced by the four grids also decreased in the same order. These findings conform with Taylor’s theory of diffusion by continuous movements. Moreover, the present study demonstrates that the fractal space-scale unfolding (SSU) mechanism does not apply to grids with the same solidity but different effective mesh sizes.

\textsuperscript{1}Supported by NSERC.

2:36PM D21.00003 ABSTRACT WITHDRAWN –

2:49PM D21.00004 Analysis of Cliff-Ramp Structures in Homogeneous Scalar Turbulence by the Method of Line Segments, MICHAEL GAUDING, TU Bergakademie Freiberg, JENS HENRIK GOEBBERT, Juélich Aachen Research Alliance, NORBERT PETERS, RWTH Aachen University, CHRISTIAN HASSE, TU Bergakademie Freiberg — The local structure of a turbulent scalar field in homogeneous isotropic turbulence is analyzed by direct numerical simulations (DNS). A novel signal decomposition approach is introduced where the signal of the scalar along a straight line is partitioned into segments based on the local extremal points of the scalar field. These segments are then parameterized by the distance between adjacent extremal points and a segment-based gradient. Joint statistics of the length and the segment-based gradient provide novel understanding about the local structure of the turbulent field and particularly about cliff-ramp-like structures. Ramp-like structures are unveiled by the asymmetry of joint distribution functions. Cliff-like structures are further analyzed by conditional statistics and it is shown from DNS that the width of cliffs scales with the Kolmogorov length scale.

3:02PM D21.00005 A Simple Parameterization of Mixing of Passive Scalars in Turbulent Flows\textsuperscript{1}, AJITHSHANTHAR NITHIANANTHAM, KARAN VENAYAGAMOORTHY, Colorado State University — A practical model for quantifying the turbulent diascalar diffusivity is proposed as $K_{ij} = K_{ij}^1 \equiv \frac{L_T}{\gamma} \frac{k^{1/2}}{\bar{L}_T}$, where $L_T$ is defined as the Thorpe length scale, $k$ is the turbulent kinetic energy and $\gamma$ is one-half of the mechanical to scalar time scale ratio, which was shown by previous researchers to be approximately 0.7. The novelty of the proposed model lies in the use of $L_T$, which is a widely used length scale in stably stratified flows (almost exclusively used in oceanography), for quantifying turbulent mixing in unstratified flows. $L_T$ can be readily obtained in the field using a Conductivity, Temperature and Depth (CTD) profiler. The turbulent kinetic energy is mostly contained in the large scales of the flow field and hence can be measured in the field or modeled in numerical simulations. Comparisons using DNS data show remarkably good agreement between the predicted and exact diffusivities.

\textsuperscript{1}Office of Naval Research and National Science Foundation

3:15PM D21.00006 Space-scale unfolding mechanism in canonical multi-scale flows\textsuperscript{1}, PAWEL BAJ, PAUL J.K. BRUCE, OLIVER R.H. BUXTON, Imperial College London — Some recent studies on fractal generated turbulence revealed a highly increased transverse turbulent scalar flux downstream of fractal grids compared to regular grids. The complexity of these flows makes it impossible to track the origins of this phenomenon, often referred to as the space-scale unfolding mechanism (SSU). Thus research on flows past canonical examples of single and multi-scale obstacles, which are arrays of bars of the same and different thicknesses, was undertaken in order to investigate the SSU’s roots. The velocity field and the scalar concentration field were measured simultaneously downstream of the obstacles by means of particle image velocimetry and laser induced fluorescence techniques. It is observed that the concentration field behind the multi-scale obstacle undergoes intense quasi-periodic transverse scalar bursts, which are believed to be the manifestation of the SSU, whereas such events are either weak or absent in the single scale configuration. Investigation of the velocity field reveals a phase locking between wakes of different scale objects in terms of the phase-conditioned transverse integral length scale. Both phenomena are observed to be triggered at the downstream position corresponding to the wakes’ intersection point.

\textsuperscript{1}The authors acknowledge support form the EU through the FP7 Marie Curie MULTISOLVE project (grant agreement No. 317269).

3:28PM D21.00007 Turbulent mixing by buoyancy-driven flows in long tubes, STUART DALZIEL, DAMTP, University of Cambridge, LIYONG ZOU, National Laboratory for Shock Wave and Detonation Physics, China Academy of Engineering Physics — We explore the buoyancy-driven turbulent flow that develops due to a change of orientation for a long tube initially filled with a statically stable stratification. For simple orientation histories, the flow may be characterised by the low mixing of a gravity current, or the much greater mixing of Rayleigh-Taylor instability. However, precise details of the orientation history can prove to be important. We present experimental results from a range of orientation histories, exploring both the temporal development of the flow and the level of mixing achieved.

3:41PM D21.00008 Structural Composition and Turbulent Mixing Mechanisms of a Subsonic Boundary Layer\textsuperscript{1}, PATRICK BECHLARS, RICHARD SANDBERG, University of Southampton, AERODYNAMICS AND FLIGHT MECHANICS GROUP - SOUTHAMPTON TEAM — Turbulent mixing is a key mechanism for redistributing energy in a wide range of flows. The effect of this mixing on the flows is similar to that of viscous diffusion and the process is therefore often described as turbulent diffusion. Turbulence models based on the Boussinesq approximation rely on the accuracy of the model’s description of the mixing to capture the correct energy redistribution. In this presentation the basic mechanism is illustrated using a subsonic turbulent boundary layer (TBL) as a case study, and the direct influence of turbulence on the mean flow is quantified. Through a characteristic analysis the structures involved in the mixing mechanism are identified and further analyzed. The key structures for the mixing in a TBL are large clusters of smaller turbulent structures that are known as large scale motions (LSMs). While the smaller structures are located in the near-wall region they mainly align in the stream-wise direction and pack densely, which affects production and dissipation. Within the LSMS the single vortices reach towards the outer regions and develop an arbitrary alignment as soon as their distance to the wall is sufficiently large. The discussed mechanisms are not limited to TBLs and a comparison to a jet flow is provided in the talk.

\textsuperscript{1}The authors acknowledge EPSRC for supporting this project under the grand number EP/I003754/1.
3:54PM D21.00009 Wave propagation in inhomogeneous media as turbulent mixing in six-dimensional incompressible flow. JAAK KALDA, MIHKEL KREE, Institute of Cybernetics, Tallinn University of Technology — Using the approximation of geometrical optics, light propagation in media with fluctuating coefficient of refraction can be described as Hamiltonian dynamics of wave vectors in 6-dimensional phase space where the spatial coordinates are complemented by the respective wave vector components. Hence, according to the Liouville’s theorem, the dynamics of the wave front can be described as mixing in an incompressible 6D velocity field. As the wave energy is transferred along the ray trajectories, the energy density fluctuations follow the dilution of the wave front. We use the theory of turbulent mixing to show that the intensity-distribution of speckles (regions of high energy density) follows a power law, and to derive the scaling exponents. If the velocity field were isotropic, these exponents would be determined by the dimensionality of the flow. However, there is a strong anisotropy of the field due to the asymmetry between the spatial and wave vector coordinates. Also, the effective dimensionality of the flow is reduced by one due to the energy (wave frequency) conservation law: any ray trajectory is bound to a five-dimensional manifold within the 6D phase space. Implications of the anisotropy, and of the effective reduction of the dimensionality are studied numerically.

1The research was supported by the European Union Regional Development Fund (Centre of Excellence TK124: Centre for Nonlinear Studies).

4:07PM D21.00010 Probability density function of a puff dispersing from the wall of a turbulent channel. QUOC NGUYEN, DIMITRIOS PAPAVASSILIOU, The University of Oklahoma — Study of dispersion of passive contaminants in turbulence has proved to be helpful in understanding fundamental heat and mass transfer phenomena. Many simulation and experimental works have been carried out to locate and track motions of scalar markers in a flow. One method is to combine Direct Numerical Simulation (DNS) and Lagrangian Scalar Tracking (LST) to record locations of markers. While this has proved to be useful, high computational cost remains a concern. In this study, we develop a model that could reproduce results obtained by DNS and LST for turbulent flow. Puffs of markers with different Schmidt numbers were released into a flow field at a frictional Reynolds number of 150. The point of release was at the channel wall, so that both diffusion and convection contribute to the puff dispersion pattern, defining different stages of dispersion. Based on outputs from DNS and LST, we seek the most suitable and feasible probability density function (PDF) that represents distribution of markers in the flow field. The PDF would play a significant role in predicting heat and mass transfer in wall turbulence, and would prove to be helpful where DNS and LST are not always available.

Sunday, November 22, 2015 2:10PM - 4:20PM — Session D22 Turbulent Boundary Layers I — 210 - Ronald Panton

2:10PM D22.00001 Vorticity Fluctuations Require a Two-term Asymptotic Representation. RONALD PANTON, Retired — Channel flow DNS data produced by several authors is analyzed. In the inner region, the vorticity fluctuations, $\langle \omega \omega \rangle$, require two-term asymptotic expansions. The first terms are scaled by the mixed velocity ($U_{u^+} r_1$). They are the viscous response of imposed potential fluctuations, decay exponentially, and therefore do not require matching terms in the outer region. The first term is zero for the normal component, $\langle \omega \omega \omega \rangle$. The second terms are scaled by $U_{u^+}$ with a gauge function $U_{u^+} (R e^+)$. They are active in the turbulence. In the log region they have an overlap behavior $\sim G$ or $C_0 / (y / \delta)$. This behavior demands a rescaling in the outer region where the proper vorticity scale is $\tau \eta = U / \varepsilon = (\nu / \nu)$ and $C_0 / (y / \delta)$. This is the Kolmogorov time scale appropriate for viscous dissipation. In the outer region all components scale nicely with $R e^+$ and have similar magnitudes.

2:23PM D22.00002 Generalized higher order two-point moments in turbulent boundary layers. XIANG YANG, The Johns Hopkins University, IVAN MARUSIC, The University of Melbourne, CHARLES MENEVEAU, The Johns Hopkins University — Generalized higher order two-point moments such as $\langle u' v' \rangle (x, y, z)$ and $\langle u' u' \rangle (x, y, z)$ are active in the turbulence. These are passive scalars and have similar magnitudes.

2:36PM D22.00003 A multi-layer description of Reynolds stresses in canonical wall bounded flows. XI CHEN, FAZLE HUSSAIN, Texas Tech University, ZHEN-SU SHE, Peking University — A complete description of the Reynolds stress tensor is obtained for all three canonical wall turbulence (channel, pipe and turbulent boundary layer - TBL). The result builds on a multi-layer description of length (order) functions and their ratios, including viscous sublayer, buffer layer, meso-layer for the near wall (inner) region, and bulk flow or a central core (absent in TBL) for the outer region. It is shown that the streamwise mean kinetic-energy profile is quantified with high accuracy over the entire flow domain. The model contains only three $R e$-dependent parameters for Reynolds number ($R e$) covering nearly three decades. Furthermore, the inner peak location is predicted to be invariant at $y^+ = 15$, while its magnitude shows notable $R e$ and geometry effects, predicted to be 0.92 for high $R e$’s pipe flows. A mechanism is proposed for the emergence of outer peak in pipes, whose magnitude is predicted to scale as $R e_{p}^{0.05}$ beyond a critical $R e_{p}$ about 10^{4}. The recently reported logarithmic dependence in the bulk is recovered, but with an alternative explanation. The result is successfully extended to TBL flows by a fractional total stress and an absence of core. Equally accurate descriptions of vertical and spanwise kinetic-energy are also presented for the three flows. The result has been used to modify turbulent engineering models (i.e. $k-\omega$ model) with significant improvement.
Interaction of free-stream turbulence with a turbulent boundary layer, R. J. HEARST, EDA DOGAN, B. GANAPATHISUBRAMANI, University of Southampton — The interaction of free-stream turbulence with a turbulent boundary layer is investigated with planar particle image velocimetry. An active grid consisting of a series of wings mounted to stepper motor-actuated rods is used to generate free-stream turbulence in a wind tunnel. In this study, turbulence intensities between 7% and 13% are investigated. The boundary layer is formed over a plate mounted downstream of the grid. When the turbulent free-stream case is compared to a canonical turbulent boundary layer with an approximately laminar free-stream, the distinct angled structure of the velocity correlation maps becomes homogeneous much closer to the wall relative to the boundary layer thickness. However, when comparisons are made between the clear stream turbulent boundary layer at lower freestream turbulence levels, their structures are approximately the same. It is also found that there is a distinct lack of uniform momentum zones in the turbulent boundary layer for high levels of free-stream turbulence. This is particularly interesting given that the outer layer of the boundary layer still exhibits a single strong low-frequency peak in the velocity spectrum.

The time signature of the turbulent/non-turbulent interface over a turbulent boundary layer, A. LASKARI, R. J. HEARST, R. DE KAT, B. GANAPATHISUBRAMANI, University of Southampton — The turbulent/non-turbulent interface (TNTI) between a turbulent boundary layer and an approximately laminar free-stream is investigated with time-resolved planar particle image velocimetry (PIV). The turbulent boundary layer ($Re_{t} \approx 4000$) was formed on the floor of a water channel and was captured by a PIV system composed of a Phantom v414 4 mega-pixel camera and a LiteSpeed 304 Laser. Images were acquired at 800 Hz, which was sufficient to resolve the motions of the TNTI in time. The instantaneous TNTI can be located by thresholding the velocity field based on physical arguments. A threshold of $0.95U_{\infty}$ can be selected based on identifying the limits of a uniform momentum zone encompassing the free-stream. The time-resolved data set allows for instantaneous tracking of the TNTI topology and detection of its convection velocity, and thus provides novel insight into the TNTI. Preliminary findings based on a sample of the data set suggest the interface is connected at a velocity between $0.6U_{\infty}$ and $0.7U_{\infty}$. The final study will include analysis of the full data set of over 400,000 time-resolved images and a more accurate estimate of the convection velocity of the interface.

Internal shear layers and interfaces in turbulent boundary layers, C. DE SILVA, J. PHILIP, D. SQUIRE, N. HUTCHINS, I. MARUSIC, University of Melbourne — We examine regions of concentrated shear in turbulent boundary layers that are observed to demarcate patches of relatively uniform streamwise momentum. To this end, we employ particle image velocimetry databases that span over a decade of Reynolds numbers ($Re_{t} = O(10^3 - 10^4)$) which are tailored to capture the larger spatial features in the order of the boundary layer thickness, but with adequate spatial resolution to resolve most structural features. These databases are also complimented by experiments with enhanced spatial resolution in the order of the Kolmogorov scale, albeit over a narrow spatial extent. Our analysis quantitatively characterises several recurrent structural features, together with their associated scaling arguments. The Reynolds number dependence of these features is also investigated. Preliminary comparisons are also drawn between the observed edges of uniform momentum zones to the turbulent/non-turbulent interface at the edge of the boundary layer.

On the development of turbulent boundary layer with wall transpiration, M. FERRO, R. S. DOUGLAS III, B. FALLENIUS, J. H. M. FRANSSON, Linné Flow Centre, KTH Royal Institute of Technology — An experimental study of the development of the transpired boundary layer in zero pressure gradient is carried out on a 6.4 m long hydrodynamically smooth and perforated plate. The relatively longer development length of the present perforated plate compared to the ones used in previous studies allows us to investigate whether an asymptotic suction boundary layer with constant thickness is achieved for the turbulent state, analogously to what happens in the laminar state. Velocity profiles are obtained via hot-wire anemometry while the wall shear stress is measured at several streamwise locations with hot-film and wall-wire probes as well as with oil-film interferometry. The threshold suction coefficient above which relaminarization starts to occur is examined. The scaling of the mean velocity and of higher order velocity moments is discussed in light of the measured wall shear stress data.

Restricted nonlinear large-eddy simulations of wall-turbulence, D. F. GAYME, J. BRETHEIM, C. MENEVEAU, Johns Hopkins University — The prominence of streamwise elongated structures motivates the use of a restricted nonlinear (RNL) model for studying the dynamics of wall-turbulence. This model is formed by partitioning the Navier Stokes equations into a streamwise constant mean flow interacting with a streamwise varying perturbation field in which the nonlinear perturbation-perturbation interactions are neglected. RNL simulations have been shown to support self-sustaining turbulence with a mean profile and structural features consistent with DNS in a number of canonical flows. Recent results have shown that the accuracy of the statistical properties predicted by the RNL model at higher Reynolds numbers can be significantly improved by limiting the streamwise varying Fourier components in the perturbation dynamics; this “band-limiting” procedure also opens the door for lower-cost simulations. Here, we extend the RNL model to high Reynolds number boundary layer flows through the creation of a RNL large-eddy simulation (LES) framework. The results indicate that a band-limited RNL-LES approach captures key flow statistics with a drastically reduced number of degrees of freedom versus a standard LES.

Direct numerical simulation of the incompressible temporally developing turbulent boundary layer, M. KOZUL, D. CHUNG, J. MONTY, University of Melbourne — We present a Direct Numerical Simulation of the incompressible temporally developing turbulent boundary layer. The approach is inspired by temporal simulations of flows which are generally thought of as developing in space, such as wakes and mixing layers, and has previously been applied to the study of compressible boundary layers. The flow is the turbulent counterpart to the laminar Stokes’ first problem where a fluid at rest is set into motion by a wall moving at constant velocity. An initial profile that models the effect of a wall-mounted trip wire is implemented allowing characterisation of initial conditions by a trip Reynolds number. Comparisons of various statistics demonstrate that the temporally developing boundary layer is a good model for the spatially developing boundary layer once initial conditions can be neglected. Analysis of similarity solutions point to their asymptotic collapse. We therefore propose its use as a tool with which to study further manipulations of the turbulent boundary layer. In this study, the development of the turbulent boundary layer under the condition of isotropic free-stream turbulence is investigated. Our temporal tool allows rapid and simplified simulation for a parameter space beyond the reasonable scope of costly spatial simulations.

ABSTRACT WITHDRAWN —

Support from the European Research Council of the Advanced Fluid Research On Drag reduction in Turbulence Experiments (AFRODITE) is acknowledged.

We gratefully acknowledge support from NSF grants IGERT 0801471, ECCS-1230788, and IIA-1243482 (WINDINSPIRE).

1Australian Research Council
Sunday, November 22, 2015 2:10PM - 4:20PM – Session D23 Biofluids: Active Fluids II 300 - Francesco Carrara, MIT

2:10PM D23.00001 A turbulence-induced switch in phytoplankton swimming behavior. FRANCISCO CARRARA, MIT; ANUPAM SENGUPTA, ROMAN STOCKER, ETH — Phytoplankton, unicellular photosynthetic organisms that form the basis of life in aquatic environments, are frequently exposed to turbulence, which has long been known to affect phytoplankton fitness and species succession. Yet, mechanisms by which phytoplankton may adapt to turbulence have remained unknown. Here we present a striking behavioral response of a motile species — the red-tide-producing raphidophyte Heterosigma akashiwo — to hydrodynamic cues mimicking those experienced in ocean turbulence. In the absence of turbulence, H. akashiwo exhibits preferential upwards swimming (‘negative gravitaxis’), observable as a strong accumulation of cells at the top of an experimental container. When cells were exposed to overturning in an automated chamber — representing a minimum experimental model of rotation by Kolmogorov-scale turbulent eddies — the population robustly split in two nearly equi-abundant subpopulations, one swimming upward and one swimming downward. Microscopic observations at the single-cell level showed that the behavioral switch was accompanied by a rapid morphological change. A mechanistic model that takes into account cell shape confirms that modulation of morphology can alter the hydrodynamic stress distribution over the cell body, which, in turn, triggers the observed switch in phytoplankton migration direction. This active response to fluid flow, whereby microscale morphological changes influence ocean-scale migration dynamics, could be part of a bet-hedging strategy to maximize the chances of at least a fraction of the population evading high-turbulence microzones.

2:23PM D23.00002 Non-classical size-dependent particle diffusion in active fluids1. ARVIND GOPINATH2, Department of Physics and Astronomy, Haverford College, ALISON PATTESON, PAULO ARRATIA, Department of Mechanical Engineering, SEAS, University of Pennsylvania — We experimentally investigate the effect of particle size on the motion of passive polystyrene spheres in suspensions of Escherchia coli, a flagellated bacterium that is approximately 2 microns long and swims using a sequence of runs punctuated by tumbles. Using particles spanning a range of sizes from 0.6 to 39 microns, we probe particle dynamics at both short and long time scales. In all cases, the particles exhibit super-diffusive ballistic behavior at short times before eventually transitioning to diffusive behavior. Surprisingly, the long-time hydrodynamic effective diffusivity is non-monotonic with particle size; an anomalous response that is fundamentally different from classical thermal diffusion. Consistent with recent theory, we find that for fixed bacterial type, the active contribution to particle diffusion can be predicted by a single dimensionless parameter, the Peclt number. Combining our experimental results, we propose a minimal model that allows us to predict the requirements for a peak in the diffusivity as well as the location and magnitude of the peak as a function of particle size and bacterial concentration. Our results have broad implications on characterizing active fluids using concepts drawn from classical (passive) thermodynamics.

1This work was supported by NSF-DMR-1104705 and NSF-CBET-1437482
2Current affiliation: School of Engineering, UC Merced

2:36PM D23.00003 Spontaneous ordering and vortex states of active fluids in circular confinement . MAXIME THEILLARD, BARATH EZHILAN, DAVID SAINTILLAN, UCSD — Recent experimental, theoretical and simulation studies have shown that confinement can profoundly affect self-organization in active suspensions leading to striking features such as directed fluid pumping in planar confinement, formation of steady and spontaneous vortices in radial confinement. Motivated by this, we study the dynamics in a suspension of biologically active particles confined in spherical geometries using a mean-field kinetic theory for which we developed a novel numerical solver. In the case of circular confinement, we conduct a systematic exploration of the entire parameter space and distinguish 3 broad states: no-flow, stable vortex and chaotic and several interesting sub-states. Our efficient numerical framework is also employed to study 3D effects and dynamics in more complex geometries.

2:49PM D23.00004 Subsonic to supersonic transition in density shocks of confined microswimmers. ALAN CHENG HOU TSANG, EVA KANSO, University of Southern California — Motile and driven particles confined in microfluidic channels exhibit interesting emergent behavior from propagating density bands to density shock waves. A deeper understanding of the physical mechanisms responsible for these emergent structures is relevant to a number of physical and biomedical applications. Here, we study in the context of an idealized model that a plug of microswimmers confined in a narrow channel and subject to a uniform external flow exhibit a transition of density shock waves from subsonic to supersonic regime depending on the intensity of the external flow. In the subsonic regime, density shock is formed at the back of the swimmers, whereas in the supersonic regime, density shock is formed at the front of the swimmers. This behavior results from a non-trivial interplay between hydrodynamic interactions and geometric confinement. We apply these findings to guide the development of novel mechanisms for controlling the emergent density distribution and average population speed, thus enabling processes such as sorting of cells in flow channels.

3:02PM D23.00005 The Force on a Boundary in Active Matter . JOHN BRADY, WEN YAN, California Institute of Technology — We present a general theory for determining the force (and torque) exerted on a boundary (or body) in active matter. The theory extends the description of passive Brownian colloids to self-propelled active particles and applies for all ratios of the thermal energy k_B T to the swimmer’s activity k_B T_s = \zeta U_0 R / 6, where \zeta is the Stokes drag coefficient, U_0 is the swim speed and \tau_R is the reorientation time of the active particles. The theory has a natural microscopic length scale over which concentration and orientation distributions are confined near boundaries, but the microscopic length does not appear in the force. The swim pressure emerges naturally and dominates the behavior when the boundary size is large compared to the swimmer’s run length \ell = U_0 \tau_R. The theory is used to predict the motion of bodies of all sizes immersed in active matter.

3:15PM D23.00006 On the distribution and swim pressure of run-and-tumble particles in confinement . ROBERTO ALONSO MATILLA, BARATH EZHILAN, DAVID SAINTILLAN, UC San Diego (UCSD) — The spatial and orientational distribution in a dilute active suspension of non-Brownian run-and-tumble spherical swimmers confined between two planar hard walls is calculated theoretically. Using a kinetic model based on coupled bulk/surface probability density functions, we demonstrate the existence of a concentration wall boundary layer with thickness scaling with the run length, the absence of polarization throughout the channel, and the presence of sharp discontinuities in the bulk orientation distribution in the neighborhood of orientations parallel to the wall in the near-wall region. Our model is also applied to calculate the swim pressure in the system, which approaches the previously proposed ideal-gas behavior in wide channels but is found to decrease in narrow channels as a result of confinement. Monte-Carlo simulations are also performed for validation and show excellent quantitative agreement with our theoretical predictions.
Inertia changes the stability of synchronized states in hydrodynamically coupled oscillators, SHANSHAN JIANG, LISA FAUCI, Tulane University — We examine the hydrodynamic interaction of two oscillators in a 2D fluid driven by a geometric switch. Motivated by the work of Kotar et al (PNAS, 107:17, 2010), the colloidal oscillators are modeled by circular membranes that support tensile forces on their boundary and forces due to an external trap that switches between two spatially embedded walls, depending upon the position of the oscillator. Numerical experiments are performed using an immersed boundary framework where the viscous, incompressible fluid is governed by either the inertia-free Stokes equations or the full Navier-Stokes equations. In the Stokes case, the anti-phase state is stable and the in-phase state is not. However, when a slight amount of inertia is added, we find that both states are stable to small perturbations. For higher, but still moderate Reynolds numbers we find that the anti-phase state is unstable and all perturbations tend to in-phase oscillations — a dramatic change from zero Reynolds number.

Fluctuation spectra underlie the behaviour of non-equilibrium systems, ALPHA LEE, DOMINIC VELLA, University of Oxford, JOHN WETTLAUFER, Yale University — A diverse set of important physical phenomena, ranging from hydrodynamic turbulence to the collective behaviour of bacteria, are intrinsically far from equilibrium. Despite their ubiquity, there are few general theoretical results that describe these non-equilibrium steady states. Here we argue that a generic signature of non-equilibrium systems is nontrivial fluctuation spectra. Based on this observation, we derive a general relation for the force exerted by a non-equilibrium system on two embedded walls. We find that for a narrow, unimodal spectrum, the force depends solely on the width and the position of the peak in the fluctuation spectrum, and will, in general, oscillate between repulsion and attraction. We demonstrate the generality of our framework by examining two apparently disparate examples. In the first we study the spectrum of wind-water interactions on the ocean surface to reveal force oscillations underlying the Maritime Casimir effect. In the second, we demonstrate quantitative agreement with force generation in recent simulations of active Brownian particles. A key implication of our work is that important non-equilibrium interactions are encoded in the fluctuation spectrum. In this sense the noise becomes the signal.

Dynamics and structure of simple suspensions of active dipoles, TONG GAO, Courant Institute of Mathematical Sciences, MEREDITH BETTERTON, University of Colorado at Boulder, AN-SHENG JHANG, MICHAEL SHELLEY, Courant Institute of Mathematical Sciences — We analyze what is perhaps the simplest active fluid with complex dynamics: a suspension of non-motile, but mobile, “extensor” or “contractor” rods that exert active dipolar stresses on a fluid in which they are immersed. This is relevant to several experimental systems, including growing filaments in isotropic to smectic phase transitions, bundles of cytoskeletal filaments driven by motor proteins, and trimetallic gold-platinum rods immersed in hydrogen peroxide solutions. We first describe the system through a kinetic theory based on microscopic modeling. The stresses produced by particle activity produces long-ranged hydrodynamic coupling, and for extensors can lead to complex time-dependent flows and, depending upon flow geometry, to a form of singularity dynamics through disclination defects production, propagation, and annihilation. We then study useful closures of the kinetic theory, particularly the “Q-tensor” Bingham closure that has been used to study suspensions of passive microscopic rods.

Fluctuations of Bacteria-laden Microbeads in a Liquid, VURAL KARA, CHARLES LISSANDRELLO, Department of Mechanical Engineering, Boston University, JOAN O’CONNOR, Lynn English High School, JOSE ALBERTO RODRIGUEZ RODRIGUEZ, Department of Mechanical Engineering, The University of Texas at El Paso, LE LI, KAMIL EKICI, Department of Mechanical Engineering, Boston University — The motion of bacteria adhered on surfaces may lead to powerful approaches for novel diagnostic tests. Examples were recently shown using microcantilevers on which bacteria were adhered using surface chemistry [1,2]. In these experiments, the presence of bacteria led to an increase in the fluctuations of the microcantilevers in the frequency range 1-100 Hz. After administrating antibiotics, the fluctuations returned to their control value. Here, we build on these studies by monitoring the fluctuations of micro-beads with bacteria adhered on their surfaces. We coat the micro-beads with Poly D Lysine (PDL) in order to attach Escherichia coli. We measure the fluctuations of the beads in motility buffer media using an optical microscope with and without bacteria. We calculate the diffusion coefficients from the mean square displacements (MSD) and correlate these with the presence of bacteria on the beads. These studies lay the foundation for the development of a rapid antibiotic susceptibility test based on bacterial activity.

**References:**


**Sunday, November 22, 2015 2:10PM - 4:20PM**

**Session D24 Biofluids: Biofilms I**

**2:10PM D24.00001** Quasi-chemostat behavior in the leading edge of B. subtilis biofilms, SIDDARTH SRINIVASAN, LAKSHMINARAYANAN MAHADEVAN, SHMUHEL RUBINSTEIN, Harvard University — Bacillus subtilis is a gram positive bacterium that is a model system commonly used to study biofilm formation. By performing wide-field time-lapse microscopy on a fluorescently labeled B. subtilis strain, we observe a well defined steady boundary layer at the edge of a biofilm growing on an nutrient enriched agar gel substrate, within which the outward radial expansion growth predominantly occurs. Using distinct fluorescent protein markers as proxies of gene expression, we quantitatively measure how the width, velocity and ratio of motile cell to matrix cell phenotypes within this boundary layer responds to changes in environmental conditions (such as substrate agar percentage & temperature). We further propose that the steady state at the leading edge can be interpreted as a quasi-chemostat which may enable well controlled response experiments on a colony scale. Finally, we show that for low agar concentration (0.5 wt%), the cells exhibit swarming behavior whose dynamics and swimming velocities are characterized using differential dynamic microscopy. We show the swarming state is associated with an unstable front which gives rise to fingering and branching growth patterns, illustrating the varied morphological response of the biofilm to environmental conditions.

**2:23PM D24.00002** Exploring the mechanisms of rising bubbles in marine biofouling prevention, MARK MENESSES, Boston Univ. JESSE BELDEN, NATASHA DICKENSON, Naval Undersea Warfare Center, JAMES BIRD, Boston Univ — Fluid motion, such as flow past a ship, is known to inhibit the growth of marine biofouling. Bubbles rising along a submerged solid surface where biofouling occurs. The flow structure characteristics were recorded using PIV. This experimental analysis allows us to compare the efficacy of each flow relative to its flow parameters. Exploration of the mechanisms at play in the prevention of biofouling by use of rising bubbles provides a foundation to predict and optimize this antifouling technique under various conditions.

**2:36PM D24.00003** ABSTRACT WITHDRAWN
2:49PM D24.00004 Biofilm formation over surface patterned with pico-liter oil micro-drop array. MARYAM JALALI, JIAN SHENG, Texas Tech University — It has been suggested that bioaccumulation by microalgae is an effective process in the cleansing of oil polluted marine environments. It has also been speculated that dispersants could further enhance processes amid no direct evidence. The studies in the relevant scales are severely hampered by lack of techniques to generate uniform micro-scale drops allowing in-situ monitoring of these processes. In this paper, we present a microfabrication technique allowing patterning microfluidic surfaces with arrays of micro oil drops. The array of oil drops was printed by micro transfer molding/printing with negative PDMS stamplings. The printed micro-drops have dimensions ranging from 5 µm to 50 µm. Non-circular shapes, such as square and triangle, can also be printed and maintained for weeks. Atomic force microscopy is used to characterize the topology and fractal structures of droplets. The results reveal that although the drop with different base shapes assumes dome-like profile asymptotically, donut and top-hat shapes are also observed. Time evolution measurement elucidates in the absence of inviscid mechanisms in comparison to a micro-liter drop, subtle interplays between interfacial forces and viscosity play crucial role in the shape of pico-liter drop. With the developed surfaces, the effects of oil drop sizes and interfacial structures on biofilm formation are studied and reported.

3:02PM D24.00005 Tear film dynamics: modeling the glyocalyx as a porous medium. JAVED SIDDIQUE, Penn State York, ANTONIO MASTROBERARDINO, bSchool of Science, Penn State Erie, RICHARD BRAD MC DANIELS, Department of Ophthalmology and Vision Science, SUNY Upstate Medical University — The human tear film is a complex fluid structure composed of multiple layers: an aqueous layer that comprises most of the film and an outermost thinner lipid layer coat a forest of large transmembrane mucins at the epithelial surface. The glyocalyx helps provide stability to the ocular surface by assisting the tear film to wet it. It is also permeable to water, but less so to ions. We formulate a thin film model based on lubrication theory in order to understand the dynamics between the aqueous layer and the glyocalyx, which we treat as a rigid porous medium. We present numerical solutions for the evolution of the tear film and discuss the roles played by the key parameters of the system.

3:15PM D24.00006 Biomechanical ordering and buckling due to microbial growth confined at oil-water interfaces. GABRIEL JUAREZ, ROMAN STOCKER, ETH Zurich — Bacteria are unicellular organisms that often exist as densely populated, surface-associated communities. They are also environmental colloids and spontaneously attach and self-assemble at liquid-liquid interfaces. Here, we present results on the growth dynamics of individual rod-shaped bacteria confined to finite oil-water interfaces of varying curvatures through experimental and time-lapse microscopy, we study the formation of macroscopic structures observed as adsorbed bacteria grow, divide, and self-assemble in a nematic phase due to biomechanical interactions. The continued growth at the interface leads to a jammed monolayer of cells, which then causes the interface to buckle and undergo large deformations including wrinkling and tubulation. These observations highlight the interplay between physical environment, such as confinement and interface curvature, and active biological processes, such as growth, at the scale of individual agents and shape our understanding of macro-scale processes such as microbial degradation of oil in the ocean.

3:28PM D24.00007 A non-destructive method for characterizing phenotypes and growth of a Bacillus subtilis biofilm using fluorescence microscopy. STEPHAN KOEHLER, SEAS, Harvard, MA, USA, XIAOLING WANG, School of Engineering, Montana State University, USA, DAVE WEITZ, SEAS, Harvard, MA, USA — We develop an imaging technique for characterizing growth of biofilms using a triple fluorescent labeled strain for the three main phenotypes of a Bacillus subtilis biofilm on an agar substrate. We find that the biofilm does not flow across the substrate and thus growth is due to colonization at the periphery and thickening of the interior regions. We obtain local height and its composition of the three main phenotypes, which are motile, matrix-producing and sporulating, as well as the non-fluorescent material, which can be spores, dormant or dead cells or extracellular matrix. This technique is suitable for the study of biofilm growth and inhibition for different conditions such as biocides or bioemediation.

3:41PM D24.00008 PIV measurements of hydrodynamic interactions between biofilms and flow. KENNETH T. CHRISTENSEN, FARZAN KAZEMIFAR, MARCELO AYBAR, PATRICIA PEREZ-CALLEJA, ROBERT NERENBERG, University of Notre Dame, SUMIT SINHA, RICHARD J. HARDY, Durham University, JIM L. BEST, University of Illinois, GREG H. SAMBROOK SMITH, University of Birmingham — Biofilms constitute an important form of bacterial life in aquatic environments and are present at the interface of fluids and solid such as riverbeds or bridge columns. They are also utilized in bioreactors for bioaccumulation and water treatment purposes. They are permeable, heterogeneous, and deformable structures that influence the flow of macroscopic structures observed as adsorbed bacteria grow, divide, and self-assemble in a nematic phase due to biomechanical interactions. The continued growth at the interface leads to a jammed monolayer of cells, which then causes the interface to buckle and undergo large deformations including wrinkling and tubulation. These observations highlight the interplay between physical environment, such as confinement and interface curvature, and active biological processes, such as growth, at the scale of individual agents and shape our understanding of macro-scale processes such as microbial degradation of oil in the ocean.

3:54PM D24.00009 Modeling Tear Film Evaporation and Breakup with Duplex Films. MICHAEL STAFF, RICHARD BRAUN, University of Delaware, CAROLYN BEGLEY, School of Optometry, Indiana University, Bloomington, IN, TOBIN DRISCOLL, University of Delaware, PETER Ewen KING-SMITH, College of Optometry The Ohio State University, Columbus, OH, USA, Tear film thinning, hyperosmolarity, and breakup can irritate and damage the ocular surface. Recent research hypothesizes deficiencies in the lipid layer may cause locally increased evaporation, inducing conditions for breakup. We consider a model for team film evolution incorporating two mobile fluid layers, the aqueous and lipid layers. In addition, we include the effects of salt concentration, osmosis, evaporation as modified by the lipid layer, and the polar portion of the lipid layer. Numerically solving the resulting model, we explore the conditions for tear film breakup and analyze the response of the system to changes in our parameters. Our studies indicate significantly fast peak values or sufficient wide areas of evaporation promote TBU, as does diffusion of an osmotic. In addition, the lefthand eftet representing polar lipids dominates viscous dissipation from the non-polar lipid layer in the model.

This work was supported in part by NSF grant 1412085 and NIH grant 1R01EY021794.

4:07PM D24.00010 We need wrinkle on the skin. SUNGSOOK AHN, SANG JOON LEE, Pohang University of Science and Technology (POSTECH) — Wrinkle formation on the skin is an unwelcome guest to everybody. But if we truly understand how wrinkles can contribute to important biological functions, then we readily admit the wrinkles positively. In this study, we show how the wrinkles are advantageous and useful in many systems. In a plant system for example, by forming line patterned wrinkles the hydrogels covering on the seed surface contribute to delay the dehydration against varying water supply environments. Inspired by this plant hydrogel, it is experimentally and theoretically investigated how lined wrinkles are useful to conserve water inside while protect the individual from drying-out. This study would contribute to a variety of humidity-sensitive system development including artificial skin, humidity-actuated sensors and the like.
2:10PM D25.00001 Characterization of Intracellular Streaming and Traction Forces in Migrating Physarum Plasmodia, SHUN ZHANG, Univ of California - San Diego, OWEN LEWIS, University of Utah, ROBERT GUY, Univ of California - Davis, JUAN CARLOS DEL ALAMO, Univ of California - San Diego — Physarum plasmodium is a model organism for cell migration that exhibits fast intracellular streaming. Single amoebae were seeded and allowed to move on polyacrilamide gels that contained 0.5-micron fluorescent beads. Joint time-lapse sequences of intracellular streaming and gel deformation were acquired respectively in the bright and fluorescent fields under microscope. These images were analyzed using particle image velocimetry (PIV) algorithms, and the traction stresses applied by the amoeba on the surface were computed by solving the elastostatic equation for the gel using the measured bead displacements as boundary conditions. These measurements provide, for the first time, a joint characterization of intracellular mass transport, the forces applied on the substrate and the signal of free intracellular calcium with high resolution in both time and space, enabling a thorough study about the locomotive mechanism, shedding light on related biomimetic research. The results reveal a pronounced auto-oscillation character in intracellular flow, contact area, centroid speed and strain energy, all with the same periodicity about 100 seconds. Locomotion modes that were distinct in flow/ traction stress pattern as well as migration speed have been discovered and studied.

2:23PM D25.00002 Cytoplasmic flows as signatures for the mechanics of mitotic spindle positioning, EHSSAN NAZOCKDAST, ABTIN RAHIMIAN, Courant Institute, NYU, DANIEL NEEDLEMAN, Harvard School of Engineering and Applied Sciences, MICHAEL SHELLEY, Courant Institute, NYU — The proper positioning of the mitotic spindle is crucial for asymmetric cell division and generating cell diversity during development. We use dynamic simulations to study the cytoplasmic flows generated by three possible active forcing mechanisms involved in positioning of the mitotic spindle in the first cell division of C.elegans embryo namely cortical pulling, cortical pushing, and cytoplasmic pulling mechanisms. The numerical platform we have developed for simulating cytoskeletal assemblies is the first to incorporate the interactions between the fibers and other intracellular bodies with the cytoplasmic fluid, while also accounting for their polymerization, and interactions with motor proteins. The hydrodynamic interactions are computed using boundary integral methods in Stokes flow coupled with highly efficient fast summation techniques that reduce the computational cost to scale linearly with the number of fibers and other bodies. We show that although all three force transduction mechanisms predict proper positioning and orientation of the mitotic spindle, each model produces a different signature in its induced cytoplasmic flow and MT conformation. We suggest that cytoplasmic flows and MT conformation can be used to differentiate between these mechanisms.

2:36PM D25.00003 Cytoskeletal Dynamics and Fluid Flow in Drosophila Oocytes, GABRIELE DE CANIO, RAYMOND GÖLDSTEIN, ERIC LAUGA, DAMTP, University of Cambridge — The biological world includes a broad range of phenomena in which transport in a fluid plays a central role. Among these is the fundamental issue of cell polarity arising during development, studied historically using the model organism Drosophila melanogaster. The polarity of the oocyte is known to be induced by the translocation of mRNAs by kinesin motor proteins along a dense microtubule cytoskeleton, a process which also induces cytoplasmic streaming. Recent experimental observations have revealed the remarkable fluid-structure interactions that occur as the streaming flows back-react on the microtubules. In this work we use a combination of theory and simulations to address the interplay between the fluid flow and the configuration of cytoskeletal filaments leading to the directed motion inside the oocyte. We show in particular that the mechanical coupling between the fluid flow and the orientation of the microtubules can lead to a transition to coherent motion within the oocyte, as observed.

2:49PM D25.00004 Mirror-symmetry breakings in human sperm rheotaxis, NORBERT STOOP, Massachusetts Inst of Tech-MIT, ANTON BUKATIN, IGOR KUKHTEVICH, Russian Academy of Sciences & St. Petersburg Academic University, JORN DUNKEL, Massachusetts Inst of Tech-MIT, VASILY KANTSLER, University of Warwick — Rheotaxis, the directed response to fluid velocity gradients, has been shown to facilitate stable upstream-swimming of mammalian sperm cells along solid surfaces, suggesting a robust mechanism for long-distance navigation during fertilization. However, the dynamics by which a human sperm orientates itself w.r.t ambient flows is poorly understood. Here, we combine microfluidic experiments with mathematical modeling and 3D flagellar beat reconstruction to quantify the response of individual sperm cells in time-varying flow fields. Single-cell tracking reveals two kinematically distinct swimming states that entail opposite turning behaviors under flow reversal. We constrain an effective 2D model for the turning dynamics through systematic large-scale scans and multiscale scans, and find good quantitative agreement with experiments. We present comprehensive 3D data demonstrating the rolling dynamics of freely swimming sperm cells around their longitudinal axis. Contrary to current beliefs, this analysis uncovers ambidextrous flagellar waveforms and shows that the cells turning direction is not defined by the rolling direction. Instead, the different rheotactic turning behaviors are linked to a broken mirror-symmetry in the midpiece section, likely arising from a buckling instability.

3:02PM D25.00005 Tumbling and quasi-tumbling motions of E.coli over a solid surface under shear flows, MEHDI MOLAEI, JIAN SHENG, Texas Tech University — Flow shear is known to alter bacterial motility by inducing Jeffery Orbit, rheotaxis, and trapping cells in the high shear region. Over a solid surface flow shear interferes with hydrodynamic interaction of cells with solid surface. Our previous study shows that in the quiescent condition the tumbles of wild E.coli are suppressed and tumble re-orientation of cells is restricted by the solid surface. In the current study, we exposed bacteria to the well controlled shear flows inside a microchannel and applying Digital Holography Microscopy to track them over time. The results show that flow shear promotes tumbling of E.coli and preserve reorientation of the cells during tumbles. Our hydrodynamic model indicates that in the low shear levels the tumble enhancement is due to shear induced flagella unbundling, while in the high shear flow regime, Jeffery Orbit causes rapid cell re-orientation which causes quasi-tumbles with similar angular displacement one would expect during a tumbling.

3:15PM D25.00006 Effect of cell size and shear stress on bacterium growth rate, HADI FADALLAH, Université Paris Diderot, MOJABELA JARRAH, Université Paris Sud, ERIC HERBERT, HASSAN PEERHOSSAINI, Université Paris Cité, JEREMY CLAUS, PEF TEAM — Bacteria growth on the growth rate of Synechocystis and Chlamydomonas cells is studied. An experimental setup was prepared to monitor the growth rate of the microorganisms versus the shear rate inside a clean room, under atmospheric pressure and 20 °C temperature. Digital magnetic agitators are placed inside a closed chamber provided with airflow, under a continuous uniform light intensity over 4 weeks. In order to study the effect of shear stress on the growth rate, different frequencies of agitation are tested, 2 vessels filled with 150 ml of each species were placed on different agitating system at the desired frequency. The growth rate is monitored daily by measuring the optical density and then correlate it to the cellular concentration. The PH was adjusted to 7 in order to maintain the photosynthetic activity. Furthermore, to measure the shear stress distribution, the flow velocity field was measured using PIV. Zones of high and low shear stress were identified. Results show that the growth rate is independent of the shear stress magnitude, mostly for Synechocystis, and with lower independency for Chlamydomonas depending on the cell size for each species.
protocols in biology often vastly underestimate the heterogeneity in biological suspensions arising from incomplete mixing and nonmotile microorganisms by real-time imaging with optical microscopy, testing theoretical predictions and demonstrating that fundamental results date back to the classical works of Purcell (JFM 1978) and Batchelor (JFM 1979), yet were mostly limited to theoretical predictions, which have awaited accurate experimental show how difficult or lengthy it can be to truly mix biological scalars, such a microorganisms. Such results date back to the classical works of virus-host infection. Protocols and approaches in the study of these processes often ignore fundamental principles on stirring and mixing, which 1

bright evidence that bacteria depletes (at low shear rate) or accumulates (at high shear rate) in the vicinity of the wall. We finally show that bacteria are not equally spread over the section. Using different strains of E-coli bacteria and bacteria with different swimming velocity, we

flow in porous media. By counting the number of bacteria as function of the distance from the center of a capillary tube, we show that the bacteria are not equally spread over the section. Using different strains of E-coli bacteria and bacteria with different swimming velocity, we

bacteria (bacillus subtilis) in a time-independent vortex flow. The flow is a pair of vortices generated in a microfluidic cell composed of either a cross or an H-shaped channel. Experiments are done with both wild-type and a genetically-mutated “smooth swimming” bacillus subtilis. We analyze the trajectories of these bacteria in terms of invisible barriers, based on a theory of “burning invariant manifolds” that act as one-way barriers that impede the motion of reaction fronts in a fluid flow. We explore whether similar one-way barriers impede the motion of bacteria.

This work was supported by the NSF Graduate Research Fellowship and University of Minnesota start-up funding

1 LabEx PALM (ANR-10-LABX-0039-PALM)

Laboratory and Field Observations of Microcystis aeruginosa in nearly homogeneous turbulent flows, ANNE WILKINSON, MIKI HONDZO, MICHELE GUALA, University of Minnesota — Microcystis aeruginosa is a single-celled cyanobacterium, forming large colonies on the surface of freshwater ecosystems during summer, and producing a toxin (microcystin) that in high concentration can be harmful to humans and animals. In addition to water temperature, light and nutrient abundance, fluid motion is also an abiotic environmental factor affecting the growth and metabolism of Microcystis. Systematic investigations in a laboratory bioreactor are paired with field measurements in the lacustrine photic zone from two sites in Lake Minnetonka (MN) to ensure that dissipation levels, water temperature, dissolved oxygen and pH are correctly reproduced under laboratory conditions. Laboratory results for biomass accrual and photosynthetic activity suggest that turbulence levels within the range observed in the field, mediates the metabolism, rather than the cell population growth, of Microcystis aeruginosa.

Rheotaxy induced localisation of E-coli in Poiseuille flow, HAROLD AURADOU, MATIAS LOPEZ, Lab FAST, Univ Paris-Sud, CNRS, CARINE DOUARCHE, Laboratoire de Physique des Solides, Univ. Paris-Sud, CNRS, ERIC CLMENT, PMMH (ESPCI /CNRS /Univ. P.M. Curie /Univ. Paris-Diderot), POROUS MEDIA TEAM, GRANULAR MEDIA GROUP TEAM — The transport of bacteria in Poiseuille flow is crucial in many situations in particular those involving flow in porous media. By counting the number of bacteria as function of the distance from the center of a capillary tube, we show that the bacteria are not equally spread over the section. Using different strains of E-coli bacteria and bacteria with different swimming velocity, we bright evidence that bacteria depletes (at low shear rate) or accumulates (at high shear rate) in the vicinity of the wall. We finally show that this phenomena comes from rheotaxy.

1 Supported by NSF Grants DMR-1361881, DUE-1317446 and PHY-1156964.

Imaging of microscale mixing in biological suspensions, KWANGMIN SON, MIT, ROMAN STOCKER, MIT, ETH — In many biological processes, reaction rates are set by the degree of mixing. A prime example is virus-host infection. Protocols and approaches in the study of these processes often ignore fundamental principles on stirring and mixing, which show how difficult or lengthy it can be to truly mix biological scalars, such a microorganisms. Such results date back to the classical works of Purcell (JFM 1978) and Batchelor (JFM 1979), yet were mostly limited to theoretical predictions, which have awaited accurate experimental testing and have not made their way into biological applications to date. Here we investigate the stirring and mixing of suspensions of motile and nonmotile microorganisms by real-time imaging with optical microscopy, testing theoretical predictions and demonstrating that fundamental protocols in biology often vastly underestimate the heterogeneity in biological suspensions arising from incomplete mixing.

Sunday, November 22, 2015 2:10PM - 4:20PM – Session D26 Biofluids: Flexible Swimmers II

Simulations of elastic, stretchable, shearable curves interacting with a liquid, MATTIA GAZZOLA, Harvard University, ANDREW MCCORMICK, Google, L MAHADEVAN, Harvard University — We present a general numerical approach for the simulations of soft filaments deforming in three-dimensional space. Unlike the vast literature on inextensible and unsharable rods, we enable all possible deformation modes at every cross-section consistent with the full Euclidean group SE(3), namely, bending, twisting, shear and stretch. Additionally, we also allow elastic curves to interact with the environment via muscular activity, self-contact, surface friction and hydrodynamics. We demonstrate the capabilities of our approach on a range of biophysical problems, with an emphasis on limbless locomotion on dry surfaces, thin liquid films and in bulk liquids.

From Tadpoles to Trout: Scale-invariant features of optimally efficient swimming, ALEXANDER WIENS, ANETTE HOSOI, Massachusetts Institute of Technology — The Strouhal number ($St$) was thought to be an invariant feature of efficient inertial swimming, however, recent studies by Eloy and Gazzola have shown that $St$ actually varies throughout nature based on animal size, shape, and speed. This finding leads us to ask whether there is any truly invariant property of efficient inertial swimming. Using Lighthill’s large-amplitude elongated-body theory, we show that there is. Lighthill’s model predicts that efficient swimmers must tune their gait such that the unsteady motions of their body generate uniform steady thrust. Mathematically, we show that this behavior can be fully quantified through a single variable which should be constant for all inertial swimmers. Comparison with data from existing literature shows that animals ranging in size from tadpoles to trout adhere to the optimum value predicted by Lighthill’s model.
2:36PM D26.00003 Biofluidynamics of balistiform and gymnotiform locomotion: Revisited1, BRENNAN SPRINKLE, Engineering Sciences and Applied Mathematics, Northwestern University, RAHUL BALE, AMNEET SINGH, NELSON CHEN, Department of Mechanical Engineering, Northwestern University, MALCOM MACIVER, Department of Mechanical Engineering, Department of Biomedical Engineering, Northwestern University, NEELESH PATANKAR, Department of Mechanical Engineering, Engineering Sciences and Applied Mathematics, Northwestern University — Gymnotiform and balistiform swimmers are those which have an undulatory fin affixed to a rigid body unlike anguilliforms who undulate their entire body. Is there a mechanical advantage to gymnotiform and balistiform swimming? This question was investigated by Lighthill & Blake in a four paper series Biofluidynamics of balistiform and gymnotiform locomotion. We revisit this work using fully resolved numerical simulations of the types of swimmers considered by Lighthill & Blake to interrogate the issue of mechanical advantage for rigid body swimmers. In doing so, we find that while there is advantage to rigid body swimming, the mechanism of ‘momentum enhancement,’ proposed by Lighthill and Blake, is not the cause. Further, we use our results and simulations to explain why some gymnotiform and balistiform swimmers have their propulsor attached to their bodies at an angle.

1This work was supported in part by NSF grants CBET-0828749, CMMI-0941674 and CBET-1066575. Computational resources were provided by Northwestern University High Performance Computing System–Quest.

2:49PM D26.00004 A model of a flexible anguilliform swimmer driven by a central pattern generator with proprioceptive feedback . CHRISTINA HAMLET, Tulane University, ERIC TYTELL, Tufts University, KATHLEEN HOFFMAN, University of Maryland, Baltimore County, LISA FAUCI, Tulane University — The swimming of a simple vertebrate, the lamprey, can shed light on how a flexible body can couple with a fluid environment to swim rapidly and efficiently. Animals use proprioceptive sensory information to sense how their bodies are bending, and then adjust the neural signals to their muscles to improve performance. We will present recent progress in the development of a computational model of a lamprey swimming in a Navier–Stokes fluid where a simple central pattern generator model, based on phase oscillators, is coupled to the evolving body dynamics of the swimmer through curvature and curvature derivative feedback. Such feedback can be positive (frequency decreasing), negative (frequency increasing), or mixed (positive to one side of the body and negative to the other, or vice versa). We will examine how the emergent swimming behavior and cost of transport depends upon these functional forms of proprioceptive feedback chosen in the model.

3:02PM D26.00005 Efficient swimmers use bending kinematics to generate low pressure regions for suction-based swimming thrust1, SEAN COLIN, Roger Williams Univ, BRAD GEMMELL, University of Southern Florida, JOHN COSTELLO, Providence College, JENNIFER MORGAN, Marine Biological Laboratory, JOHN DABIRI, Stanford University — A longstanding tenet in the conceptualization of animal swimming is that locomotion occurs by pushing against the surrounding water. Implicit in this perspective is the assumption that swimming involves lateral body accelerations that generate locally elevated pressures in the fluid, in order to achieve the expected downstream push of the surrounding water against the ambient pressure. Here we show that to the contrary, efficient swimming animals primarily pull themselves through the water by creating localized regions of low pressure via waves of body surface rotation that generate vortices. These effects are observed using laser diagnostics applied to normal and spinally-transected lampreys. The results suggest rethinking evolutionary adaptations observed in swimming animals as well as the mechanistic basis for bio-inspired underwater vehicles.

1NSF CBET (1510929)

3:15PM D26.00006 Swimming Performance of Toy Robotic Fish . NINA PETELINA, LEAH MENDELSON, ALEXANDRA TECHET, Massachusetts Institute of Technology — HEXBUG AquaBotsTM are a commercially available small robot fish that come in a variety of "species". These models have varying caudal fin shapes and randomly-varied modes of swimming including forward locomotion, diving, and turning. In this study, we assess the repeatability and performance of the HEXBUG swimming behaviors and discuss the use of these toys to develop experimental techniques and analysis methods to study live fish swimming. In order to determine whether these simple, affordable model fish can be a valid representation for live fish movement, two models, an angelfish and a shark, were studied using 2D Particle Image Velocimetry (PIV) and 3D Synthetic Aperture PIV. In a series of experiments, the robotic fish were either allowed to swim freely or towed in one direction at a constant speed. The resultant measurements of the caudal fin wake are compared to data from previous studies of a real fish and simplified flapping propulsors.

3:28PM D26.00007 A bioinspired aquatic robot propelled by an internal rotor , PHANINDRA TALLAPRAGADA, BEAU POLLARD, Clemson University — Low dimensional models of fish-like swimming of a deformable Joukowski foil shedding singular distributions of vorticity have been well known for two decades. The deformation of the foil can be interpreted to be periodic changes in an abstract shape space and the creation of vorticity can be shown to act as a nonholonomic constraint. With this geometric insight, it can be demonstrated that a Joukowski foil (or in general any body) can possibly swim to the motion of an internal rotor, that acts as a shape variable. The motion of the rotor pumps in angular momentum and the simultaneous creation of vorticity allows this to be ‘converted’ into linear momentum of the foil. We demonstrate the feasibility of this theoretical prediction with a robot shaped as a Joukowski foil propelled by the motion of an internal momentum wheel. We also demonstrate that the internal rotor acts both as a means of propulsion as well as a means of controlling the heading of the robot. Some maneuvers of the robot and features of its physical and ‘mathematical’ resemblance to fish-like motion are demonstrated.

3:41PM D26.00008 CFD Study of Pectoral Fins of Larval Zebrafish: Effect of Reynolds Number and Fin Bending in Fluid Structures and Transport , TOUKIR ISLAM, OSCAR M. CURET, Department of Ocean and Mechanical Engineering, Florida Atlantic University — Zebrafish exhibits significant changes in fin morphology as well as fin actuation during its physical development. In larval stage (Re ~ 10), they beat pectoral fins asymmetrically during slow swimming and prey tracking and a hypothesis suggests pectoral fin motion enhances fluid mixing to assist respiration. We performed a series of computational simulations to study effect of Reynolds number (Re) and pectoral fin kinematics in the fluid dynamics and mixing around a larval zebrafish. The CFD algorithm is based on a constraint formulation where the kinematics of the zebrafish are specified. We simulated experimental zebrafish kinematics at different Re (17 to 300) and considered variations on the fin kinematics to evaluate role of fin deformation in the fluid structures generated by the pectoral fins. Using Lagrangian Coherent Structures and Lagrangian fluid tracers, we identified distinctly dynamic fluid regions and found that mixing around the pectoral fin significantly increases with Re and fin bending enhance fluid mixing at low Re. However, as zebrafish matures and its Re increases, the need to beat the pectoral fins to enhance mixing is reduced.
from the wall. These measurements are challenging because of the high spatial resolution requirement over a relatively large measurement area. This is why MRV's strengths and limitations to address a variety of flow configurations is yet to be demonstrated. Investigated in the present work is the viability of MRV to provide boundary-layer-resolved measurements of a 3D disturbance created by a circular cylindrical element protruding into the shock layer on a flat plate. In comparison to Particle Image Velocimetry, MRV is substantially underutilized, and a modern flow diagnostic technique with unique advantages including the ability to efficiently capture volumetric measurements of velocity fields in complex geometry without the need for optical access. In summary, MRV has been demonstrated to be a promising technique for flow measurements in complex geometries.

Small aquatic arthropods, such as water striders and fishing spiders, are able to jump off water to a height several times their body length. Inspired by the unique biological motility on water, we study a simple model using a flexible hoop to provide fundamental understanding and a mimicking principle of small jumpers on water. Behavior of a hoop on water, which is coated with superhydrophobic particles and initially bent into an ellipse from an equilibrium circular shape, is visualized with a high speed camera upon launching it into air by releasing its initial elastic strain energy. We observe that jumping of our hoops is dominated by the dynamic pressure of water rather than surface tension, and thus it corresponds to the dynamic condition experienced by fishing spiders. We calculate the reaction forces provided by water adopting the unsteady Bernoulli equation as well as the momentum loss into liquid inertia and viscous friction. Our analysis allows us to predict the jumping efficiency of the hoop on water in comparison to that on ground, and to discuss the evolutionary pressure rendering fishing spiders select such dynamic behavior.

Sunday, November 22, 2015 2:10PM - 4:07PM
Session D27 Experiments: Velocity and Vorticity Measurements

2:10PM D27.00001 Boundary-Layer Resolved Measurements of a Three-Dimensional Disturbance Using Magnetic Resonance Velocimetry

Rian Wassermann, Daniel Freudenhammer, Sven Grundmann, TU Darmstadt — Magnetic Resonance Velocimetry (MRV) is a modern flow diagnostic technique with unique advantages including the ability to efficiently capture volumetric measurements of velocity fields in complex geometry without the need for optical access. In comparison to Particle Image Velocimetry, MRV is substantially underutilized, and hence MRV's strengths and limitations to address a variety of flow configurations is yet to be demonstrated. Investigated in the present work is the viability of MRV to provide boundary-layer-resolved measurements of a 3D disturbance created by a circular cylindrical element protruding into the shock layer on a flat plate. These measurements are challenging because of the high spatial resolution requirement over a relatively large measurement area (100 x 100 x 250 mm3), the weak cross-stream disturbance velocities (less than 0.1% of the freestream velocity), and the difficulties associated with the data acquisition. Data are acquired using a portable water-flow loop with an acrylic test section placed on the bed of an MRI machine. The cylindrical element is mounted through the test-section's side wall where the boundary layer Reynolds number (Re) is 162 based on displacement thickness. Several element heights are investigated, ranging from a fraction of, to a full boundary layer thickness. The results provide an assessment of the ability of MRV to perform boundary-layer-resolved measurements of weak disturbances.

2:23PM D27.00002 Multi-plane Particle Shadow Velocimetry to Quantify Integral Length Scales

Jeff Harris, Christine Truong, Steven Hinkle, Kyle Sinding, Pennsylvania State Univ, Tiffany Camp, General Electric, Arnie Fontaine, Michael Krane, David Devilbiss, Pennsylvania State Univ — Multi-plane PIV has been used for several years to assist in quantifying the integral length scales in turbulent flow. Particle shadow velocimetry (PSV) enables illumination of a volume and is an efficient means of obtaining multi-plane illumination. The combination of two colors in the LED backlight and a dichroic mirror makes possible the imaging of two planes in space without the complexity of aligning two different light sources. The velocity fields obtained in these two vector fields are then correlated to obtain length scales using the definitions in the literature. The length scales and multi-plane measurements are compared with previous studies which used proven measurement methods.

2:36PM D27.00003 Advances of Fluid-Structure Interaction Measurements by Multi-Pulse Particle Image Velocimeter/Accelerometer

Liuyang Ding, Ronald Adrian, Arizona State University, Sivaram Gogineni, Spectral Energies, LLC — Multi-pulse particle image velocimeter/accelerometer (PIV/A) is recently developed to improve the performance of conventional PIV and expand the application area. A multi-pulse system consisting of four independent lasers and a high-speed CMOS camera is used for fluid-structure interaction measurements. The test section is an oscillating cylinder (Plexiglas) immersed in refractive-index-matching (RIM) solution and supported by two elastic rods. The unsteady flow field with moving cylinder is imaged at a frequency of 16 sets per cycle, with each set containing four fast consecutive frames. The cylinder motion is tracked by image segmentation technique. Fluid instantaneous and phase averaged velocity and acceleration fields are measured by triple- and quadruple-pulse PIV/A, and their results are compared. Furthermore, the fluid force acting on the cylinder is evaluated using multi-pulse PIV/A data with control volume approach. The rod tip deflection is then calculated by Euler-Bernoulli beam theory and the force measurements, and compared with the ground truth showing good measurement accuracy. The simultaneous velocity, acceleration and force measurements provide a great way for understanding the fluid-structure interaction.

2:49PM D27.00004 Current collector geometry and mixing in liquid metal electrodes

Rakan Ashour, Douglas Kelley, Univ of Rochester — Liquid metal batteries are emerging as an efficient and cost effective technology for large-scale energy storage on electrical grids. In these batteries, critical performance related factors such as the limiting current density and life cycle are strongly influenced by fluid mixing and transport of electrochemical species to and from the electrode-electrolyte interface. In this work, ultrasound velocimetry is used to investigate the role of negative current collector location on the induced velocity, flow pattern, and mixing time in liquid metal electrodes. Ultrasound velocity measurements are obtained at a range of operating current densities. Furthermore, a comparison between velocity profiles produced by current collectors with different sizes is also presented.

This work is supported by ONR under STTR program.
3:02PM D27.00005 Holographic measurement of wall stress distribution and 3D flow over a surface textured by microfibers, HUMBERTO BOCANEGRA, SEDER GORUMLU, BURAK AKSAK, LUCIANO CASTILLO, JIAN SHENG, Texas Tech Univ. — Understanding how fluid flow interacts with micro-textured surfaces is crucial for a broad range of key biological processes and engineering applications including particle dispersion, pathogenic infections, and drag manipulation by surface topology. Existing methods, such as µPIV, suffer from low spatial resolution and fail to track tracer particle motion very close to a rough surface and within roughness elements. In this paper, we present a technique that combines high speed digital holographic microscopy (DHM) with a correlation based de-noising algorithm to overcome the optical interference generated by surface roughness and to capture a large number of 3D particle trajectories. It allows us to obtain a 3D velocity field with an uncertainty of 0.01% and 2D wall shear stress distribution at the resolution of ~ 65 µPa. Applying the technique to a microfluidics with a surface textured by microfibers, we find that the flow is three-dimensional and complex. While the microfibers affect the velocity flow field locally, their presence is felt globally in terms of wall shear stresses. The study of effect of microfiber patterns and flow characteristics on skin frictions are ongoing and will be reported.

3:15PM D27.00006 Three-dimensional flow measurements in a tesla turbine rotor, THOMAS FUHR, CONSTANTIN SCHOSER, RAINER HAIN, CHRISTIAN KAEHLER, Bundeswehr University Munich — Tesla turbines are fluid mechanical devices converting flow energy into rotation energy by two physical effects: friction and adhesion. The advantages of the tesla turbine are its simple and robust design, as well as its scalability, which makes it suitable for custom power supply solutions, and renewable energy applications. To this day, there is a lack of experimental data to validate theoretical studies, and CFD simulations of these turbines. This work presents a comprehensive analysis of the flow through a tesla turbine rotor gap, with a gap height of only 0.5 mm, by means of three-dimensional Particle Tracking Velocimetry (3D-PTV). For laminar flows, the experimental results match the theory very well, since the measured flow profiles show the predicted second order parabolic shape in radial direction and a fourth order behavior in circumferential direction. In addition to these laminar measurements, turbulent flows at higher mass flow rates were investigated.

3:28PM D27.00007 Evaluation of performance of multi-sensors hot-wire probes using Neural-Networks in-situ calibration, DAN LIBERZON, Faculty of Civil and Environmental Engineering, Technion, Haifa, ELIEZER KIT, School of Mechanical Engineering, Tel Aviv University, Tel Aviv — Neural Networks (NN) based in-situ calibration of hot-wire anemometers was recently successfully implemented in field measurements. Although proving feasibility of field measurements using this, relatively new, calibration method the acquired field data also revealed some significant ambiguities in use of combined two- or three-sensor probes. A clearly better behavior of the probe comprised of four sensors (a pair of X shaped probes) has motivated the presented here work, aimed to investigate the NN based procedure performance dependence on the number of wires in the probe. Hypothesizing that the main reason for performance differences is in the fact that a 3-wire probe lacks any special features to withstand the noise in the signal due to temperature fluctuations and sensors’ contamination, series of wind tunnel experiments with grid generated turbulence were designed and performed. Performance of a various multi-sensor probes’ geometries was examined using the NN based method, while standard calibration data sets were also obtained prior to each set of measurements serving as a reference and as alternative training sets for the NN. The obtained results clearly indicated an advantage in using a symmetrical geometry, and especially using the four-sensor probe to obtain a reasonable description of the 3D velocity field. This is argued to be a result of redundant information on one or several velocity components present in four-sensor probes and serving as an efficient tool for noise reduction.

3:41PM D27.00008 A novel time-to-space conversion method bypassing the problems with Taylor’s hypothesis caused by fluctuating convection velocities, CLARA VELTE, Technical University of Denmark, PREBEN BUCHHAVE, Intarsia Optics — A novel conversion of point-measured temporal turbulence power spectra to wavenumber space is proposed. By converting the temporal measurement records into spatial connected streakline elements, the classical assumption of a local mean velocity in Taylor’s hypothesis can be completely bypassed. Laser Doppler velocimetry measurements, which in themselves are particularly suitable for application of this technique, taken at different off-center positions in a round turbulent jet are then used to demonstrate the difference between the current and the classical temporal-to-spatial domain conversions. The novel method displays the same behavior as observed from true spatial spectra measured along homogeneous directions in the very same turbulent axisymmetric jet, while the classical Taylor’s hypothesis, as expected, shows increasing deviation further away from the center axis where the turbulence intensity grows rapidly. Interpretation of first- and second-order statistics including different kinds of spectral estimates are discussed in a related talk by P Buchhave.
1. Denmark U. of Tech., Kgs. Lyngby, Denmark
2. Intarsia Optics, Birkerød, Denmark

3:54PM D27.00009 Turbulence power and kinetic energy spectra measured by a temporal-to-spatial record conversion, PREBEN BUCHHAVE, No Company Provided, CLARA VELTE, Technical University of Denmark — A method of converting a time record of turbulent velocity measured at a point in a flow to a spatial velocity record consisting of consecutive streak line elements that allows computation of turbulent kinetic wavenumber spectra is briefly introduced (more detail in a related paper at this conference, see ref). The method completely bypasses the problems with Taylor’s hypothesis caused by fluctuating convection velocities. In the present contribution, we discuss the interpretation of the first order static moments (e.g. mean and rms velocity) and second order dynamic moments (e.g. spatial correlation function and energy spectrum) computed from the spatial record, which was derived from the measured temporal record. We compare several possible versions of the new energy spectra with the classical 1D and 3D energy spectra and the so-called total kinetic energy spectrum and discuss the range of validity of any equivalence between the new computations and the classical ones. Ref.: Clara M. Velte: A novel time-to-space conversion methods bypassing the problems with Taylor’s hypothesis caused by fluctuating convection velocities

Sunday, November 22, 2015 2:10PM - 4:20PM –
Session D28 Wind Turbines: Blade Design

309 - Fotis Sotiropoulos, University of Minnesota
2:10PM D28.00001 Instability of outer tip vortices for a 2.5 MW wind turbine: integrating snow PIV with LES\(^1\), FOTIS SOTIROPOULOS, XIAOLEI YANG, JIARONG HONG, University of Minnesota, MATTHEW BARONE, Sandia National Laboratories — Recent field experiments conducted around a 2.5 MW wind turbine using super-large-scale PIV (SLPIV) using natural snow particles have revealed tip vortex cores (visualized as areas devoid of snowflakes) of complex shape, consisting of both round and elongated void patterns. Here we employ large-eddy simulation to elucidate the structure and dynamics of the complex tip vortices identified experimentally. The LES is shown to reproduce vortex cores in remarkable agreement with the SLPIV results, essentially capturing all vortex core patterns observed in the field in the tip shear layer. We show that the stretched elongated vortex cores observed in 2D planes are the footprints of a second set of counter-rotating spiral vortices that emanates along the tip shear layer immediately downwind of the blades and is intertwined with the tip vortices. We argue that this large-scale instability is of centrifugal type since the mean flow characteristics in the outer tip shear layer resemble those of the Taylor-Couette flow. This study highlights the feasibility of employing snow voids to visualize tip vortices and demonstrates the enormous potential of integrating SLPIV with LES as a powerful tool for gaining novel insights into the wakes of utility scale wind turbines.

\(^1\)This work was supported by Department of Energy DOE (DE-EE0002980, DE-EE0005482 and DE-AC04-94AL85000), Sandia National Laboratories and NSF Career Award (NSF-CBET-1454259) for Jiarong Hong. Computational resources were provided by SNL and MSI.

2:23PM D28.00002 Experimental Study on the Effects of Winglets on the Wake of a Model Wind Turbine, NICOLAS TOBIN, ALI M. HAMED, LEONARDO P. CHAMORRO, Univ of Illinois - Urbana — Wind tunnel particle image velocimetry was used to investigate the effects of winglets on the wake dynamics of a model wind turbine. The behavior of a turbine with downstream-facing winglets was compared with a turbine without winglets. The turbines were placed in a turbulent boundary layer that reached up to the hub height, allowing for investigation of behavior in both turbulent and uniform flow. The winglets did not significantly change the strength of the tip vortices in the region of uniform incoming flow. The tip vortices in the more turbulent region, however, decayed much faster, diminishing to near-zero within the first ~1.5 rotor diameters, whereas the upper tip vortices persisted potentially up to ~4 rotor diameters. The winglets also increased the power coefficient by 7.5\%.

2:36PM D28.00003 Fluid-structure coupling for wind turbine blade analysis using OpenFOAM\(^1\), BASTIAN DOSE, IVAN HERRAEZ, JOACHIM PEINKE, ForWind - Institute of Physics, University of Oldenburg — Modern wind turbine rotor blades are designed increasingly large and flexible. This structural flexibility represents a problem for the field of Computational Fluid Dynamics (CFD), which is used for accurate load calculations and detailed investigations of rotor aerodynamics. As the blade geometries within CFD simulations are considered stiff, the effect of blade deformation caused by aerodynamic loads cannot be captured by the common CFD approach. Coupling the flow solver with a structural solver can overcome this restriction and enables the investigation of flexible wind turbine blades. For this purpose, a new Finite Element (FE) solver was implemented into the open source CFD code OpenFOAM. Using a beam element formulation based on the Geometrically Exact Beam Theory (GEBT), the structural model can capture geometric non-linearities such as large deformations. Coupled with CFD solvers of the OpenFOAM package, the new framework represents a powerful tool for aerodynamic investigations. In this work, we investigated the aerodynamic performance of a state of the art wind turbine. For different wind speeds, aerodynamic key parameters are evaluated and compared for both, rigid and flexible blade geometries.

\(^1\)The present work is funded within the framework of the joint project Smart Blades (0325601D) by the German Federal Ministry for Economic Affairs and Energy (BMWi) under decision of the German Federal Parliament.

2:49PM D28.00004 Dominant mechanism of load fluctuations on a wind turbine in a realistic atmosphere through Hybrid URANS-LES\(^1\), GANESH VIJAYAKUMAR, ADAM LAVELY, BALAJI JAYARAMAN, BRENT CRAVEN, JAMES BRASSEUR, Pennsylvania State University — Atmospheric turbulence causes load fluctuations on a wind turbine through various forcing mechanisms across a wide span of time scales relative to the rotation time scale. We identify the dominant mechanisms of load fluctuation through blade-boundary-layer-resolved hybrid URANS-LES of a single rotating blade of the NREL-5MW turbine in a daytime moderately convective atmospheric boundary layer (ABL) on flat terrain with surface heating simulated with high-fidelity LES. We find that the integral scale motions in the atmosphere cause the largest fluctuations over multiple rotations of the turbine, while the rotation of the turbine through eddies in the ABL cause fluctuations at the rotation time scale. Blade-boundary-layer dynamicseparation, dynamic stall and rotational augmentation, cause further load fluctuations at time scales much smaller than the rotation time scale. At all time-scales, however, we find that the dominant mechanism underlying load fluctuation on the blade is from local spatio-temporal fluctuations in the angle of attack (AoA) associated with atmospheric eddy passage. By integrating fundamental kinematic analysis with high-resolution CFD, we describe the fundamental role of ABL-turbulence-forced AoA fluctuations on nonsteady wind turbine loadings.

\(^1\)Supported by DOE and College of Engineering, Penn State. Computer resources by XSEDE.

3:02PM D28.00005 An Aeroelastic Perspective of Floating Offshore Wind Turbine Wake Formation and Instability, STEVEN N. RODRIGUEZ, JUSTIN W. JAWORSKI, Lehigh University — The wake formation and wake stability of floating offshore wind turbines are investigated from an aeroelastic perspective. The aeroelastic model is composed of the Sebastian-Lackner free-vortex wake aerodynamic model coupled to the nonlinear Hodges-Dowell beam equations, which are extended to include the effects of blade profile asymmetry, higher-order torsional effects, and kinetic energy components associated with periodic rigid-body motions of floating platforms. Rigid-body platform motions are also assigned to the aerodynamic model as varying inflow conditions to emulate operational rotor-wake interactions. Careful attention is given to the wake formation within operational states where the ratio of inflow velocity to induced velocity is over 50\%. These states are most susceptible to aerodynamic instabilities, and provide a range of states about which a wake stability analysis can be performed. In addition, the stability analysis used for the numerical framework is implemented into a standalone free-vortex wake aerodynamic model. Both aeroelastic and standalone aerodynamic results are compared to evaluate the level of impact that flexible blades have on the wake formation and wake stability.
3:15PM D28.00006 POD based analysis of three-dimensional stall over a pitching wind turbine blade. MATTHEW MELIUS, RAUL BAYOAN CAL, Portland State University, KAREN MULLENERS, COLE POLYTECHNIQUE FDRALE DE LAUSANNE — Aerodynamic performance of a wind turbine blade is a predominant factor in its power production. Under dynamic loading conditions, predicted aerodynamic loads often do not match operational loads. In the interest of gaining understanding of the complex flow over wind turbine blades, a three-dimensional scaled blade model has been designed and manufactured to be dynamically similar to a rotating full-scale NREL 5MW wind turbine blade. Time resolved particle image velocimetry (PIV) measurements collected over the surface section of an inboard section of the experimental turbine blade. Flow characteristics are analyzed using coherent structure identification techniques to capture dynamic stall behavior. Proper orthogonal decomposition (POD) is applied to the velocity field providing information about separation point and stall development time scales based on the associated time coefficients and modes. Additionally, continuity and circulation calculations are used to capture three dimensional effects within stalled volumes during developing stall and re-attachment phases of dynamic stall.

3:28PM D28.00007 Reduced-order FSI simulation of NREL 5 MW wind turbine in atmospheric boundary layer turbulence. JAVIER MOTTA-MENA, ROBERT CAMPBELL, ADAM LAVELY, PANKAJ JHA, Pennsylvania State University — A partitioned fluid-structure interaction (FSI) solver based on an actuator-line method solver and a finite-element modal-dynamic structural solver is used to evaluate the effect of blade deformation in the presence of a day-time, moderately convective atmospheric boundary layer (ABL). The solver components were validated separately and the integrated solver was partially validated against FAST. An overview of the solver is provided in addition to results of the validation study. A finite element model of the NREL 5 MW rotor was developed for use in the present simulations. The effect of blade pitching moment and the inherent bend/twist coupling of the rotor blades are assessed for both uniform inflow and the ABL turbulence cases. The results suggest that blade twisting in response to pitching moment and the bend/twist coupling can have a significant impact on rotor out-of-plane bending moment and power generated for both the uniform inflow and the ABL turbulence cases.

3:41PM D28.00008 Turbulent Potential Model Predictions of High Re Flow Around the S809 Airfoil. NATHANIE DEVELDER, Univ of Mass - Amherst — Utility scale wind turbines operate at a range of chord-based Reynolds numbers often between $10^6$ and $10^7$; Reynolds Averaged Navier-Stokes (RANS) models offer computational efficiency at high Reynolds numbers. As a model that avoids the eddy-viscosity hypothesis, the Turbulent Potential Model, a time-varying RANS model, is identified as an appropriate balance between computational resource usage and physical fidelity. Development of the Turbulent Potential Model is discussed. Comparisons are made between Turbulent Potential Model results and Moser’s Direct Numerical Simulation $Re_c=590$ channel flow. S809 airfoil simulations at $\alpha = 0.02^\circ$, $\alpha = 4.03^\circ$, $\alpha = 10.03^\circ$, and $\alpha = 20.11^\circ$ are compared to results from the $k-\omega_SST$, Spalart-Allmaras, and $\omega^2-f$ models, as well as wind tunnel results from Ohio State University.

3:54PM D28.00009 ABSTRACT WITHDRAWN —

4:07PM D28.00010 Effect of blade loading and rotor speed on the optimal aerodynamic performance of wind turbine blades. CHRISTOPHER BRYSON, FAZLE HUSSAIN, ALAN BARHORST, Texas Tech University — Optimization of wind turbine torque as a function of angle of attack - over the entire speed range from start-up to cut-off - is studied by considering the full trigonometric relations projecting lift and drag to thrust and torque. Since driving force and thrust are geometrically constrained, one cannot be changed without affecting the other. Increasing lift to enhance torque simultaneously increases thrust, which subsequently reduces the inflow angle with respect to the rotor plane via an increased reduction in inflow velocity. Reducing the inflow angle redirects the lift force away from the driving force generating the torque, which may reduce overall torque. Similarly, changes in the tip-speed ratio (TSR) affect the inflow angle and thus the optimal torque. Using the airfoil data from the NREL 5 MW reference turbine, the optimal angle of attack over the operational TSR range (4 to 15) was computed using a BEM model to incorporate the dynamic coupling, namely the interdependency of blade loading and inflow angle. The optimal angle of attack is close to minimum drag during start-up phase (high TSR) and continuously increases toward maximum lift at high wind speeds (low TSR).

Sunday, November 22, 2015 2:10PM - 4:20PM –

Session D29 Nonlinear Dynamics: Coherent Structures II 310 - Greg Chini, University of New Hampshire

2:10PM D29.00001 Asymptotic descriptions of self-sustaining processes in shear and Langmuir turbulence: A comparative study. GREG CHINI, ZHEXUAN ZHANG, University of New Hampshire, CEDRIC BEAUME, Imperial College London, EDGAR KNOBLOCH, University of California Berkeley, KEITH JULIEN, University of Colorado Boulder — It has long been observed that stress-driven turbulence either in the presence or the absence of surface waves is characterized by streamwise-oriented roll vortices and streamwise streaks associated with spanwise variations in the streamwise flow. To elucidate the fundamental differences between wave-free (“shear”) and wave-catalyzed (“Langmuir”) turbulence, two separate asymptotic theories are developed in parallel. First, a large Reynolds number analysis of the Navier–Stokes equations that describes a self-sustaining process (SSP) operative in linearly stable wall-bounded shear flows is recounted. This theory is contrasted with that emerging from an asymptotic reduction in the strong wave-forcing limit of the Craik–Leibovich (CL) equations governing Langmuir turbulence. The comparative analysis reveals important structural and dynamical differences between the SSPs in shear flows with and without surface waves and lends further support to the view that Langmuir turbulence in the upper ocean is a distinct turbulence regime.

GPC gratefully acknowledges funding from NSF CBET Award 1437851.
have been shown previously to act as one-way barriers for reaction fronts propagating in two-dimensional fluid flows.

We focus on the issue of the chemo-hydrodynamic coupling between forced advection opposed to self-sustained chemical waves which can lead to the range over which we do observe these Frozen Fronts. We compare the shape of the observed Frozen Fronts to the computed ones in the so-called eikonal, thin front limit. In this limit, we are able to provide a scenario for the selection of the observed frozen states.

The fronts are observed to encounter tube- and sheet-like barriers, produced by the excitable, Ruthenium-catalyzed Belousov-Zhabotinsky chemical reaction. When illuminated with a near-UV laser beam, the chain of nested horizontal and vertical vortices, a flow that has been shown to produce chaotic mixing even if time-independent, is visible. The numerical simulations, however, use a strictly 2D model which incorporates the effects of the finite thickness of the fluid layer in the experiment. During its evolution, there are instances when the dynamics of a weakly turbulent flow slow down, rather dramatically. Using experimental flow fields from such instances, and by means of a Newton-Solver, we numerically compute several unstable equilibria. Additionally, using numerical simulations, we show that the dynamics of a turbulent flow in the neighbourhood of an equilibrium are accurately described by the unstable manifold of the equilibrium.

This work is supported in part by the National Science Foundation under grants CBET-0900018, and CMMI-1234436.

Supported by NSF CAREER DMS-1255422, NSF GRFP.

We present experiments that study the behavior of reaction fronts propagating in three-dimensional, laminar fluid flows. The primary flow is a chain of nested horizontal and vertical vortices, a flow that has been shown to produce chaotic mixing even if time-independent. The fronts are produced by the excitable, Ruthenium-catalyzed Belousov-Zhabotinsky chemical reaction. When illuminated with a near-UV laser beam, the Ru indicator fluoresces everywhere except where there is a reaction front. By scanning the laser beam and imaging from above, we are able to do a full 3D-visualization of the reaction front propagating through the flow. The fronts are observed to encounter tube- and sheet-like barriers, whose properties we measure experimentally. We interpret the results by generalizing a recent theory of "burning invariant manifolds" which have been shown previously to act as one-way barriers for reaction fronts propagating in two-dimensional fluid flows.
3:28PM D29.00007 Experimental studies of pinned and unpinned reaction fronts in two-dimensional, vortex-dominated flows. LAURA SKINNER, JOSEPH-JOHN SIMONS, TOM SOLOMON, Bucknell University — We present experiments that study the propagation and pinning of reaction fronts in laminar, two-dimensional fluid flows. The flows are forced using magnetohydrodynamic techniques and are composed of vortex chains and arrays with or without an imposed wind. The reaction fronts are produced by the excitable, ferroin-catalyzed Belousov-Zhabotinsky chemical reaction. We consider how the addition of time-periodic oscillations of the flow can affect the pinning of reaction fronts. Furthermore, we measure the speed at which reaction fronts propagate in the flow, looking for scaling of the measured front propagation speed with the non-dimensional reaction-diffusion (no flow) speed. We analyze all of these results by considering the role of one-way barriers produced by “burning invariant manifolds.”

1Supported by NSF Grants DMR-1361881, DUE-1317446 and PHY-1156964.
2Current address: Point Loma Nazarene University, San Diego, CA

3:41PM D29.00008 Vertically localized equilibrium solutions in the large eddy simulation of homogeneous shear flow. ATSUSHI SEKIMOTO, JAVIER JIMENEZ, Univ Politecnica de Madrid — Equilibrium solutions in a large eddy simulation (LES) of statistically-stationary homogeneous shear flow with zero molecular viscosity are numerically obtained by a Newton-Krylov-hookstep method. The energy input is done by the mean shear at scales comparable to the spanwise width $L_z$ of the computational domain, while energy dissipation is represented by the eddy viscosity term at the small scale of the order of the Smagorinsky length $C_S\Delta$ ($C_S$ is the static Smagorinsky constant and $\Delta$ is the grid scale). It is shown that these solutions appear by a saddle-node bifurcation as $C_S\Delta / L_z$ decreases, and have the sinuous symmetry of Nagata’s equilibrium solution in Couette flow (JFM 217, 519-527 (1990)). Both lower- and upper-branch solutions are vertically localized. The upper-bound solution is characterized by tall structures, while the lower-bound forms in the critical layer as in the asymptotic theory of shear flows at high-Reynolds numbers (K. Deguchi & P. Hall, Phil. Trans. R. Soc A, 372:20130352 (2014)).

Funded by the ERC Multiflow project.

3:54PM D29.00009 Exact localized free-stream coherent structures in a parallel boundary layer. TOBIAS SCHNEIDER, EPFL - Lausanne, JOHN GIBSON, University of New Hampshire, TOBIAS KREILOS, EPFL - Lausanne — The dynamical systems description of transitional turbulence is based on exact invariant solutions of the 3D Navier-Stokes equations. We present a new family of exact traveling wave solutions in the asymptotic suction boundary layer. The solutions are localized in wall-normal and spanwise direction. Instead of being attached to the wall, the solutions are dominated by vortical structures reaching far into the free-stream region. These invariant solutions thus suggest that dynamical systems concepts, so far mostly studied in confined geometries, can carry over to open boundary layers and are relevant for turbulence far from confining walls.

4:07PM D29.00010 Exact laminar solutions for flows in channels with sinusoidal walls. SABARISH VADAREVU, ATI SHARMA, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton — We compute exact solutions for steady, incompressible, laminar flows in sinusoidal channels using Newton’s method, employing domain transformation with spectral resolution in all spatial directions. Aligning the walls to be in phase has made computations considerably cheap (runtime/case ~ 10 minutes on 1 core); Newton’s method has allowed tracing solutions into high Reynolds number ranges, where solutions are temporally unstable. We identify four parameters: the amplitude, maximum slope, and streamwise inclination of the grooves/furrows in the surfaces, as well as the mean pressure gradient that drives the flow. Results are presented for amplitudes ranging from 0.1% to 10% of channel half-height, and maximum slopes ranging from 0.3 to 3.0, for a set of inclinations and Reynolds numbers. We look at the onset and sizes of steady recirculation zones, their effect on the volume flux, and relative contributions of pressure and wall-shear to total drag. The strengths of shear layers and the wall-normal gradients of circulation are considered as indicators for Kelvin-Helmholtz and centrifugal instabilities respectively. Future work will focus on computing other classes of exact solutions and understanding their significance to transition and turbulence.

Sunday, November 22, 2015 2:10PM - 4:20PM —
Session D30 DFD: Geophysical Fluid Dynamics: Wakes and Boundaries in Stratified Flows 311 - James Riley, University of Washington

2:10PM D30.00001 Characteristics of turbulent/non-turbulent interfaces in wakes in stably-stratified fluids. TOMOAKI WATANABE, JAMES RILEY, University of Washington, STEPHEN DE BRUYN KOPS, University of Massachusetts, PETER DIAMESSIS, QI ZHOU, Cornell University — The evolution of turbulent patches generated by the wake of a sphere in stably-stratified fluids is studied using direct numerical simulations. The DNS data analysis focuses on the investigation of the characteristics of turbulent/non-turbulence interfaces forming at the wake edge. Unlike the case for non-stratified fluids, because of the non-negligible enstrophy level in internal waves outside the stratified wake, enstrophy cannot be used as a marker for turbulent regions. We show that potential enstrophy is appropriate as a marker for turbulent regions in flows where both turbulence and internal waves exist. Therefore the interface is detected as an isosurface of constant potential enstrophy, and statistics can be calculated conditioned on the distance from the interface. Various quantities are examined from the wake interior to the region outside the wake, and show how the flow properties are adjusted between turbulent and non-turbulent regions near the interface. Based on the conditional analysis, we also report evidence for the strong influence of internal waves on turbulence inside the wake.

ONR Grants N00014-13-1-0665 and N00014-12-1-0583; HPCMP Frontier Project FPCFD-FY14-007; JSPS KAKENHI No. 25002531
2:23PM D30.00002 Dynamic mode decomposition identifies internal wave and vertical modes in stably stratified wakes

XINJIANG XIANG, KEVIN CHEN, TRYSTAN MADISON, GEOFFREY SPEDDING, University of Southern California — Though detailed information has been assembled to describe the late wakes behind various objects in stably stratified fluids, less is known about the dynamics at early stages, when the flow first interacts with the ambient density gradient, beginning the transition to the late wake regime. Detailed velocity fields (and derivatives) were reported by Xiang et al. (J. Fluid Mech. 775, 149-177, 2015) for the near wake of a towed grid, with $Re \in \{2700, 11000\}$ and $Fr \in \{0.6, 9.1\}$. Here using dynamic mode decomposition (DMD), the spatial and temporal evolution of the lee wave and shearing modes are extracted and examined for the same data set. Both dynamic modes show systematic dependence on $Fr$ and $Re$, consistent with previous analysis. The results show the potential of DMD in analyzing the contribution of different modes in a complex, near wake evolution, including, but not limited to towed grids, and the wakes of more complicated towed geometries.

Support from ONR N00014-11-1-0553 is most gratefully acknowledged.

2:36PM D30.00003 ABSTRACT WITHDRAWN

2:49PM D30.00004 Stratified Shear Flows In Pipe Geometries

GEORGE HARABIN, ROBERTO CAMASSA, RICHARD MCLAUGHLIN, University of North Carolina at Chapel Hill, UNC JOINT FLUIDS LAB TEAM — Exact and series solutions to the full Navier-Stokes equations coupled to the advection diffusion equation are investigated in tilted three-dimensional pipe geometries. Analytic techniques for studying the three-dimensional problem provide a means for tackling interesting questions such as the optimal domain for mass transport, and provide new avenues for experimental investigation of diffusion driven flows. Both static and time dependent solutions will be discussed.

NSF RTG DMS-0943851, NSF RTG ARC-1025523, NSF DMS-1009750

3:02PM D30.00005 Density overturns and local stability measures in early stratified wakes

TRYSTAN MADISON, PRABU SELLAPPAN, XINJIANG XIANG, GEOFFREY SPEDDING, University of Southern California — Though the dynamics of decaying stratified turbulence are sensitive to certain details of the initial generating conditions, the late-time evolution has also general characteristics that depend only on local stratification parameters, often characterised by a buoyancy Reynolds number, $Re_B \approx Re Fr^2$. Bluff-body wakes, for example, have been shown to have universal characteristics that do not depend on details of the generator. Recent experiments on the near wake of a towed grid (Xiang et al. J. Fluid Mech. 775, 149-177, 2015) show that the trajectory of solutions entering the late stratified turbulence regime vary significantly with both $Re$ and $Fr$, reflecting different balance between wake-edge shear instabilities and local, grid turbulence-generated motions. Here we show density profiles taken through the grid wakes for $Re = 2700$ and $Fr = \{2, 4, 9\}$. The relative importance of stabilising density gradients vs. destabilising shear flows is customarily measured by a global and/or local Richardson number, $Ri$, and such measures will be compared and contrasted to form a more complete and quantitative picture of the early wake instabilities than has been available hitherto.

Support from ONR N00014-11-1-0553 is most gratefully acknowledged.

3:15PM D30.00006 Dynamics of particle laden plume in linearly stratified environment

HARISH MIRAJKAR, SRIDHAR BALASUBRAMANIAN, Department of Mechanical Engineering, Indian Institute of Technology Bombay, India — Particle laden plumes, which are common in geophysical flows, were simulated experimentally and their flow dynamics was studied. Particles having mean size, $d_p = 100\mu m$, density, $\rho_p = 2500$ kg m$^{-3}$, and volume fraction, $\phi_v = 0-0.7\%$, were injected along with lighter buoyant fluid (N = 0.05, $\beta_f = 0.65\%$, $\beta_\phi = 0.5\%$). It was observed that a particle-laden plume intruding at the neutral layer is characterized by four spreading regimes: (i) radial momentum flux balanced by the inertia force; (ii) inertia buoyancy regime; (iii) fluid-particle inertia regime, and (iv) viscous buoyancy regime. The maximum height, $Z_m$ for $\phi_v > 0\%$ was observed to be consistently lower than the single-phase case. In the inertia-buoyancy regime, the radial spread, $R_f$, for the particle laden plume advanced in time as $R_f \approx t^0.68$ which is slower compared to the single-phase plume that propagates at $R_f \approx t^{0.74}$. It was observed that the jet cone angle is higher for the case of particle-laden plume owing to flaring of the plume. Due to the presence of particles, ‘particle fall out’ effect occurs forming a parabolic cloud below the plume spreading height. With increasing $\phi_v$, secondary umbrella formation was also observed.

3:28PM D30.00007 Interaction of two spheres settling in a linearly stratified fluid

MATTHIEU MERCIER, CLEMENT TOUPOINT, PATRICIA ERN, IMFT (Univ Toulouse III-INPT-UPS-CNRS) — The settling dynamics of small objects in stratified fluids is important to understand the fate of the biomass in lakes or oceanic environments, for industrial applications such as waste-water disposal. More specifically, the interaction of two settling bodies is a fundamental problem recently studied numerically for spheres. Experimental results are needed for validation, especially at low and moderate values of the Reynolds number, for different values of the Froude number, the other parameter of interest. We present experimental results on the interaction of two spheres settling in a linearly stratified fluid. The settling dynamics is investigated by tracking their trajectories in three dimensions, using a pair of cameras imaging two perpendicular planes. Two typical cases are observed, the horizontal repulsion of particles initially aligned horizontally, and the Drafting-Kissing-Tumbling of spheres initially aligned vertically. The influence of the initial positions of the spheres, the Reynolds and Froude numbers, is investigated to quantify these effects and their robustness, in comparison to the dynamics in an homogeneous fluid.

3:41PM D30.00008 Internal hydraulic jumps with large upstream shear

KELLY OGDEN, Massachusetts Institute of Technology / Woods Hole Oceanographic Institution, KARL HELFRICH, Woods Hole Oceanographic Institution — Internal hydraulic jumps in approximately two-layered flows with large upstream shear are investigated using numerical simulations. The simulations allow continuous density and velocity profiles, and a jump is forced to develop by downstream topography, similar to the experiments conducted by Wilkinson and Wood (1971). High shear jumps are found to exhibit significantly more entrainment than low shear jumps. Furthermore, the downstream structure of the flow has an important effect on the jump properties. Jumps with a slow upper (inactive) layer exhibit a velocity minimum downstream of the jump, resulting in a sub-critical downstream state, while flows with the same upstream vertical shear and a larger barotropic velocity remain super-critical downstream of the jump. A two-layer theory is modified to account for the vertical structure of the downstream density and velocity profiles and entrainment is allowed through a modification of the approach of Holland et al. (2002). The resulting theory can be matched reasonably well with the numerical simulations. However, the results are very sensitive to how the downstream vertical profiles of velocity and density are incorporated into the layered model, highlighting the difficulty of the two layer approximation when the shear is large.
Unlike the case of a solid substrate, no chimney instability was found in the range of drop size studied. A circular neck. The influences of the superheat and of the drop size are parametrically investigated. A number of scaling laws are established.

...the shape of the drop. As in the case of a solid substrate, the vapor layer generally appears to be composed of a vapor pocket surrounded by... its complete evaporation if the temperature is not high enough. This may be due to local cooling of the substrate under the drop. We also investigate the evaporation of these drops, and find scalings... to perform computations in these transient regimes, therefore, we propose in this paper a novel numerical method to achieve this challenging task. Finally, we present several accurate validations against experimental results on Leidenfrost Droplets to strengthen the relevance of this new method.

Sunday, November 22, 2015 2:10PM - 4:20PM –
Session D31 Drops: Leidenfrost Effects 312 - Guillaume Riboux, Escuela Superior de Ingenieros, Universidad de Sevilla, Spain

Direct numerical simulation (DNS) has been performed to investigate internal gravity waves in flow past a sphere at $Re = 3700$. In this case, boiling occurs in the film of saturated vapor which is entrapped between the bottom of the drop and the plate, whereas the top of the water droplet evaporates in contact of ambient air. Thus, boiling and evaporation can occur simultaneously on different regions of the same liquid interface or occur successively at different times of the history of an evaporating droplet. Usual numerical methods are not able to perform computations in these transient regimes, therefore, we propose in this paper a novel numerical method to achieve this challenging task. Finally, we present several accurate validations against experimental results on Leidenfrost Droplets to strengthen the relevance of this new method.

...the body generation mechanism become stronger and waves exhibit upstream propagation. In the downstream direction, there is a very clear distinction in the temporal and spatial structure between waves generated by the mean and those by the turbulent flow. Turbulence leads to waves with high frequency modes that propagate into the background and interact with the low frequency lee waves. Quantitative analysis of the potential energy (PE) distribution as a function of Fre and downstream distance has also been carried out. For Fr = 1, a significant amount of the energy produced by the displacement of the fluid by the body is converted into PE of the lee waves. In contrast, most of the input energy goes into the turbulent wake for Fr = 3. The Fr = 1 case manifests a substantial decrease in the recirculation length and an increase in the wave drag as compared to Fr = 3.

Sponsored by ONR Turbulence and Wakes program.
2:49PM D31.00004 Vapor layer evolution during drop impact on a heated surface . SANGHYEON LEE, SANGJUN LEE, JISAN LEE, X-ray Imaging Center, Department of Materials Science and Engineering, POSTECH, South Korea, KAMEL FEZZAA, X-ray Science Division, Advanced Photon Source, Argonne National Laboratory,USA, JUNG HO JE, X-ray Imaging Center, Department of Materials Science and Engineering, POSTECH, South Korea — When a liquid drop impacts on a sufficiently hot surface above the boiling point, a vapor layer is formed between the drop and the surface, preventing direct contact between them and as a result levitating the drop. We study the evolution of the vapor layer under various impact conditions. Using X-ray phase contrast imaging, we directly visualize the profiles of the vapor layers during liquid drop impact on a hot surface and elucidate the evolution of the vapor layers during spreading and retraction of the drop as functions of impact height and surface temperature. We reveal that the evolution is governed by the propagation of capillary waves generated in retraction and the wavelength of capillary waves is inversely proportional to the impact height with a relation $\propto \frac{h}{\sigma} \propto We^{-1}$ where We is Weber number. Capillary waves that converge at the center of the vapor layers are linked to the bouncing behavior of the drop.

3:02PM D31.00005 A Leidenfrost Engine . GARY WELLS, RVIDRIO LEDESMA-AgüILLAR, GLEN MCHALE, Northumbria University, Newcastle upon Tyne NE1 8ST, UK, KHELLIL SEFIANE, University of Edinburgh, Scotland EH9 3FB. — The Leidenfrost effect, the sustained levitation of evaporating liquid droplets by a cushion of their own vapour on very hot surfaces, has received increased attention over the past few years. On patterned surfaces, rectification of the vapour layer flow can lead to rich dynamics of evaporating drops or sublimating blocks of dry ice, including self-propulsion, orbiting and conjoint rotation. In this paper we show that the Leidenfrost effect can be exploited to drive the rotation of rigid objects, such as solid hydrophilic plates coupled to water droplets and blocks of dry ice, by using turbine-like substrates. Using a hydrodynamic model, we show that drag-based rotation is achieved at low-Reynolds number by a rectification mechanism of the flow in the vapour layer caused by the underlying turbine-like geometry. Our theoretical model describes the maximum weight of Leidenfrost rotors and the net torque driving their motion in terms of operational parameters, showing an excellent agreement with experiments using dry-ice blocks. We generalise the concept of rotation into a new concept for a heat engine capable of harvesting thermal energy using either thin-film boiling or sublimation as a phase-change mechanism. As a proof of principle, we implement the new sublimation engine in the lab to create a simple electromagnetic generator. Our results support the feasibility of low-friction in situ energy harvesting from both liquids and ices in challenging situations such as deep drilling, outer space exploration or micro-mechanical manipulation.

3:15PM D31.00006 Maximum drop radius and critical Weber number for splashing in the dynamical Leidenfrost regime . GUILLAUME RIBOUX, JOSE MANUEL GORDILLO, Escuela Superior de Ingenieros, Universidad de Sevilla, Spain — At room temperature, when a drop impacts against a smooth solid surface at a velocity above the so called critical velocity for splashing, the drop loses its integrity and fragments into tiny droplets violently ejected radially outwards. Below this critical velocity, the drop simply spreads over the substrate. Splashing is also reported to occur for solid substrate temperatures above the Leidenfrost temperature, for which a vapor layer prevents the drop from touching the substrate. In this case, the splashing morphology largely differs from the one reported at room temperature because, thanks to the presence of the gas layer, the shear stresses on the liquid do not decelerate the ejected lamella. Our purpose here is to predict, for wall temperatures above $T$, the dependence of the critical impact velocity on the substrate temperature of the substrate as well as the maximum spreading radius for impacting velocities below the critical velocity for splashing. This is done making use of boundary integral simulations, where the velocity and the height of the liquid layer at the root of the ejected lamella are calculated numerically. This information constitutes the initial conditions for the one dimensional mass and momentum equations governing the dynamics of the toroidal rim limiting the edge of the lamella.

3:28PM D31.00007 Relevant time- and length scale of touch-down for drops impacting on a heated surface . MICHEL A.J. VAN LIMBEEK, MINORI SHIROTA, University of Twente - Netherlands, CHAO SUN, University of Twente - Netherlands; Tsinghua University - China, ANDREA PROSPERETTI, University of Twente - Netherlands; Johns Hopkins University - USA, DETLEF LOHSE, University of Twente - Netherlands — The vapor generated from a liquid drop impacting a hot solid surface can prevent drop impact. Depending on the solid temperature, the minimum temperature when no contact is made between the drop and the solid is called the dynamical Leidenfrost temperature. The latent heat needed to generated the vapor is drawn from the solid, and in general the Leidenfrost temperature depends on the solid thermal properties. Here we show experiments conducted on a sapphire plate, to minimize the cooling of the solid and ensuring nearly isothermal conditions. By using high speed total internal reflection imaging, we observe the drop base during impact up to about 100nm above the substrate surface. By this technique we are able to study the processes responsible for the transition between fully wetting and fully levitating drop impact conditions as the solid temperature increases. We reveal the relevant length- and time-scales for the dimple formation under the drop and explain their relevance for the late-time dynamics. As the transition regime is traversed from low to high temperature, the liquid-solid contact gradually decreases which reduces the friction with the solid, enhancing the spreading of the drop considerably.

3:41PM D31.00008 Pool impacts of Leidenfrost drop . BAPTISTE DARBOIS TEXIER, Grasp, ULg, Liege, Belgium, ELINE DEHANDSCHOEWERCKER, Ecole Polytechnique, France, ZHAO PAN, Splash lab, Brigham Young University, TODD LIEGE, Liege University, Belgium — This work concerns the impact of a droplet made of a volatile liquid (typically HFE) on a pool of an other liquid (typically silicone oil) which temperature is above the boiling point of the drop. Depending on the properties of the two liquids and the impacting conditions, four different regimes are observed. For low impacting speeds, the droplet bounces on the surface of the bath and finally levitates above it in a Leidenfrost state. Such a regime occurs as soon as the pool temperature exceeds the boiling point of the drop. This observation means that there is no threshold in temperature for a Leidenfrost effect on a liquid surface contrary to the case of a solid substrate. For intermediate impacting velocities, the pinch-off of the surface of the pool entraps the drop in the liquid bulk. The entrapped drop is separated from the pool by a layer of its own vapour in a similar way of antibubbles. For increasing impacting speeds, the vapour layer between the drop and the pool does not hold during the pinch-off event. The contact of the drop with the hot liquid provokes a sudden and intense evaporation. At very large impacting speeds, the drop rapidly contacts the pool, spreads and finally induces a hemi-spherical cavity. In the end, these four different regimes are summarized in a Froud-Weber diagram which boundaries are discussed.

3:54PM D31.00009 “Cold” Leidenfrost effect . PHILIPPE BOURRIANNE, CHRISTOPHE CLANET, DAVID QUERE, PMMH, ESPCI / LaDHyX, Ecole Polytechnique — An evaporating Leidenfrost drop placed on a hot substrate can levitate on its own vapor if the temperature of the substrate is high enough. We discuss the possibility to decrease this critical Leidenfrost temperature using a super-hydrophobic coating. Measuring adhesion and observing the liquid-solid interface, we suggest a possible explanation for this “cold” regime of levitation.
Droplet fragmentation on leaves shapes foliar disease dispersal

LYDIA BOUROUIBA, Massachusetts Institute of Technology, TRISTAN GILET, Liege University — Although the dispersal of pathogens from plant to plant remains poorly understood, a strong statistical correlation exists between rainfall patterns and plant disease outbreaks. This correlation suggests that rain is a culprit in the dispersal of foliar pathogens. In this combined experimental and theoretical study, we unravel the mechanisms at play when a raindrop impacts an infected plant leaf. We identify two main fragmentation processes that shape rain-induced dispersal mechanisms. In both, pathogens are initially contained in water residues left on leaves by previous raindrops. As most leaves are partially wetting, residues take the shape of sessile drops. The impact of another raindrop in the vicinity triggers fragmentation of the sessile drop and subsequent ejection of contaminated droplets towards neighboring plants. Each scenario yields a different distribution of ejected droplets and brings a distinct contribution to the epidemic onset pattern. We show that leaf mechanical properties govern both fragmentation scenarios. Dimensionless parameters and scaling laws are provided to rationalize our observations.

The break-up of a viscous liquid droplet in a high Reynolds number shear flow

CHIN HEI NG, ALBERTO ALISEDA, University of Washington — The break-up of a viscous liquid droplet in a sheared turbulent flow evolves in several steps, the most visually dominant of which is the formation of high aspect ratio ligaments. This feature takes them apart from the various break-up models based on the Hinze-Kolmogorov paradigm of eddy-spherical particle collisions. We investigate the development of ligaments in a high Reynolds number (up to 250,000) submerged round jet, within the high viscosity, near-unity density ratio regime. Unlike in H-K theory, applicable to the break-up of inviscid fluid particles, break-up of inertial-scale viscous droplets occurs through a sequence of eddy collisions and long-term deformation, as evidenced by measurements of the aspect ratio that fluctuates and increases progressively during the deformation stage, and results in non-binary break-up. Additionally, the ligament formation stretches a droplet multiple times its original size, bringing the influence of integral-scale structures. High speed imaging has been statistically analyzed to inform and validate theoretical models for the break-up time and the break-up probability. In addition, a particle size scaling model has been developed and compared with the experimental measurements of the frozen-state particle size.

Droplet deformation and breakup in flows with and without shear

TIMEA KÉKESI, GUSTAV AMBERG, LISA PRAHL WITTBERG, KTH — The deformation and breakup of liquid drops in gaseous flows are studied numerically using the Volume of Fluid method. Fragmentation of fuel drops has a key role in combustion, determining the rate of mixing and the efficiency of the process. It is common to refer to Weber number 12 as the onset of breakup, and to define breakup mode regimes as a function of Weber number. These definitions are established for simple flows and do not take density and viscosity ratios into account. The main objective of this work is the dynamics of the drop leading to breakup. Fully developed uniform flows and flows with various shear rates are considered. A Weber number of 20, Reynolds numbers 20-200, density ratios 20-80, and viscosity ratios 0.5-50 were used. Results for uniform flows are presented in Kékesi T., et al. (2014). The final aim of the project is to extend existing atomization models for fuel sprays by accounting for density and viscosity ratios in addition to the Reynolds and Weber numbers already present in current models. Estimations for the lifetime of the drop are provided; furthermore, the history of the drag coefficient is compared for several cases. Examples of the observed phenomena and ideas for possible model modifications will be presented.

Study on Electric field assisted low frequency (20 kHz) ultrasonic spray

ILKYEONG CHAE, BAEKHOON SEONG, DARMAWAN MARTEN, DOYOUNG BYUN, Sungkyunkwan Univ. — Ultrasonic spray is one of the fabulous techniques to discharge small size of droplets because it utilizes ultrasonic vibration on nozzle. However, spray patterns and size of ejected droplet is hardly controlled in conventional ultrasonic spray method. Therefore, here we present electric field assisted ultrasonic spray, which combined conventional technique with electric field in order to control spray pattern and droplet size precisely. Six kinds of various liquid (D.I water, Ethanol, Acetone, Iso-propanol, Toluene, Hexane) with various dielectric constants were used to investigate the mechanism of this method. Also, PIV (Particle Image Velocimetry) was used and various variables were obtained including spray angle, amplitude of liquid vibration, current, and size distribution of ejected droplets. Our electric field assisted ultrasonic spray show that the standard deviation of atomized droplet was decreased up to 39.6%, and it shows the infinite possibility to be utilized in various applications which require precise control of high transfer efficiency.

Does buoyancy matter in the melting dynamics of ice?

JICHEONG GUO, MUSTAFA ORDU, SOUMENDRA BASU, JAMES BIRD, Boston University — Ice in a horizontal cylindrical container will melt when placed in a sufficient warm environment. Because of the density difference between the ice and the continuously forming water, the ice can rise close to the boundary, separated by a thin gap of water. The melting dynamics of the ice appear qualitatively similar to the evaporation of a drop under Leidenfrost conditions; however, the extent of the analogy is unclear. Here we investigate the melting dynamics of ice in thin-walled cylindrical containers. Through a combination of experiments and physical modeling, we identify a characteristic melting time and gap thickness, which we compare to evaporating droplets.

Does buoyancy matter in the melting dynamics of ice?

JICHEONG GUO, MUSTAFA ORDU, SOUMENDRA BASU, JAMES BIRD, Boston University — Ice in a horizontal cylindrical container will melt when placed in a sufficient warm environment. Because of the density difference between the ice and the continuously forming water, the ice can rise close to the boundary, separated by a thin gap of water. The melting dynamics of the ice appear qualitatively similar to the evaporation of a drop under Leidenfrost conditions; however, the extent of the analogy is unclear. Here we investigate the melting dynamics of ice in thin-walled cylindrical containers. Through a combination of experiments and physical modeling, we identify a characteristic melting time and gap thickness, which we compare to evaporating droplets.

Study on Electric field assisted low frequency (20 kHz) ultrasonic spray

ILKYEONG CHAE, BAEKHOON SEONG, DARMAWAN MARTEN, DOYOUNG BYUN, Sungkyunkwan Univ. — Ultrasonic spray is one of the fabulous techniques to discharge small size of droplets because it utilizes ultrasonic vibration on nozzle. However, spray patterns and size of ejected droplet is hardly controlled in conventional ultrasonic spray method. Therefore, here we present electric field assisted ultrasonic spray, which combined conventional technique with electric field in order to control spray pattern and droplet size precisely. Six kinds of various liquid (D.I water, Ethanol, Acetone, Iso-propanol, Toluene, Hexane) with various dielectric constants were used to investigate the mechanism of this method. Also, PIV (Particle Image Velocimetry) was used and various variables were obtained including spray angle, amplitude of liquid vibration, current, and size distribution of ejected droplets. Our electric field assisted ultrasonic spray show that the standard deviation of atomized droplet was decreased up to 39.6%, and it shows the infinite possibility to be utilized in various applications which require precise control of high transfer efficiency.

This work was supported by the Swedish Research Council and the Linné FLOW Centre.

Spray atomization

GARETH MCKINLEY, MIT, MIT COLLABORATION, AXALTA COATING SYSTEMS COLLABORATION — Animals drying their wet fur disintegrate liquid films into smaller fragments. Narrower size distributions and well-defined geometrical fluid pathlines (similar to the involute pattern) are the main advantages of this type of atomization as compared to air-assisted atomization. Despite these inherent advantages there are some drawbacks to this method. Also, PIV (Particle Image Velocimetry) was used and various variables were obtained including spray angle, amplitude of liquid vibration, current, and size distribution of ejected droplets. Our electric field assisted ultrasonic spray show that the standard deviation of atomized droplet was decreased up to 39.6%, and it shows the infinite possibility to be utilized in various applications which require precise control of high transfer efficiency.

This work was supported by the Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Education, Science and Technology (2014-023284).
The results are compared against direct numerical simulations. A low-order approach is used by solving the biharmonic equation in a coordinate system naturally mapping to the droplet shape. We then approach the dynamic problem: in this case, the full Stokes equations throughout the drop must be considered. A low-order approach is used by solving the biharmonic equation in a coordinate system naturally mapping to the droplet shape. This behavior was suppressed in the number probability distribution. Additionally, we employ an in-house particle detection code to isolate the rim fragment size distribution from the total probability distributions. Our experiments showed that the bag fragments are smaller in diameter and larger in number, while the rim fragments are larger in diameter and smaller in number. Furthermore, with increasing We for a given Oh we observe a large number of small-diameter drops and small number of large-diameter drops. On the other hand, with increasing Oh for a fixed We the opposite is seen.

The results are compared against direct numerical simulations.

3:41PM D32.00008 Experimental investigation of the breakup of a round liquid jet in a shock-induced crossflow. JOSEPH OLLES, DANIEL GUILDENBECHER, JUSTIN WAGNER, EDWARD DEMAURO, PAUL FARIAIS, THOMAS GRASSER, Sandia National Laboratories, PAUL SOJKA, Purdue University — The breakup of a round water jet due to a step change in the convective air velocity following a 1D air-shock was experimentally investigated. Variations of this experiment have been conducted in the past, however here quantitative results on the breakup sizes and trajectories are shown. A shock tube was utilized to create the jet breakup, and the primary shape of the liquid and secondary droplet sizes were recorded optically. Through the use of digital in-line holography (DIH), the sizes, 3D position, and 3C velocities of secondary droplets were measured at kHz rates. Care was taken to ensure that the jet was kept round throughout the shock tube test section (absent of Plateau-Rayleigh instability). While the liquid jet geometry and velocity was kept constant, various gas-phase velocities allowed for the investigation of multiple breakup morphologies, as a function of the crossflow Weber number. The typical breakup regimes are seen: bag, multimode, and sheet-thinning. With high temporal and spatial resolution, interfacial and liquid column instabilities are seen in the jet breakup.

3:54PM D32.00009 Drop shaping and fragmentation by laser-pulse impact. ALEXANDER L. KLEIN, WILCO BOUWHUIS, CLAAS WILLEM VISSER, Physics of Fluids Group, Faculty of Science and Technology, University of Twente, The Netherlands, HENRI LHUISSIER, Laboratoire Matière et Systèmes Complexes, Université Paris Diderot, France, JACCO H. SNOEIJER, Physics of Fluids Group, Faculty of Science and Technology, University of Twente, The Netherlands, EMMANUEL VILLERMAUX, Aix-Marseille Université, IRPHE, France, DETLEF LOHSE, HANNEKE GELDERBLOM, Physics of Fluids Group, Faculty of Science and Technology, University of Twente, The Netherlands — We show how the deposition of laser energy in a superficial layer of an unconfined liquid drop can lead to propulsion, strong deformation and eventually fragmentation of the drop. Combining high-speed with stroboscopic imaging, we reveal that the laser-induced vaporization at the drop surface is the driving mechanism for the hydrodynamic response of the drop. We provide scaling arguments for the linear relations between the absorbed laser energy and both the propulsion speed and the lateral expansion of the drop prior to its fragmentation. The resulting drop shape is well reproduced by Boundary Integral simulations. Last, we show by high speed imaging in a front and side-view configuration how instabilities develop on the deforming liquid sheet, which eventually results in the drop breaking-up in smaller fragments. We characterize this fragmentation process and its dependence on the laser pulse properties.

4:07PM D32.00010 Compressibility effects in droplets impacted by a laser pulse. STEN REIJERS, Physics of Fluids, Faculty of Science and Technology, University of Twente, The Netherlands, FEDERICO TOSCHI, Department of Applied Physics and Department of Mathematics and Computer Science, Eindhoven University of Technology, The Netherlands, DETLEF LOHSE, Physics of Fluids, Faculty of Science and Technology, University of Twente, The Netherlands, JACCO SNOEIJER, Mesoscopic Transport Phenomena, Eindhoven University of Technology, The Netherlands, HANNEKE GELDERBLOM, Physics of Fluids, Faculty of Science and Technology, University of Twente, The Netherlands — The impact of a laser pulse onto a liquid droplet can induce a strong deformation and propulsion of the droplet. We can model this laser impact as a one-sided pressure pulse applied on the liquid-vapor interface of the droplet. We aim to understand the fluid dynamic response of the droplet in a regime where the duration of the pressure pulse is very short, i.e., of the order of the timescale on which pressure waves travel through the droplet. We use Lattice-Boltzmann simulations to study the effects of pressure-wave propagation on a number of phenomena: energy partition, wave reflection on interfaces, droplet deformation and cavitation. We complement the simulation by a perturbation analysis of the Navier-Stokes equations in the weakly compressible regime. In the weakly compressible regime, we observe good agreement between the Lattice-Boltzmann simulations and this analytical model.

Sunday, November 22, 2015 2:10PM - 4:20PM — Session D33 Drops: Wetting and Spreading I Ballroom A - Omar Matar, Imperial College London

2:10PM D33.00001 Low-order modelling of droplets on hydrophobic surfaces. OMAR MATAR, ALEX WRAY, LYES KAHOUADJI, Imperial College London, STEPHEN DAVIS, Northwestern University — We consider the behaviour of a droplet deposited onto a hydrophobic substrate. This and associated problems have garnered a wide degree of attention due to their significance in industrial and experimental settings, such as the post-rupture dewetting problem. These problems have generally defied low-order analysis due to the multi-valued nature of the interface. We show here how to overcome this in this instance. We first discuss the static problem: when the droplet is stationary, its shape is prescribed by an ordinary differential equation (ODE) given by balancing gravitational and capillary stresses at the interface. This is dependent on the contact angle, the Bond number and the volume of the drop. In the high Bond number limit, we derive several low-order models of varying complexity to predict the shape of such drops. These are compared against numerical calculations of the ODE. We then approach the dynamic problem: in this case, the full Stokes equations throughout the drop must be considered. A low-order approach is used by solving the biharmonic equation in a coordinate system naturally mapping to the droplet shape. The results are compared against direct numerical simulations.

1EPSRC Programme Grant, MEMPHIS, EP/K0039761/1, EPSRC Doctoral Prize Fellowship (AWW)
We explored the condensation of water drops on a lubricant-impregnated surface, i.e., a micropillar patterned surface impregnated with a ionic liquid. Growing drops were imaged in 3D using a laser scanning confocal microscope equipped with a temperature and humidity control. On a lubricant-impregnated hydrophobic micropillar array, different stages of condensation can be discriminated: - Nucleation on a lubricant surface. - Regular alignment between micropillars and formation of a three-phase contact line on a bottom of the substrate. - Deformation and bridging by coalescence, leading to a detachment of the drops from the bottom substrate to pillars’ top faces. However, on a lubricant-impregnated hydrophilic micropillar array, the condensed water covers the micropillars by dewetting the lubricant. As a result, the surface loses its slippery property. Our results provide fundamental concepts how these solid/liquid hybrid surfaces can be applied for facile removal of condensed water, as well as necessity of the appropriate surface treatment.

1Financial support from ERC for the advanced grant 340391-SUPRO is gratefully acknowledged.

2:23PM D33.00002 Optical Imaging of Water Condensation on Lubricant Impregnated Micropillar Arrays1. TADASHI KAJIYA, FRANK SCHELLENBERGER, Max Planck Institute for Polymer Research, PERIKLIS PAPADOPOULOS, University of Ioannina, DORIS VOLLMER, HANS-JRGEN BUTT, Max Planck Institute for Polymer Research — We explored the condensation of water drops on a lubricant-impregnated surface, i.e., a micropillar patterned surface impregnated with a ionic liquid. Growing drops were imaged in 3D using a laser scanning confocal microscope equipped with a temperature and humidity control. On a lubricant-impregnated hydrophobic micropillar array, different stages of condensation can be discriminated: - Nucleation on a lubricant surface. - Regular alignment between micropillars and formation of a three-phase contact line on a bottom of the substrate. - Deformation and bridging by coalescence, leading to a detachment of the drops from the bottom substrate to pillars’ top faces. However, on a lubricant-impregnated hydrophilic micropillar array, the condensed water covers the micropillars by dewetting the lubricant. As a result, the surface loses its slippery property. Our results provide fundamental concepts how these solid/liquid hybrid surfaces can be applied for facile removal of condensed water, as well as necessity of the appropriate surface treatment.

1Financial support from ERC for the advanced grant 340391-SUPRO is gratefully acknowledged.

2:36PM D33.00003 Dynamics of Spreading on Micro-Textured Surfaces, ALIREZA MOHammad KARIM, University of California, Los Angeles, JONATHAN ROTHSTEIN, UMassAmherst, PIROUZ KAVEHPOUR, University of California, Los Angeles — Ultra hydrophobic surfaces, due to their large water-repellency characteristic, have a vast variety of applications in technology and nature, such as de-icing of airplane wings, efficiency increase of power plants, and efficiency of pesticides on plants, etc. The significance of ultra-hydrophobic surfaces requires enhancing the knowledge on the spreading dynamics on such surfaces. The best way to produce an ultra-hydrophobic surface is by patterning of smooth hydrophobic surfaces with micron sized posts. In this research, the micro-textured surfaces have been fabricated by patterning several different sizes of micro-textured posts on Teflon plates. The experimental study has been performed using forced spreading with Tensometer to obtain the dependency of dynamic contact angle to the contact line velocity to describe the spreading dynamics of Newtonian liquids on the micro-textured surfaces. The effect of the geometrical descriptions of the micro-posts along with the physical properties of liquids on the spreading dynamics on micro-textured Teflon plates have been also studied.
Droplet spreading and absorption on rough permeable substrates:  LEONARDO ESPIN, SATISH KUMAR, University of Minnesota — Wetting of permeable substrates by liquids is an important phenomenon in many natural and industrial processes. Substrate heterogeneities may significantly alter liquid spreading and interface shapes, which in turn may alter liquid imbibition. A new lubrication-theory-based model for droplet spreading on permeable substrates that incorporates surface roughness is developed in this work. The substrate is assumed to be saturated with liquid, and the contact-line region is described by including a precursor film and disjoining pressure. A novel boundary condition for liquid imbibition is applied that eliminates the need for a droplet velocity. Geometries for different geometries (coarse-grained and fine-grained substrates with roughness on the micrometer scale) are studied, confined between two plates, droplet release, and droplet capture. For the latter two geometries, some part of the incline consists of a “sandwich,” free sliding, and jumping and the Reynolds number based on the droplet radius are 0.6-4 and 260-680, respectively. The capillary number is of order $10^{-2}$. Mathematical Sciences, Nanyang Technological University — We seek to understand the effect of confinement on the transport properties of water droplets on a substrate. Good agreement is found between the model and experimental results.

Droplet spreading and absorption on rough permeable substrates:  LEONARDO ESPIN, SATISH KUMAR, University of Minnesota — Wetting of permeable substrates by liquids is an important phenomenon in many natural and industrial processes. Substrate heterogeneities may significantly alter liquid spreading and interface shapes, which in turn may alter liquid imbibition. A new lubrication-theory-based model for droplet spreading on permeable substrates that incorporates surface roughness is developed in this work. The substrate is assumed to be saturated with liquid, and the contact-line region is described by including a precursor film and disjoining pressure. A novel boundary condition for liquid imbibition is applied that eliminates the need for a droplet velocity. Geometries for different geometries (coarse-grained and fine-grained substrates with roughness on the micrometer scale) are studied, confined between two plates, droplet release, and droplet capture. For the latter two geometries, some part of the incline consists of a “sandwich,” free sliding, and jumping and the Reynolds number based on the droplet radius are 0.6-4 and 260-680, respectively. The capillary number is of order $10^{-2}$. Mathematical Sciences, Nanyang Technological University — We seek to understand the effect of confinement on the transport properties of water droplets on a substrate. Good agreement is found between the model and experimental results.

A boundary condition for fluid/fluid flow at the solid interface:  SHAHRHARAF AKHAMI, New Jersey Institute of Technology, USA, STEPHANE ZALESKI, Sorbonne Univ, UPMC Univ Paris 06, UMR 7190, Inst Jean Le Rond d'Alembert, France, ALEXANDRE GUIGNON, JACOPO BUONGIORNO, Massachusetts Institute of Technology, USA — When using classical hydrodynamics, the imposition of no-slip boundary condition on a solid surface results in a logarithmically singular shear force at the fluid/fluid/solid line of contact. Boundary conditions that admit some form of slip at the solid interface, such as the linear slip-shear relation also known as Navier slip boundary condition, have been used to regularize this singularity. The widely used Cox's theoretical work is also derived under the assumption of slip at the solid interface. Here we present a boundary condition, based on Cox's analysis, that resolves the contact line singularity in the numerical simulation of contact line flows using a volume of fluid method. This boundary condition provides a universal formula that determines the contact angle as a function of the grid size. The model can be thought of as a subgrid-scale model for computations, where the resolution is not sufficient to accurately describe the contact line inner region frictional drag. We provide numerical examples for a wide range of steady flow problems.

Phase-field simulations of contact-line dynamics on rough surfaces:  FENGCHAO YANG, Northwestern Polytechnical University, China, and Virginia Tech, PENGTAO YUE, Virginia Tech, XIAOPENG CHEN, Northwestern Polytechnical University. Shaanxi, China — Wetting of solid surfaces is ubiquitous in nature, and in most cases the surfaces are not smooth. In this work, we will investigate how surface roughness influences contact-line dynamics by simulating the forced wetting in a capillary tube with microposts or microgrooves on its wall. A phase-field method is used to capture fluid interfaces as well as moving contact lines. The governing equations are solved by an implicit finite-element method on an adaptive triangular mesh, which conforms to the topological patterns on the tube wall and also refines at the fluid interface. With our computational setup, we can capture the advancing and receding contact angles. As the contact line moves, it jumps from one pillar to the next; consequently, the apparent contact angle exhibits a periodic behavior. By comparing with the result on a smooth surface, we will explain how surface roughness affects contact-line dynamics by modifying the effective contact angle and slip length. Numerical results on different arrangements and shapes of microposts will be presented.

Sunday, November 22, 2015 2:10PM - 4:20PM — Session D34: Superhydrophobic Surfaces Ballroom BC - Jonathan Boreyko, Virginia Tech

2:10PM D34.00001 Self-Propelled Sweeping Removal of Dropwise Condensate on Two-Tier Superhydrophobic Surfaces, JONATHAN BOREYKO, Virginia Tech, XIAOPENG QU, FANGJIE LIU, Duke University, REBECCA AGAPOV, NICKOLAY LAVRIK, SCOTT RETTERER, Oak Ridge National Laboratory, JAMES FENG, University of British Columbia, PATRICK COLLIER, Oak Ridge National Laboratory, CHUAN-HUA CHEN, Duke University, NATURE-INSPIRED FLUIDS AND INTERFACES TEAM, MICROSCE Microscale Physicochemical Hydrodynamics Laboratory Team, Center for Nanophase Materials Sciences Team, Department of Mathematics Team — Dropwise condensation can be enhanced by nanostructured superhydrophobic surfaces, on which the condensate drops spontaneously jump upon coalescence. However, the self-propelled jumping in prior reports is mostly perpendicular to the substrate. Here we propose a substrate design with regularly spaced micropillars. Coalescence on the sidewalls of the micropillars leads to self-propelled jumping in a direction nearly orthogonal to the pillars and therefore parallel to the substrate. This in-plane motion in turn produces sweeping removal of multiple neighboring drops. The spontaneous sweeping mechanism may greatly enhance dropwise condensation in a self-sustained manner.

2:23PM D34.00002 Wenzel to Cassie transition for droplet impingement, CRISTIAN CLAVIJO, JULIE CROCKETT, DANIEL MAYNES, Brigham Young University — Advantages posed by self-cleaning, superhydrophobic surfaces quickly diminish as the liquid penetrates gas-filled cavities resulting in the so-called Wenzel state. To prevent penetration, surfaces must exhibit nanoscale features since penetrating pressure increases significantly for decreasing feature size. However, certain applications require microscale roughness such as those seeking to relax the no-slip condition and thus penetration reversal in microscale features remains of interest. Unfortunately, recent efforts to accomplish such reversal are complicated or locally-disruptive to the flow such as electrically-tunable surfaces and boiling. Here, we show that a Wenzel-to-Cassie transition is possible with a modest surface temperature increase. Dynamics are discussed for a water droplet impinging (We=100) on a wide range of superhydrophobic surfaces with features varying in height from 4 microns to 18 microns and separation distance of 8 microns to 16 microns. Results reveal that dewetting rates increase with increasing feature height and temperature up to 30 mL/s. A first order model is constructed to validate our hypothesis that surface tension and triple line dissipation are the two dominating forces during dewetting. Good agreement is found between the model and experimental results.

We gratefully acknowledge the National Science Foundation for funding this work. We gratefully acknowledge the National Science Foundation for funding this work.

2:36PM D34.00003 Droplets on inclined super hydrophobic substrates: between “sandwich,” free sliding and jumping, JULIAN MARTINEZ MERCADO, CLAUS-DIETER OHL, School of Physical and Mathematical Sciences, Nanyang Technological University — We seek to understand the effect of confinement on the transport properties of droplets on super hydrophobic surfaces. In a straightforward experiment, the droplet slides down an incline while being sandwiched between two plates. The dynamics is captured from two views, the lateral centre of mass motion and the three-dimensional motion. The range of Weber and Reynolds number based on the droplet radius are 0.6-4 and 260-680, respectively. The capillary number is of order $10^{-3}$. Three geometries are studied, confined between two plates, droplet release, and droplet capture. For the latter two geometries, some part of the incline consists of lower and upper plates. The experimental observations are that the acceleration of a “sandwiched” droplet is considerably reduced to a free sliding one. Droplets being released jump off the substrate converting considerable amount of the surface energy into potential energy. Droplet capture obeys a limit kinetic energy, below that, they are reflected from the construction. We hope to present detail of the flow within the sandwiched droplet by the time of presentation.

This work was supported by the Singapore National Research Foundations Competitive Research Program funding (NRF-CRP9-2011-04).
2:49PM D34.00004 Drop impact dynamics on liquid-infused superhydrophobic surfaces , JEONG-HYUN KIM, JONATHAN ROTHSTEIN, Univ of Mass - Amherst — In this talk, we present a series of experiments investigating the drop impact dynamics on hydrophobic, air-infused and lubricant-infused superhydrophobic surfaces. To create the superhydrophobic surfaces, smooth Teflon (PTFE) surfaces were roughened by a 240-grit sandpaper. The immiscible and incompressible silicone oils with different viscosities were infused into features of the superhydrophobic surfaces by a skim coating technique. The spreading and retraction dynamics on a series of the tested surfaces will be presented. We will show that the maximal deformation of the drops on lubricant-infused surfaces grows with increasing viscosity ratio between a water drop and the infused oil. We will show that this increase in the maximal deformation with the viscosity ratio is consistent with increasing the velocity and the viscosity of the drops but the rims of the drops destabilize with increasing the drop velocity. Finally, we will demonstrate that increasing the viscosity of the infused oil induces higher viscous force at the contact line, resulting in reduction in the movement of the drops during retraction and corresponding increase in the final drop size.

3:02PM D34.00005 Anti-fogging surfaces , TIMOTHE MOUTERDE, PMMH, ESPCI / LadjHyX, Ecole Polytechnique, ANTONIO CHECCO, Condensed Matter Physics and Materials Science Department, Brookhaven National Laboratory, CHARLES BLACK, ATUKIR RAHMAN, Center for Functional Nanomaterials, Brookhaven National Laboratory, CHRISTOPHE CLANET, DAVID QUR, PMMH, ESPCI / LadjHyX, Ecole Polytechnique — Achieving an anti-fogging material is more challenging than achieving an anti-rain material. A relevant way to investigate the resistance to fog consists of depositing hot water on a cold surface. We show that classical superhydrophobic surfaces with micron-size microstructures lose their superhydrophobic behaviour due to vapour condensation. To understand this phenomenon, we measured the adhesion force of hot water drops on different substrates and propose a quantitative description of this force generated by condensation. Our main result is that reducing the scale of the structures can strongly promote antifogging properties.

3:15PM D34.00006 Orientation Dependence of Jumping Droplet Condensation . AUSTIN BERRIER, JONATHAN BOREYKO, Virginia Tech, NATURE-INSPIRED FLUIDS AND INTERFACES TEAM — On nanostructured superhydrophobic surfaces, microscopic condensate exhibits out-of-plane jumping that minimizes the average droplet size for maximal phase-change heat transfer. This jumping-droplet phenomenon occurs independently of gravity and is due to surface energy being partially converted to kinetic energy upon coalescence events. Although the initial departure of the jumping droplets is independent of gravity, the subsequent trajectories exhibit a dependence upon the orientation of the substrate. The drop size distribution of jumping-droplet condensation growing on a smooth superhydrophobic substrate was characterized for both horizontal and vertical surface orientations. With the horizontal orientation, jumping condensate returns to the substrate by gravity. While this can result in chain reactions with other droplets to trigger further jumping events, eventually the rebounding droplets become too large to jump and are stuck on the surface. In contrast, droplets jumping off a vertically oriented surface do not return to the substrate. For this reason, the maximum droplet diameters during condensation growth were found to be significantly larger on the horizontally oriented superhydrophobic surface than on the vertical orientation.

3:28PM D34.00007 On the origin of surface fraction scaling for receding contact angles on textured superhydrophobic surfaces , ETIENNE BARTELH, ESPCI/CNRS, JEREMIE TESSIERE, CNRS/Saint-gobain, MARCO RIVETTI, MPI DS / Goettingen — It has long been recognized that surface fraction is a relevant parameter to rationalize the receding contact angle on textured superhydrophobic surfaces [1]. This notion can easily be rationalized from a simple surface energy averaging procedure, which recently been challenged because it neglects the Cassie relation. See the recent paper of Rivetti et al. Phys. Rev. Lett. 115 (2015) 016101 for a discussion. In this talk, we revisit this problem by exploring strongly anisotropic surfaces for which surface fraction and line fraction scalings are clearly differentiated. Our experimental and simulation results suggest that surface fraction scaling originates from line defects. Since these defects straddle rows, they probe both lattice dimensions, whereby surface fraction scaling emerges. However, our results also show that strict proportionality as predicted by the Cassie relation does not hold: a much more singular behavior is found at low surface fractions, in keeping with the near-threshold behaviour expected from a depinning process [3]. [1] QUERE, D. Annu. Rev. Mater. Res. 38 (2008) 71 [2] CHOI, W.; A. al et. J. Colloid Interf. Sci. 339 (2009) 208 [3] RIVETTI et al. Phys. Rev. Lett. 115 (2015) 016101

3:41PM D34.00008 Layers of Porous Superhydrophobic Surfaces for Robust Water Repellency , FARZAD AHMADI, JONATHAN BOREYKO, Virginia Tech, NATURE-INSPIRED FLUIDS AND INTERFACES TEAM — In nature, birds exhibit multiple layers of superhydrophobic feathers that repel water. Inspired by bird feathers, we utilize porous superhydrophobic surfaces and compare the wetting and dewetting characteristics of a single surface to stacks of multiple surfaces. The superhydrophobic surfaces were submerged in water in a closed chamber. Pressurized gas was regulated to measure the critical pressure for the water to fully penetrate through the surfaces. In addition to using duck feathers, two-tier porous superhydrophobic surfaces were fabricated to serve as synthetic mimics with a controlled surface structure. The energy barrier for the wetting transition was modeled as a function of the number of layers and their orientations with respect to each other. Moreover, after partial impalement into a subset of the superhydrophobic layers, it was observed that a full dewetting transition was possible, which suggests that natural organisms can exploit their multiple layers to prevent irreversible wetting.

3:54PM D34.00009 A computational study of the impingement of water droplets onto freezing superhydrophobic surfaces1 , WEN JIN, BEHROOZ AMIRZADEH, MAZDAK TOOTKABONI, MEHDI RAESSI, University of Massachusetts Dartmouth, UNIVERSITY OF MASSACHUSETTS DARTMOUTH TEAM — We present computational simulations of the impingement of micron-size water droplets onto freezing superhydrophobic surfaces at various Weber numbers, droplet initial temperatures, and surface temperatures. The simulation results are from an in-house volume-of-fluid based, free-surface flow solver with phase change. The objective is to investigate the conditions under which the droplets bounce off the surface or stick to the surface and freeze. The transition between the bouncing and sticking regimes is shown. Then, using a dimensional analysis of the timescales for droplet freezing and drop-surface contact, a theoretical model is proposed for predicting the above transition. Finally, the predictions of the theoretical model are compared to the transition conditions observed in the computational simulations.

1Funding from the National Science Foundation CBET-1336232 grant is gratefully acknowledged.

4:07PM D34.00010 Transferring heat during a bounce , SAMIRA SHIRI, JAMES BIRD, Boston University — When a hot liquid drop impacts a cold non-wetting surface, the temperature difference drives heat transfer. If the drop leaves the surface before reaching a supercooling equilibrium, the rate of heat transfer may depend on the quench time. Past studies exploring finite-time heat exchange with droplets focus on the Leidenfrost condition where heat transfer is regulated by a thin layer of vapor. Here, we present systematic experiments to measure the heat transferred by a bouncing droplet in non-Leidenfrost conditions. We propose a physical model of this heat transfer and compare our model to the experiments.
2:10PM D37.00001 The fate of electrospay drops . OSMAN BASARAN, Purdue University, ROBERT COLLINS, Oak Ridge National Laboratory, KRISHNARAJ SAMBATH, MICHAEL HARRIS, Purdue University — Drops subjected to strong electric fields emit thin fluid jets from conical structures (Taylor cones) that form at their surfaces. Such behavior has practical, e.g., electrospray mass spectrometry, and fundamental, e.g., raindrops in thunderclouds, implications. Theoretical analysis of the temporal development of such EHD tip-streaming phenomena is challenging given the large disparity in length scales between the macroscopic drops and the microscopic jets. Furthermore, there exist conflicting theories and measurements on the size and charge of these small electrospay droplets. We use theory and simulation to show that conductivity can be tuned to yield three scaling regimes for droplet radius and charge, a finding missed by previous studies. The amount of charge Q that electrospay droplets carry determines whether they are coulombically stable and charged below the Rayleigh limit of stability R or are unstable and hence prone to further explosions once formed. Previous experiments reported droplet charge values ranging from 1/10th to in excess of R. Simulations unequivocally show that electrospay droplets are coulombically stable at the instant they are created and that there exists a universal scaling law for droplet charge, Q=0.44 R.

2:23PM D37.00002 Electro-rotation of drops at large electric Reynolds numbers . EHUD YARIV, ITZCHAK FRANKEL, Technion — We analyze spontaneous electrohydrodynamic rotation of drops under a uniform electric field by applying the Taylor-Melcher leaky-dielectric model to a two-dimensional system. The dimensionless problem is governed by the ratios of electric conductivities, dielectric permittivities and shear viscosities in the respective drop- and suspending liquid phases as well as the electric Reynolds number Re, quantifying surface-charge convection. We address the asymptotic limit of large Re where the dominant balance in the boundary conditions results in the flow scaling as Re^{-1/2}. This flow is governed by a nonlinear boundary-value problem which does not admit a fore-aft symmetric solution, thus necessitating drop rotation. This problem, which is invariant to the inversion of the velocity field, is transformed into a universal one, independent of the conductivity and permittivity ratios. Thermodynamic arguments reveal that a solution exists only when charge relaxation within the suspending liquid is faster than that in the drop. Under these conditions, the rescaled angular velocity is obtained as a function of the viscosity ratio. Comparable numerical solutions, obtained using the exact equations, indeed collapse at large Re upon the asymptotic universal solution.

2:36PM D37.00003 Electrohydrodynamic deformation of drops and bubbles at large Reynolds numbers . ORY SCHNITZER, Department of Mathematics, Imperial College London — In Taylor’s theory of electrohydrodynamic drop deformation by a uniform electric field, inertia is neglected at the outset, resulting in fluid velocities that scale with E^2, E being the applied-field magnitude. When considering strong fields and low viscosity fluids, the Reynolds number predicted by this scaling may actually become large, suggesting the need for a complementary large-Reynolds-number analysis. Balancing viscous and electrical stresses reveals that the velocity scales with E^{1/3}. Considering a gas bubble, the external flow is essentially confined to two boundary layers propagating from the poles to the equator, where they collide to form a radial jet. Remarkably, at leading order in the Capillary number the unique scaling allows through application of integral mass and momentum balances to obtain a closed-form expression for the O(E^2) bubble deformation. Owing to a concentrated pressure load at the vicinity of the collision region, the deformed profile features an equatorial dimple which is non-smooth on the bubble scale. The dynamical importance of internal circulation in the case of a liquid drop leads to an essentially different deformation mechanism. This is because the external boundary layer velocity attenuates at a short distance from the interface, while the internal boundary-layer matches with a Prandtl-Batchelor (PB) rotational core. The dynamic pressure associated with the internal circulation dominates the interfacial stress profile, leading to an O(E^{8/3}) deformation. The leading-order deformation can be readily determined, up to the PB constant, without solving the circulating boundary-layer problem. To encourage attempts to verify this new scaling, we shall suggest a favourable experimental setup in which inertia is dominant, while finite-deformation, surface-charge advection, and gravity effects are negligible.

2:49PM D37.00004 The relaxation of a prolate leaky dielectric drop in a uniform DC electric field . ADITYA KHAIIR, JAVIER LANAUZE, LYNN WALKER, Department of Chemical Engineering, Carnegie Mellon University — We quantify the relaxation of a prolate leaky dielectric drop upon removal of a uniform DC electric field. Experiments consisting of a castor oil drop suspended in a silicone oil are compared against boundary integral simulations that account for transient charging of the interface. Charge relaxation causes a marked asymmetry in the drop evolution during deformation and relaxation. In particular, during relaxation a prolate to oblate shape transition is observed before the drop recovers its equilibrium spherical shape. Furthermore, the high field strengths utilized in the experiments yield a fast drop relaxation in comparison with the transient development towards the steady deformation. The storage and release of capacitive energy and capillary energy is then quantified during deformation and relaxation, respectively. Finally, we present computational results for a drop that does not relax back to its initial spherical shape upon removal of the field; rather, the drop breaks up.

3:02PM D37.00005 Nonlinear electrohydrodynamics of leaky dielectric drops in the Quincke regime: Numerical simulations1 . DEBASISH DAS, DAVID SAINTILLAN, Univ of California - San Diego — The deformation of leaky dielectric drops in a dielectric fluid medium when subject to a uniform electric field is a classic electrohydrodynamic phenomenon best described by the well-known Melcher-Taylor leaky dielectric model. In this work, we develop a three-dimensional boundary element method for the full leaky dielectric model to systematically study the deformation and dynamics of liquid drops in strong electric fields. We compare our results with existing numerical studies, most of which have been constrained to axisymmetric drops or have neglected interfacial charge convection by the flow. The leading effect of convection is to enhance deformation of prolate drops and suppress deformation of oblate drops, as previously observed in the axisymmetric case. The inclusion of charge convection also enables us to investigate the dynamics in the Quincke regime, in which experiments exhibit a symmetry-breaking bifurcation leading to a tank-treading regime. Our simulations confirm the existence of this bifurcation for highly viscous drops, and also reveal the development of sharp interfacial charge gradients driven by convection near the drop’s equator.

1 American Chemical Society, Petroleum Research Fund
3:15PM D37.00006 Simulations of particle structuring driven by electric fields, YI HU, Northwestern University. PETIA VLAHOVSKA, Brown University. MICHAEL MIKESIS, Northwestern University — Recent experiments (Ouriemi and Vlahovska, 2014) show intriguing surface patterns when a uniform electric field is applied to a droplet covered with colloidal particles. Depending on the particle properties and the electric field intensity, particles organize into an equatorial belt, pole-to-pole chains, or dynamic vortices. Here we present 3D simulations of the collective particle dynamics, which account for electrohydrodynamic flow and dielectrophoresis of particles. In stronger electric fields, particles are expected to undergo Quincke rotation and impose disturbance to the ambient flow. Transition from ribbon-shaped belt to rotating clusters is observed in the presence of the rotation-induced hydrodynamical interactions. Our results provide insight into the various particle assemblies discovered in the experiments.

3:28PM D37.00007 Electrohydrodynamic Displacement of Polarizable Liquid Interfaces in an Alternating Current Electric Field, ZACHARY GAGNON, Johns Hopkins University — In this work, we investigate Maxwell-Wagner polarization at electrically polarizable liquid interfaces. An AC electric field is applied across a liquid electrical interface created between two co-flowing microfluidic fluid streams with different electrical properties. When potentials as low as 2 volts are applied, we observe a frequency dependent interfacial displacement that is dependent on the relative differences in the electrical conductivity and dielectric constant between the two liquids. At low frequency this deflection is dependent on electric field strength, and only depends on dielectric constant at high frequency. At intermediate frequencies, we observe a crossover that is independent of applied voltage, sensitive to both fluid electrical properties, and where no displacement is observed. An analytical polarization model is presented that predicts the liquid interfacial crossover frequency, the dependence of interfacial displacement on liquid electrical conductivity and dielectric constant, and accurately scales the interface displacement measurements. The results show that liquid interfaces are capable of polarizing under AC electric fields and being precisely deflected in a direction and magnitude that is dependent on the applied electric field frequency.

3:41PM D37.00008 Fluctuation and dynamics of a lipid bilayer membrane under an electric field, YUAN-NAN YOUNG, New Jersey Institute of Technology. MICHAEL MIKESIS, Northwestern University. PETIA VLAHOVSKA, Brown University — Membrane fluctuation and dynamics under an electric field is investigated, and results show that the membrane instability and dynamics depend not only on the mismatch in conductivity and permittivity between the bulk fluids, but also on the membrane charging time. In addition, the (entropic) membrane tension is found to depend on the electric field. Lubrication theory is utilized to examine the nonlinear dynamics of a planar lipid bilayer membrane with and without electrokinetics.

3:54PM D37.00009 Shape fluctuations of a giant lipid vesicle in an external electric field, NICO FRICKE, PETIA VLAHOVSKA, Brown University — We experimentally study the influence of an applied electric field on the physical properties of lipid bilayer membranes. Global and regional analyses of the shape fluctuations of a giant quasi-spherical vesicle (“flicker spectroscopy”) are used to infer membrane tension, and bending rigidity from a time series of microscope images. The parameters of the electric field (frequency and amplitude) are chosen such that there is no global vesicle deformation, and hence any renormalization of the tension and bending rigidity arises only from electric stress in the membrane. Using this approach we examine the effect of the electrotension on the main phase transition temperature of lipid membranes, where we observe that increasing field strength decreases, albeit slightly (about 0.1K), the melting temperature.

Supported by NSF/DMS 1222550, 1412789

1Partial support from NSF/CMMI 1232477.

Sunday, November 22, 2015 2:10PM - 4:20PM — Session D38 Flow Instability: Boundary Layers I

2:10PM D38.00001 Experimental Study of Leading Edge Receptivity to Freestream Local Disturbance, YU NISHIO, TOMONARI OKA, SEICHIRO IZAWA, YU FUKUNISHI, Tohoku University — Leading edge receptivity of a flat plate is investigated in a wind tunnel experiment. Sheet-jet type velocity fluctuations are introduced into the freestream using a wing-shaped disturbance generator installed upstream of the plate. Steady or pulsating jet is ejected from the generator filling the velocity deficit of the generator body. So the device generates velocity fluctuations without changing the velocity profile. Whether the generated velocity fluctuations affect the growth of fluctuations inside boundary layer is examined in detail. When the generator is placed at the same height as the stagnation point of the leading edge, the velocity fluctuations taken into the boundary layer just monotonically decay. On the other hand, when the generator is placed slightly higher than the stagnation point level, the velocity fluctuations inside the boundary layer tentatively grow showing higher receptivity, which gradually decay downstream. The strength of the velocity fluctuations inside the boundary layer are related to the velocity fluctuations outside the boundary layer. The effects of pulsating the jet are limited in both experiments.

2:23PM D38.00002 Biglobal stability analysis of spatially developing axisymmetric boundary layers, VINOD NARAYANAN, RAMESH BHORANIYA, Indian Institute of Technology, Gandhinagar — Global stability analysis of incompressible axisymmetric boundary layers is performed. In the present analysis, we consider two geometries: axial flow over circular cylinders and circular cones. The Bi-global stability boundary equations together with the boundary conditions form an eigenvalue problem and are solved using Arnoldi’s algorithm. Chebyshev spectral collocation method is used for discretization of the stability equations. Spatial growth rate of disturbance waves at different Reynolds numbers and azimuthal wave numbers are computed. The results show that the disturbances have non-wave like behavior. The normalized spatial amplification in streamwise direction increases with increase in Reynolds number for axisymmetric mode. However, for non-axisymmetric modes it reduces with increase in Reynolds number. In case of flow over circular cone, stability analysis is performed for different cone angles, range of Reynolds numbers and different azimuthal wave numbers. The disturbances do not show wavelike behavior in this too. The flow is found to be spatially destabilizing in the case of circular cones at low Reynolds numbers. Thus the effect of transverse curvature here is destabilizing the flow. Detailed results will be presented at the time of conference.
Boundary layer transition is implied by the temperature distribution and power spectral evolution of the fluctuation pressure. It is found there exists a streamwise high-temperature strip near the central meridian line of the leeward side and transition is most likely to occur along the strip (earlier than the windward side and 0º angle of attack). Besides, the radius of the cone tip has an effective influence on the transition location. Transition is more likely to occur on the cone with the sharper tip.

Moreover, since this configuration is characterized by a slowly-varying flow field in streamwise direction, a local stability analysis is applied to define the neutral stability curves for the BL flow controlled by this type of wall modifications. These results give the possibility of investigating this control strategy and understanding the effect of the free parameters on the stabilization mechanism. [1]Fransson, Downs and Sattarzadeh. TSFP-9, Melbourne 05/30-06/03, 2015

This work was supported by NSF grant CBET-1228195. Computer time was provided by the Extreme Science and Engineering Discovery Environment (XSEDE).
3:54PM D38.00009 Numerical modeling of the transitional boundary layer over a flat plate. DIMITRY IVANOV, ANDREI CHORNY, Heat and Mass Transfer Institute, Minsk, Belarus — Our example is connected with fundamental research on understanding how an initially laminar boundary layer becomes turbulent. We have chosen the flow over a flat plate as a prototype for boundary-layer flows around bodies. Special attention was paid to the near-wall region in order to capture all levels of the boundary layer. In this study, the numerical software package OpenFOAM has been used in order to solve the flow field. The results were used in a comparative study with data obtained from Large Eddy Simulation (LES). The composite SGS-wall model is presently incorporated into a computer code suitable for the LES of developing flat-plate boundary layers. Presently this model is extended to the LES of the zero-pressure gradient, flat-plate turbulent boundary layer. In current study the time discretization is based on a second order Crank-Nicolson/Adams-Bashforth method. LES solver using Smagorinsky and the one-equation LES turbulence models. The transition models significantly improve the prediction of the onset location compared to the fully turbulent models. LES methods appear to be the most promising new tool for the design and analysis of flow devices including transition regions of the turbulent flow.

4:07PM D38.00010 Transitional regime and laminarturbulent coexistence in the asymptotic suction boundary layer. TARAS KHAPKO, PHILIPP SCHLATTER, Linn FLOW Centre, KTH Mechanics, YOHANN DUGUET, LIMSI-CNRS, UPR 3251, Campus Universitaire d’Orsay, DAN HENNINGSON, Linn FLOW Centre, KTH Mechanics — We study numerically the asymptotic suction boundary layer (ASBL) in the transitional regime on the verge of laminarization. Starting from a turbulent state the Reynolds number $Re$ is decreased in small steps until the laminar state is established. This study protocol allows not only to investigate the regime at the onset of turbulence, but also to identify the critical Reynolds number $Re_c \approx 270$, below which turbulence is not sustained. In other planar shear flows the transitional regime at the onset takes the form of stable laminar-turbulent bands, however in ASBL no regime of sustained laminar-turbulent coexistence has been identified. The flow stays fully turbulent even at the lowest $Re$ before laminarization. During the laminarization process streamwise turbulent and laminar avenues are created with no oblique interfaces between the two. This behavior is connected with the existence of a large-scale vertical transport, the feature that distinguishes ASBL from the other wall-bounded shear flows. After an artificial forcing is added canceling all spanwise and wall-normal fluctuations above $y^+ = 100$, transient oblique bands are observed similar to the ones in other subcritical shear flows, while the flow later laminarizes or becomes fully turbulent again.

Sunday, November 22, 2015 2:10PM - 4:20PM — Session D39 Flow Instability: Richtmyer-Meshkov I Sheraton Back Bay C - Oleg Schilling, Lawrence Livermore National Laboratory

2:10PM D39.00001 A Comparative Study of Two-, Three- and Four-Equation Multi-component Reynolds-Averaged Navier-Stokes Model Predictions of Turbulent Mixing in Reshocked Richtmyer-Meshkov Instability. OLEG SCHILLING, Lawrence Livermore National Laboratory — A multicomponent implementation of two-, three- and four-equation Reynolds-averaged Navier-Stokes models using either the turbulent kinetic energy dissipation rate or lengthscale as the second mechanical turbulence quantity is applied to model a Mach 1.5 reshocked Richtmyer-Meshkov instability in the light-to-heavy and heavy-to-light cases. The model includes mixture molecular transport terms, enthalpy diffusion terms, pressure-dilatation and dilatation dissipation models, and a molecular diffusion flux with contributions from baro- and thermodiffusion. The four-equation models couple transport equations for the mass flux $a_i$, and the negative density-specific volume correlation $b$ to the $K-V$ or $K-L$ equations, while the three-equation models use an algebraic closure for $b$. The evolution of various turbulence statistics, fields, and turbulent transport equation budgets are compared among these models to identify any differences in the turbulence production, dissipation and diffusion physics represented by the closures used in these models.

2:23PM D39.00002 Richtmyer-Meshkov mixing: experiments on the effect of initial conditions. STUART CRAIG, RICARDO MEJIA-ALVAREZ, BRANDON WILSON, KATHY PRESTRIDGE, Los Alamos National Laboratory — The development of the Richtmyer-Meshkov instability (RMI) is sensitive to a number of parameters, including incident Mach number and the initial perturbation to the interface. A set of experiments at Los Alamos National Laboratory are underway using the Vertical Shock Tube (VST) with the aim of exploring the relationships between these two parameters. These experiments have been carried out with a single initial condition at three Mach numbers and at a single Mach number with three different initial conditions. This talk will focus specifically on the results on the effects of the different initial conditions on the early-time development of the RMI mixing at an air-SF$_6$ interface. Simultaneous measurements of the velocity (PIV) and density (PLIF) fields are used to explore the relationships between three types of initial conditions and the resulting early-time mixing at a single Mach number. Phase averaging of the flow field is employed in order to reduce intermittency and improve the statistical convergence of a number of turbulence statistics such as Favre-averaged Reynolds stresses, mixing rate, and enstrophy.

2:36PM D39.00003 Experimental study of Mach number effects on the evolution of Richtmyer-Meshkov instabilities. RICARDO MEJIA-ALVAREZ, BRANDON WILSON, ALEX CRAIG, KATHY PRESTRIDGE, Los Alamos National Laboratory — The evolution of Richtmyer-Meshkov instabilities from the initial linear growth stages to the subsequent non-linear interactions and the eventual (sometimes elusive) transition to turbulence, is strongly dependent on a number of factors such as shock strength (Mach number), Atwood number, and the initial structure of the fluid interface. Mach number controls the effective value of the Atwood number after compression, and thus the distribution and total amount of kinetic energy deposited at shock interaction. The initial scale-content in the fluid interface defines how quickly and to what extent growing instabilities interact with each other, ultimately conditioning transition to turbulence. These effects are entirely independent of each other, and the extent of their relative importance is not well understood. To shed light on this subject, we designed a parameter space consisting of three different Mach numbers (1.1, 1.3, and 1.45) and three different interface configurations of varying scale content. This parameter space is being explored experimentally by means of simultaneous PIV/PLIF measurements on a single air-SF$_6$ interface as it evolves after shock interaction. This talk will focus on the observation of Mach number effects for an early stage of evolution.
2:49PM D39.00004 Comparison of the Effects of Mach Number on the Spatiotemporal Evolution of Turbulence and Mixing in Reshocked Richtmyer-Meshkov Instability1, TIBERIUS MORAN-LOPEZ, National Nuclear Security Administration, OLEG SCHILLING, Lawrence Livermore National Laboratory — The predictions of a multicomponent K∗ Reynolds-averaged Navier-Stokes model applied to reshocked Richtmyer-Meshkov instability experiments with progressively larger incident shock Mach numbers are compared in detail. The model includes molecular dissipation and diffusion, mean and turbulent enthalpy diffusion, and closure models for pressure-dilatation and dilatation dissipation. This model was previously shown to give mixing layer widths in very good agreement with experimental data for a wide range of cases, including the Vetter-Sturtevant, Poggi et al., Leinov et al., and Houas-Chemouni experiments with Mach numbers ranging from 1.20 to 4.20. The spatiotemporal evolution of various statistics, fields, and transport equation budgets are compared among the cases considered here to quantify the effects of increasing Mach number on the intensity of turbulence and mixing both before and after reshock.

1This work was performed under the auspices of the US Department of Energy by Lawrence Livermore National Laboratory under Contract DE-AC52-07NA27344.

3:02PM D39.00005 Numerical Simulations of the turbulent Richtmyer-Meshkov instability in a spherically convergent geometry1, ISMAEL DJIBRILLA BOUREIMA, PRAVEEN RAMAPRABHU, University of North Carolina at Charlotte — We investigate the development of the turbulent Richtmyer-Meshkov instability in a spherically convergent geometry. The three-dimensional simulations were performed using the astrophysical FLASH code [1], with a resolution of 1024 × 512 × 512 in the radial, azimuthal and polar directions for the multimode case. We present results from two sets of simulations, namely a spherical RM driven by a self-similar Chisnell [2] shock and an implosion problem defined by [3]. In both configurations, the shock travels from an outer fluid layer to an inner fluid that is denser. The implosion problem produces significantly greater convergence than the standard RM problem, allowing for significant enhancement of the turbulent mixing zone due to Bell-Plesset effects. We report on several quantities of interest from both simulations.

1This work was supported in part by the (U.S.) Department of Energy (DOE) under Contract No. DE-AC52-06NA25396.

3:15PM D39.00006 Numerical, Dimensional, and Computational considerations in Large Eddy Simulations of the Richtmyer-Meshkov Instability1, BRITTON OLSON, Lawrence Livermore National Laboratory — The shock induced mixing of two gases separated by a perturbed interface is investigated through Large Eddy Simulation using two different high-order numerical methods. Results from a recently published collaborative study are presented which show remarkable similarities between quantities and metrics representing mixing and turbulence. Small differences between the results, however, do elucidate the differences in the two numerical methods and their strengths and weaknesses. Results from two-dimensional calculations of the same problem are also shown to highlight differences from the three-dimensional case. Finally, the feasibility in a hybrid compressible/incompressible calculation is discussed, which shows considerable computational savings as compared to the fully compressible case.

1This work was performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under contract number DE-AC52-07NA27344.

3:28PM D39.00007 Linear Stability Analysis of Magnetohydrodynamic Richtmyer-Meshkov Instability in Cylindrical Geometry1, ABEER BAKHSH, RAVI SAMTANEY, King Abdullah University of Science and Technology — Numerical simulations and analysis in Cartesian slab geometry for nonlinear ideal magnetohydrodynamics (MHD) indicate that the Richtmyer-Meshkov instability (RMI) is suppressed in the presence of a magnetic field. An analytical solution of incompressible 2-D MHD RMI of an impulsively accelerated interface was investigated by Wheatley et al. (Phys. Rev. Lett. 2005; J. Fluid Mech. 2005) who found that, for a finite magnetic field, although the initial growth rate of the interface is unaffected by the presence of magnetic field, the late-time amplitude of the interface asymptotes to a constant value. In the framework of incompressible MHD, we examine analytically the behavior of an impulsively accelerated interface separating conducting fluids of different densities in cylindrical geometry. We investigate the stability properties of such a system and study the influence of the magnetic field on the growth rate of the interface. In converging cylindrical geometry, the RMI is followed by a Rayleigh-Taylor (RT) phase. Our analysis does not account for the RT phase of the instability but is valid for the duration of the RMI phase. We compare results of the incompressible analysis with linear compressible MHD simulations.

1Supported by the KAUST Office of Competitive Research Funds under Award No. URF/1/2162-01.

3:41PM D39.00008 Suppression of the spherically converging magnetohydrodynamic Richtmyer-Meshkov instability in an octahedrally symmetric seed magnetic field1, WOUTER MOSTERT, Graduate Aerospace Laboratories, California Institute of Technology, VINCENT WHEATLEY, School of Mechanical and Mining Engineering, University of Queensland, DALE FULLIN, Graduate Aerospace Laboratories, California Institute of Technology, RAVI SAMTANEY, Mechanical Engineering, King Abdullah University of Science and Technology — We present results of ideal magnetohydrodynamics simulations investigating the Richtmyer-Meshkov instability in near-spherical implosions in the presence of an octahedrally symmetric seed magnetic field. The problem is motivated by the desire to maintain a symmetrical collapse of the primary shock wave, minimally distorted by the effect of the seed magnetic field, while retaining the seed-field-induced suppression of the Richtmyer-Meshkov instability. The field is generated by a set of six current loops arranged around the target as on the faces of a cube. The instability is generated on a perturbed spherical density interface that is accelerated from the outside by impeding magnetohydrodynamic shocks, which are in turn generated by a spherical Riemann problem. The perturbation on the density interface is formed with a single-dominant-mode spherical harmonics expansion. We investigate the evolution of the interface and the transport of baroclinic vorticity near the interface, and examine the extent of the distortion to the primary magnetohydrodynamic shock system induced by the seed field.

1This work was partially supported by the KAUST Office of Sponsored Research under Award URF/1/2162-01.
3:54PM D39.00009 Multiphase Instabilities in Explosive Dispersal of Particles\textsuperscript{1}.
BERTRAND ROLLIN, FREDERICK OUELLET, SUBRAMANIAN ANNAMALAI, S. “BALA” BALACHANDAR, Center for Compressible Multiphase Turbulence - Univ. of Florida — Explosive dispersal of particles is a complex multiphase phenomenon that can be observed in volcanic eruptions or in engineering applications such as multiphase explosives. As the layer of particles moves outward at high speed, it undergoes complex interactions with the blast wave structure following the reaction of the energetic material. Particularly in this work, we are interested in the multiphase flow instabilities related to Richmyer-Meshkov (RM) and Rayleigh-Taylor (RT) instabilities (in the gas phase and particulate phase), which take place as the particle layer disperses. These types of instabilities are known to depend on initial conditions for a relatively long time of their evolution. Using a Eulerian-Lagrangian approach, we study the growth of these instabilities and their dependence on initial conditions related to the particulate phase – namely, (i) particle size, (ii) initial distribution, and (iii) mass ratio (particles to explosive). Additional complexities associated with compaction of the layer of particles are avoided here by limiting the simulations to modest initial volume fraction of particles. A detailed analysis of the initial conditions and its effects on multiphase RM/RT-like instabilities in the context of an explosive dispersal of particles is presented.

\textsuperscript{1}This work was supported by the U.S. Department of Energy, National Nuclear Security Administration, Advanced Simulation and Computing Program, as a Cooperative Agreement under the Predictive Science Academic Alliance Program, Contract No. DE-NA0002378.

4:07PM D39.00010 Numerical Simulation of Multi-Material Mixing in an Inclined Interface Richtmyer-Meshkov Instability\textsuperscript{1}, AKSHAY SUBRAMANIAM, SANJIVA LELE, Stanford Univ — The Richtmyer-Meshkov instability arises when a shock wave interacts with an interface separating two fluids. In this work, high fidelity simulations of shock induced multi-material mixing between $N_2$ and $CO_2$ in a shock tube are performed for a Mach 1.55 shock interacting with a planar material interface that is inclined with respect to the shock propagation direction. In the current configuration, unlike the classical perturbed flat interface case, the evolution of the interface is non-linear from early time onwards. Our previous simulations of this problem at multiple spatial resolutions have shown that very small 3D perturbations have a large effect on vortex breakdown mechanisms and hence fine scale turbulence. We propose a comparison of our simulations to the experiments performed at the Georgia Tech Shock Tube and Advanced Mixing Laboratory (STAAML). Results before and after reshock of the interface will be shown. Results from simulations of a second case with a more complex initial interface will also be presented. Simulations shown are conducted with an extended version of the Miranda solver developed by Cook et. al (2007) which combines high-order compact finite differences with localized non-linear artificial properties for shock and interface capturing.

\textsuperscript{1}This research is part of the Blue Waters sustained-petascale computing project, which is supported by the National Science Foundation (awards OCI-0725070 and ACI-1238993) and the state of Illinois.

Sunday, November 22, 2015 2:10PM - 4:20PM –
Session D40 Flow Instability: Rayleigh-Taylor I
Sheraton Back Bay D - Andrew Cook, Lawrence Livermore National Laboratory

2:10PM D40.00001 Lessons Learned from Numerical Simulations of Interfacial Instabilities\textsuperscript{1}, ANDREW COOK, LLNL — Rayleigh-Taylor (RT), Richtmyer-Meshkov (RM) and Kelvin-Helmholtz (KH) instabilities serve as efficient mixing mechanisms in a wide variety of flows, from supernovae to jet engines. Over the past decade, we have used the Miranda code to temporally integrate the multi-component Navier-Stokes equations at spatial resolutions up to 29 billion grid points. The code employs 10th-order compact schemes for spatial derivatives, combined with 4th-order Runge-Kutta time advancement. Some of our major findings are as follows: The rate of growth of a mixing layer is equivalent to the net mass flux through the equi-molar plane. RT growth rates can be significantly reduced by adding shear. RT instability can produce shock waves. The growth rate of RM instability can be predicted from known interfacial perturbations. RM vortex projectiles can far outrun the mixing region. Thermal fluctuations in molecular dynamics simulations can seed instabilities along the braids in KH instability. And finally, enthalpy diffusion is essential in preserving the second law of thermodynamics.

\textsuperscript{1}This work was performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract DE-AC52-07NA27344.

2:23PM D40.00002 Exploring elastic and plastic regimes of Rayleigh-Taylor instability in solids\textsuperscript{1}, RINOSH POLAVARAPU, ARINDAM BANERJEE, Lehigh University — The elastic-plastic (EP) transition stage of Rayleigh-Taylor (RT) instability in an accelerated non-Newtonian material (soft solid) is investigated. The material exhibits both elastic and plastic behavior when the applied stress is less than yield stress. Different combinations of perturbation amplitude and wavelength are employed at the solid-gas interface. Plastic deformation of a stable interface under various acceleration profiles is analyzed. In addition, the evolution of the RT instability at various strain rates is examined by altering the angular acceleration of rotating disk on which the test section is mounted. These findings are used to characterize the effects of strain rate variation on the instability growth rate and the experimental results are compared to several analytical models on RTI in solids. The instability threshold for a perturbation of given amplitude and wavelength is observed to increase with an increase in angular acceleration. The perturbation velocities measured and used to estimate values of growth-rate parameter in the unstable phase.

\textsuperscript{1}Authors acknowledge financial support through a DOE-SSAA Grant # DE-NA0001975 and a LANL subcontract.

2:36PM D40.00003 Reynolds and Atwood Numbers Effects on Homogeneous Rayleigh Taylor Instability\textsuperscript{1}, DENIS ASLANGIL, Lehigh University, DANIEL LIVESCU, Los Alamos National Laboratory, ARINDAM BANERJEE, Lehigh University — The effects of Reynolds and Atwood numbers on turbulent mixing of a heterogeneous mixture of two incompressible, miscible fluids with different densities are investigated by using high-resolution Direct Numerical Simulations (DNS). The flow occurs in a triply periodic 3D domain, with the two fluids initially segregated in random patches, and turbulence is generated in response to buoyancy. In turn, stimulation produced by turbulence breaks down the scalar structures, accelerating the molecular mixing. Statistically homogeneous variable-density (VD) mixing, with density variations due to compositional changes, is a basic mixing problem and aims to mimic the core of the mixing layer of acceleration driven Rayleigh Taylor Instability (RTI). We present results covering a large range of kinematic viscosity values for density contrasts including small (A=0.04), moderate (A=0.5), and high (A=0.75 and 0.9) Atwood numbers. Particular interest will be given to the structure of the turbulence and mixing process, including the alignment between various turbulence and scalar quantities, as well as providing fidelity data for verification and validation of mix models.

\textsuperscript{1}Arindam Banerjee acknowledges support from NSF CAREER award # 1453056.
2:49PM D40.00004 Understanding the Rayleigh-Taylor instability through 1D and 3D simulations , MARK MIKHAIEL, Georgia Institute of Technology, NICHOLAS DENISSEN, Los Alamos National Laboratory, DEVESH RANJAN, Georgia Institute of Technology — A series of Rayleigh-Taylor instability numerical simulations were completed using the Arbitrary Lagrangian-Eulerian hydrocode FLAG developed at Los Alamos National Laboratory. One-dimensional simulations employed a Reynolds-averaged Navier-Stokes approach with turbulence closure models selected from the Besnard-Harlow-Raunzahn family of models. Growth rate parameters and turbulence statistics are derived from these simulations and compared between closure models. Variations from existing theories are explored and the bubble-spike growth and mass transfer symmetry is also investigated. High resolution three-dimensional large eddy simulations (LES) are also completed and presented. LES were initialized using a multi-modal perturbation prescribed from experimental data collected at the Georgia Institute of Technology multi-layer Gas Tunnel facility. Turbulence statistics are gathered by averaging many simulations started with different initial conditions. Late time development is compared to Gas Tunnel experimental results and previous LES.

3:02PM D40.00005 Direct Numerical Simulations of Immiscible Rayleigh-Taylor Instability, ZHAORUI LI, Texas A&M University—Corpus Christi, DANIEL LIVESCU, Los Alamos National Laboratory — Accurate simulations of multi-mode immiscible Rayleigh-Taylor instability (RTI) are presented with the recently developed generalized Cahn-Hilliard Navier-Stokes (GCHNS) equations method. In immiscible turbulent flows, besides the viscous cut-off scale, there are two additional characteristic length scales, which also affect the flow. One is the so-called “cut-off” length scale caused by the presence of surface tension and the other is the physical interface thickness. While in some practical applications the interface thickness can be large, in many other cases this thickness approaches the molecular scales. Accurate results can be obtained for these cases if the interface thickness is maintained smaller than all the cut-off scales of the flow, but still much larger than the molecular scales (e.g. mean free path). Our study shows that, as long as the scale-separation (e.g. the ratio of Kolmogorov scale to the interface thickness) is above a certain value (4 to 6 for the RTI problem considered in this study), the numerical results are fully converged with respect to the interface thickness. The results are used to study the physics of multi-mode immiscible RTI and contrasted to those obtained for the miscible case.

3:15PM D40.00006 DSMC Simulations of the Rayleigh-Taylor Instability in Gases, MICHAEL GALLIS, TIMOTHY KOEHLER, JOHN TORCZYNSKI, STEVEN PLIMPTON, Sandia National Labs — The Direct Simulation Monte Carlo (DSMC) method of molecular gas dynamics is applied to simulate the Rayleigh-Taylor instability (RTI) in atmospheric-pressure monatomic gases (e.g., argon and helium). The computational domain is a 1 mm × 4 mm rectangle divided into 50-nm square cells. Each cell is populated with 1000 computational molecules, and time steps of 0.1 ns are used. Simulations are performed to quantify the growth of a single-mode perturbation on the interface as a function of the Atwood number and the gravitational acceleration. The DSMC results qualitatively reproduce all observed features of the RTI and are in reasonable quantitative agreement with existing theoretical and empirical models. Consistent with previous work in this field, the DSMC simulations indicate that the growth of the RTI follows a universal behavior. For cases with multiple-mode perturbations, the numbers of bubble-spike pairs that eventually appear are found to be in agreement with theoretical results for the most unstable wavelength. Sandia National Laboratories is a multi-program laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy’s National Nuclear Security Administration under contract DE-AC04-94AL85000.

3:28PM D40.00007 Evolution of Rayleigh-Taylor growth following an initial Richtmyer-Meshkov instability, JEREMY MELVIN, Stony Brook University, BAOLIAN CHENG, Los Alamos National Lab, JAMES GLIMM, Stony Brook University, DAVID H. SHARP, Los Alamos National Lab — We investigate the evolution of a Rayleigh-Taylor (RT) growth on top of an already developing Richtmyer-Meshkov (RM) instability. Using numerical simulations utilizing a front tracking algorithm and theoretical analysis through a buoyancy-drag equation, an initially perturbed density discontinuity is subjected to a shock wave and then after a delay, a gravitational field is applied. The early time growth of the RM/RT instability is compared with a pure RT instability and asymptotic growth rate predictions for the RT evolution. A dependence on the RM seeding is shown to produce a meaningful change in the perturbation growth at early times. The implications of this to Inertial Confinement Fusion analysis, which largely focuses on the RT instability, is discussed.

3:41PM D40.00008 The Compressible Rayleigh-Taylor Instability and Vortex Dynamics in Stratified Media, SCOTT A. WIELAND, University of Colorado at Boulder, DANIEL LIVESCU, Los Alamos National Laboratory, OLEG V. VASILYEV, University of Colorado at Boulder, SCOTT J. RECKINGER, Montana State University — Fully resolved adaptive wavelet-based direct numerical simulations (WDNS) of the single-mode, compressible, and miscible Rayleigh-Taylor instability (RTI) have been performed at Reynolds numbers significantly larger than those previously attained. To ensure that WDNS properly captures the full extent of the length and time scales, an exhaustive resolution study was completed. The ensuing results explore the effects of compressibility and background stratification on the vortex generation and interaction that serves as the driver behind the RTI development beyond the early stages. To better understand the eventual suppression that arises at large background stratification, the simplified cases of a pair of counter rotating vortices (2D) and a vortex ring (3D) in stratified media are also presented for the purpose of isolating and explaining the physics behind these effects on RTI growth.

3:54PM D40.00009 Rayleigh-Taylor Instability in non-premixed reacting flames, NITESH ATTAL, PRAVEEN RAMAPRABHU, University of North Carolina at Charlotte — The Rayleigh-Taylor instability (RTI) occurs at a perturbed interface between fluids of different densities when a light fluid pushes a heavier fluid. The mixing driven by RTI affects several physical phenomena, such as Inertial Confinement Fusion, Supernovae detonation, centrifugal combustors and liquid rocket engines. The RTI in such flows is often coupled with chemical/nuclear reactions that may form complex density stratifications in the form of flames or ablative layers. We investigate such a non-premixed fuel-air interface subject to a constant acceleration and developing under the influence of chemical reactions using high-resolution, Navier-Stokes simulations [1]. The H2 fuel is diluted with N2 to vary the density difference across the interface in thermal equilibrium (at 1000K). The intervening layer between fuel and air is subject to exothermic combustion reactions to form a flame. Following combustion, initially unstable fuel-air interfaces at an Atwood number (A1) <0.5, transform into stable (fuel-flame) and unstable (flame-air) interfaces. We report on interfaces (A1 = 0.2 and 0.6) with single wavelength, sinuosoidal perturbations and a broadband spectrum of multimode perturbations. [1] Attal, N., et al. Comput. Fluids 107 (2015): 59-76.
Effects of the Rayleigh-Taylor Instability Subject to a Changing Body Force, REBECCA BERTSCH, ROBERT CORE. Los Alamos National Laboratory — Modeling turbulent mixing in variable density (VD) fluid flows is a key topic of interest in multi-physics applications due to the complex instability characteristics they exhibit. RANS models continue to be accurate and efficient tools to investigate the evolution of turbulence in these complex flow problems. Many RANS models are well validated for prototypical variable density flows such as Rayleigh-Taylor (RT) and Richtmeyer-Meshkov (RM). However, most lack the ability to accurately capture mix features in VD flows with changing body forces, like those seen in rocket rig experiments that undergo phases of acceleration and deceleration. This talk will present some simulations of an improved RANS model which substitutes the molecular diffusion term in the species equation with a demix term that is dependent on the turbulent mass flux and species micro-densities. Results from these simulations will be compared with previous RANS models, DNS, and experimental data to validate the new models ability to capture the mixing physics in RT flow subject to a changing body force.

Sunday, November 22, 2015 4:50PM - 6:08PM —
Session E1 Porous Media Flows: Clogging and Filtration  Auditorium - Ian Griffiths, University of Oxford

4:50PM E1.00001 How gradients in porosity can make a better filter, IAN GRIFFITHS, MARIA BRUNA, MOHIT DALWADI, University of Oxford — Depth filters are a common device for removing contaminants from fluid. Porosity-graded filters, whose porosities decrease with depth, have been shown experimentally to offer improved filtration efficiency over filters with uniform porosity, by allowing contaminants to be trapped more evenly within the filter media. However, experiments are unable to probe the microscopic behavior, and so the underlying mechanisms that are responsible for this improved filtration are unclear. We use homogenization theory to derive a macroscopic model for the fluid flow and particle trapping within a porosity-graded depth filter. We find that gradients in porosity induce a macroscale particle advection in the direction of reducing porosity and show how particle trapping is more evenly spread through the filter for a decreasing porosity compared with a uniform porosity. By quantifying the removal rate, we show how a given operating regime can be fine-tuned to improve filter efficiency. The talk is accompanied by an online demonstration of MEMFI, a software package in which audience members may explore for themselves the effect of porosity gradients in user-specified operating regimes.

5:03PM E1.00002 Gravity filtration of suspensions: permeability effects, TEJASWI SOORI, MENGYU WANG, THOMAS WARD, Department of Aerospace Engineering, Iowa State University — This paper examines the filtration rates of mono-modal suspensions as a function of time and a cake layer builds up through theory and experimentation. Darcy’s Law, which describes fluid flow through porous media, was applied along with the Kynch theory of sedimentation, which provides the basis for analyzing low concentration (\(\phi \leq 20\%\)) cake formation. Experiments were performed to study the effects of varying particle sizes (\(45 < d < 1400 \mu m\)) and total solid concentration \(\phi\) on both the formation rate of the cake layer and its flow permeability \((k)\) in conjunction with the filter media. A CCD camera was used to capture images of the cake formation and fluid drainage processes, and subsequent image and theoretical analysis found the fluid flow experienced a constant pressure loss due to the permeability of the filter media, whereas the experienced pressure loss due to the cake formation varies as a function of time, \(\phi\) and \(d\). The rate of cake formation was also found to be independent of \(\phi\) but dependent on \(d\) which can be attributed to a change in porosity affecting filtration. Studies on similar systems with multi-modal suspensions are in-progress.

5:16PM E1.00003 Flow and fouling in membrane filters: Effects of membrane morphology, PEJMAN SANAEI, LINDA J. CUMMINGS, New Jersey Institute of Technology — Membrane filters are widely-used in microfiltration applications. Many types of filter membranes are produced commercially, for different filtration applications, but broadly speaking the requirements are to achieve fine control of separation, with low power consumption. The answer to this problem might seem obvious: select the membrane with the largest pore size and void fraction consistent with the separation requirements. However, membrane fouling (an inevitable consequence of successful filtration) is a complicated process, which depends on many parameters other than membrane pore size and void fraction; and which itself greatly affects the filtration process and membrane functionality. In this work we formulate mathematical models that can (i) account for the membrane internal morphology (internal structure, pore size & shape, etc.); (ii) fouling of membranes with specific morphology; and (iii) make some predictions as to what type of membrane morphology might offer optimum filtration performance.

5:29PM E1.00004 Permeability modification in 3D porous media due to polymer retention, SHIMA PARSA, School of Engineering and Applied Sciences, Harvard University, HUBERT SIZARET, ESPCI ParisTech, PSL Research University, DAVID WEITZ, School of Engineering and Applied Sciences, Harvard University — We combine confocal microscopy and bulk transport measurements to quantify the changes in the permeability of a model porous media after flow of a polymer solution. The 3D micromodel is made of closely packed glass beads with diameter of 150 micrometers. By matching the index of refraction of the fluid with beads we are able to measure the fluid velocities at pore level deep in the medium using particle image velocimetry. Our measurement shows that the medium permeability decreases 60% after flow of multiple pore volumes of polymer solution and then flushing with water. At constant flow rate we estimate that the pore velocity increases almost 23% due to this reduction in permeability. Our microscopic measurements of the velocities in pores shows that the average velocity increases considerably more than estimated bulk value. Also the distribution of velocities has a slower decay indicating somewhat higher probability of large velocities in the medium after retention of polymer. These changes in velocities are not uniform and depends on the pore size.

5:42PM E1.00005 DEM Simulation of Particle Clogging in Fiber Filtration1, RAN TAO, MENGMENG YANG, SHUIQING LI, Tsinghua Univ — The formation of porous particle deposits plays a crucial role in determining the efficiency of filtration process. In this work, an adhensive discrete element method (DEM), in combination with CFD, is developed to dynamically describe these porous deposit structures and to characterize flow through them under the periodic boundary conditions. For the first time, it is clarified that the structures of clogged particles are dependent on both the adhesion parameter (defined as the ratio of interparticle adhesion to particle inertia) and the Stokes number (as an index of impaction efficiency). The relationship between the pressure-drop gradient and the coordination number along the filtration time is explored, which can be used to quantitatively classify the different filtration regimes, i.e., clean filter stage, clogging stage and cake filtration stage. Finally, we investigate the influence of the fiber separation distance on the particle clogging behavior, which affects the collecting efficiency of the fibers significantly. The results suggest that changing the arrangement of fibers can improve the filter performance.

1This work has been funded by the National Key Basic Research and Development Program (2013CB238506).
Effect of long-range electrostatic interaction on pore clogging in viscous particle flow

SHENG CHEN, MENGMENG YANG, SHUIQING LI, Tsinghua Univ — In this study, we implement the long-range electrostatic interactions (both Coulomb and dipole interactions) into the discrete-element simulation of small adhesive particles to investigate their influence on the formation of clogging patterns at single-pore level. The relationship between microscopic interparticle forces and the macroscopic clogging quantities, i.e., the flow permeability and clogging structures, is established. Simulated results indicate that the early-stage capture of charged particles is enhanced by the attraction between these particles and their induced charge on the wall surface. However, further aggregation is suppressed by the repulsive Coulomb interaction between the deposited particles and the suspended ones. Meanwhile, the attraction among polarized particles causes the formation of long particle chains on the surface. These particles chains, bended by flow stress, enhance the bridging phenomenon that leads to a rapid pore clogging. Comparatively, the final clogging structures have lower volume fraction and higher flow permeability in contrast to the neutral case. The results suggest that the controlled charging or polarizing of particles provide a feasible way to tune the formation process and the final state of pore clogging.

This work has been funded by the National Key Basic Research and Development Program (2013CB228506)

Sunday, November 22, 2015 4:50PM - 6:08PM — Session E2 Suspensions. Rheology II

101 - Itai Cohen, Cornell University

Direct visualization of particle scale internal stresses in a colloidal glass

ITAI COHEN, NEIL LIN, MATT BIERBAUM, JAMES SETHNA, Department of Physics, Cornell University — Bullet proof windshields, smart phone screens, and Prince Ruperts drop are all examples of how internal stresses can dramatically affect the strength of glass. Imaging the way internal stresses are distributed and their evolution under an applied load remains prohibitively difficult. For example, work on disordered granular packs suggests that stress heterogeneity is single-particle, while resolving stresses at the single atom scale is not feasible, measurements of stresses at the single particle scale in colloidal glasses, a widely used model system for atomic glasses, can be achieved by using Stress Assessment from Local Structural Anisotropy (SALSA). This method relies solely on the particle configurations obtained via high speed confocal microscopy. Here, we use SALSA to visualize the three dimensional stress network in a colloidal glass. By placing the suspension under shear we determine the evolution of this network and how it alters the bulk mechanical behavior of the suspension. Our work constitutes a first step towards understanding how local variations in the stress networks of glasses can lead to the dramatic mechanical properties of tempered glass.

Active microrheology of Brownian suspensions via Accelerated Stokesian Dynamics simulations

HENRY CHU, YU SU, Cornell University, KEVIN GU, Stanford University, NICHOLAS HOH, ROSEANNA ZIA, Cornell University — The non-equilibrium rheological response of colloidal suspensions is studied via active microrheology utilizing Accelerated Stokesian Dynamics simulations. In our recent work, we derived the theory for micro-diffusivity and suspension stress in dilute suspensions of hydrodynamically interacting colloids. This work revealed that force-induced diffusion is anisotropic, with qualitative differences between the pair to the driving force, and on the strength with which the probes are focused relative to the entropic restoring force of the suspension particles. This work also revealed that these forces play a similar qualitative role in the anisotropy of the stress and in the evolution of the non-equilibrium osmotic pressure. Here, we show that theoretical predictions hold for suspensions ranging from dilute to near maximum packing, and for a range of flow strengths from near-equilibrium to the pure-hydrodynamic limit.

Using Shear Reversal and Biaxial Shear Flows to Investigate Anisotropic Shear Thickening in Colloidal Suspensions

NEIL LIN, Department of Physics, Cornell University — Shear thickening is a ubiquitous phenomenon in suspension flow where an increase in shear rate gives rise to an increase in viscosity. Whether contact forces play a role in the rheology of colloidal suspensions or hydrodynamically evolving colloidal systems where hydrodynamic contributions are thought to dominate remains highly controversial. By performing shear reversal experiments on silica and latex colloidal particles, we directly measure the hydrodynamic and contact force contributions to the suspension viscosity. We find that contact forces are not only present, but dominate the shear thickening response. Since there are no system-spanning force networks in our low-volume fraction suspensions, it is not clear whether the thickening is isotropic or biased resulting in an anisotropic viscosity. To answer this question we employ biaxial shear rheology to determine whether thickening in such suspensions is isotropic. We apply a primary dominant shear flow to thicken the suspension, and simultaneously measure the suspension viscosity along the orthogonal direction using a secondary weak flow. We report on the evolution of this orthogonal viscosity as the system is driven into the shear thickening regime.

Non-equilibrium depletion interactions: first things attract, then they repel.

BENJAMIN DOLATA, ROSEANNA ZIA, Cornell University — Non-equilibrium depletion interactions in colloidal suspensions are studied theoretically via a combination of asymptotic and numerical solutions of the Smoluchowski equation. A pair of probes at arbitrary separation is driven by an external force at arbitrary orientation through a suspension, deforming the surrounding microstructure. The degree to which the structure is distorted, and the shape of this deformation, depends on the separation between the probes, on the orientation of the pair to the driving force, and on the strength with which the probes are focused relative to the entropic restoring force of the suspension particles. The resultant non-equilibrium osmotic pressure gradients give rise to both drag and interactive forces between the probes. When the external force is zero, the depletion attraction of Asakura and Oosawa is recovered. When an external force is applied, the interactive force can lead either to attraction or repulsion, as well as deterministic re-orientation of the probes relative to the external force, depending on initial separation, orientation, and strength of forcing. The use of this model for interrogation of non-continuum and elastically networked materials is explored.

Effect of confinement induced structures on suspension rheology

MEERA RAMASWAMY, BRIAN LEAHY, YEN-CHIH LIN, ITAI COHEN, Department of Physics, Cornell University — Understanding the flow behavior of confined colloidal systems is important in many industrial settings ranging from inkjet printers to pharmaceuticals. Confined colloidal suspensions under shear demonstrate many fascinating responses including vorticity-aligned strings in colloidal liquids and buckled phases in crystals. Despite the extensive studies of these confinement induced structures, the interplay between these exotic structural responses and the suspension rheology remains poorly understood. Here, we use a confocal rhapscope to image the suspension particle configuration while simultaneously measuring its stress responses. The confocal rhapscope has two precisely-aligned parallel plates that can confine the suspension with a variable gap size ranging from 3 to 20 particle diameters, allowing us to measure the response of the system as a function of the gap size. Moreover, we alter the rheological properties of the sample by adding a small amount of dimers. The dimers undergo Jeffery orbits at large strains and deform the confinement induced structures of the spheres, leading to a viscosity change. We discuss the results of these experiments and their implications in the areas of micro and nanofluidics.
to examine what makes science appealing to general audiences and suggest methods researchers can use to shape their work's broader impact. Appeals to a wide spectrum of readers in both age and field of study. This talk will utilize five years' worth of site content and reader feedback. Over the past five years, fluid dynamics outreach blog FYFD has published more than 1300 articles and gained an audience of over 215,000 readers. The site's demographics focused on include gender, race, age, college major, highest level of education, and profession. One of the main findings of this survey was doctors were chosen for this study), and determine if there are any patterns within different demographic groups. The demographics that were from a recent study on the perceptions of different professions will be presented. The study was designed based off of "draw-an-engineer" and Gonzaga University — One of the main reasons students do not join the STEM fields is that they lack interest in technical topics. But do people (young students, the general public, or even our own engineering students) know what an engineer is and/or does? In this talk, results from a recent study on the perceptions of different professions will be presented. The study was designed based off of "draw-an-engineer" and "draw-a-scientist" tests used on elementary schools kids. The idea is to have participants visualize professionals (engineers, lawyers, and medical doctors were chosen for this study), and determine if there are any patterns within different demographic groups. The demographics that were focused on include gender, race, age, college major, highest level of education, and profession. One of the main findings of this survey was that participants had the most difficult time visualizing an engineer compared to a lawyer or a medical doctor. Therefore, maybe we need more famous engineers (and fluid dynamists)?

1 Grant: Penn NSF MRSEC (DMR-1120901)

5:55PM E2.00006 Brownian Particles Under Shear: Rheology & Microstructure\(^1\). SOMAYEH FARHADI, MADHURA GURJAR, University of Pennsylvania, NATHAN KEIM, California Polytechnic State University — We present 2D rheological experiments of dense suspensions of Brownian (1 \(\mu\)m) particles. The particles, which are purely repulsive, are adsorbed at an oil-water interface and are sheared periodically by a magnetized needle. The area fraction of the sample is kept fixed at approximately 40%, which is above its glass transition. We measure the bulk rheology at low strain amplitudes while simultaneously track the particles in order to understand the microstructural contributions to yielding and plasticity in thermal systems. Previous studies on nonthermal colloids (of size 4-6 \(\mu\)m) identified a regime, below yielding point, where localized regions of space with reversible cycles undergo plastic deformations. For thermal systems, we anticipate to observe a transition from plastically reversible to irreversible states as the Péclet number (which characterizes the shear-induced to diffusion-induced displacements) is decreased. We also investigate the direction of diffusion of particles by probing anisotropy of the diffusion matrix, which gives us information on how the thermal and convective effects add up for highly packed systems.

\(^1\)This material is based upon work supported by the National Science Foundation under Grant No. EC-1240294.

5:03PM E3.00002 Do we need more famous fluid dynamicists?\(^2\), SHANON RECKINGER, Montana State University, BETHANY BRINKMAN, James Madison University, RAENITA FENNER, Loyola Maryland University, MARA LONDON, Gonzaga University — One of the main reasons students do not join the STEM fields is that they lack interest in technical topics. But do people (young students, the general public, or even our own engineering students) know what an engineer is and/or does? In this talk, results from a recent study on the perceptions of different professions will be presented. The study was designed based off of "draw-an-engineer" and "draw-a-scientist" tests used on elementary schools kids. The idea is to have participants visualize professionals (engineers, lawyers, and medical doctors were chosen for this study), and determine if there are any patterns within different demographic groups. The demographics that were focused on include gender, race, age, college major, highest level of education, and profession. One of the main findings of this survey was that participants had the most difficult time visualizing an engineer compared to a lawyer or a medical doctor. Therefore, maybe we need more famous engineers (and fluid dynamists)?

5:16PM E3.00003 F*** Yeah Fluid Dynamics: On science outreach and appealing to broad audiences. NICOLE SHARP, FYFD — Sharing scientific research with general audiences is important for scientists both in terms of educating the public and in pursuing funding opportunities. But it’s not always apparent how to make a big splash. Over the past five years, fluid dynamics outreach blog FYFD has published more than 1300 articles and gained an audience of over 215,000 readers. The site appeals to a wide spectrum of readers in both age and field of study. This talk will utilize five years’ worth of site content and reader feedback to examine what makes science appealing to general audiences and suggest methods researchers can use to shape their work’s broader impact.

5:29PM E3.00004 APS Education and Diversity Efforts. KATHERINE PRESTRIDGE, Los Alamos National Laboratory, THEODORE HODAPP, American Physical Society — The American Physical Society (APS) has a wide range of education and diversity programs and activities, including programs that improve physics education, increase diversity, provide outreach to the public, and impact public policy. We present the latest programs spearheaded by the Committee on the Status of Women in Physics (CSWP), with highlights from other diversity and education efforts. The CSWP is working to increase the fraction of women in physics, understand and implement solutions for gender-specific issues, enhance professional development opportunities for women in physics, and remedy issues that impact gender inequality in physics. The Conferences for Undergraduate Women in Physics, Professional Skills Development Workshops, and our new Professional Skills program for students and postdocs are all working towards meeting these goals. The CSWP also has site visit and conversation visit programs, where department chairs request that the APS assess the climate for women in their departments or facilitate climate discussions. APS also has two significant programs to increase participation by underrepresented minorities (URM). The newest program, the APS National Mentoring Community, is working to provide mentoring to URM undergraduates, and the APS Bridge Program is an established effort that is dramatically increasing the number of URM PhDs in physics.
5:42PM E3.00005 Engineering education research: Impacts of an international network of female engineers on the persistence of Liberian undergraduate women studying engineering. SARA RIMER, SAHITHYA REDDIVARI, ALINE COTEL, Univ of Michigan - Ann Arbor — As international efforts to educate and empower women continue to rise, engineering educators are in a unique position to be a part of these efforts by encouraging and supporting women across the world at the university level through STEM education and outreach. For the past two years, the University of Michigan has been a part of a grassroots effort to encourage and support the persistence of engineering female students at University of Liberia. This effort has led to the implementation of a leadership camp this past August for Liberian engineering undergraduate women, meant to: (i) to empower engineering students with the skills, support, and inspiration necessary to become successful and well-rounded engineering professionals in a global engineering market; and (ii) to strengthen the community of Liberian female engineers by building cross-cultural partnerships among students resulting in a international network of women engineers. This session will present qualitative research findings on the impact of this grassroots effort on Liberian female students? persistence in engineering, and the future directions of this work.

5:55PM E3.00006 STEM for Females goes from a Ripple to a Wave. RITA ROY, inTEST Thermal Solutions — At the 2015 World Conference of Science Journalists in Seoul on June 9, 2015 Nobel Prize Winner Sir Tim Hunt gave a speech with remarks which offended females in science around the world. Following the social media reaction several STEM education foundation for girls received justified attention. Programs are being supported for the education of girls and women around the world and they are gaining media attention. I will discuss the growing popularity and availability of all-girl science programs. These eager-to-learn young females are preparing themselves for serious careers in the hopes that their generation will close the gender gap. I will present a statistical analysis of workplace gender equality in professional STEM environments, including a focus on select fluid dynamics corporations, which will show that there is room for improvement across the industry. The goal of this presentation is to endorse and promote the ever-growing wave of qualified female applicants to graduate programs and industrial job placements.

Sunday, November 22, 2015 4:50PM - 5:55PM
Session E4 Compressible Flows: General 103 - Roy Baty, Los Alamos National Laboratory

4:50PM E4.00001 Group Invariance Properties of the Inviscid Compressible Flow Equations for a Modified Tait Equation of State. SCOTT RAMSEY, ROY BATY, Los Alamos National Laboratory — This work considers the group invariance properties of the inviscid compressible flow equations (Euler equations) under the assumptions of one-dimensional symmetry and a modified Tait equation of state (EOS) closure model. When written in terms of an adiabatic bulk modulus, a transformed version of these equations is found to be identical to that for an ideal gas EOS. As a result, the Lie group invariance structure of these equations and their subsequent reduction to a lower-order system is identical to the published results for the ideal gas case. Following the reduction of the Euler equations to a system of ordinary differential equations, a variety of elementary closed-form solutions are derived. These solutions are then used in conjunction with the Rankine-Hugoniot conditions to construct discontinuous shock wave and free surface solutions that are analogous to the classical Noh, Sedov, Guderley, and Hunter similarity solutions of the Euler equations for an ideal gas EOS. The versions of these problems for the modified Tait EOS are found to be semi-analytic in that a transcendental root extraction (and in some cases numerical integration of ordinary differential equations) enables solution of the relevant equations.

5:03PM E4.00002 Asymptotic Solutions of Detonation Propagation in a 2D Circular Arc. MARK SHORT, CHAD MEYER, JAMES QUIRK, Los Alamos National Laboratory — The large pressure of the product gas generated by detonating high explosives causes lateral motion of the explosive at the material interface between the explosive and its confinement. In turn, this leads to streamline divergence and curvature of the detonation front (typically in a divergent fashion). The propagation of a detonation front in a given geometry depends on the amount of curvature generated. Here we describe an asymptotic analysis of detonation propagation in a 2D circular arc, examining dependencies of the motion on the size of the inner and outer arc radii, and the relation between the detonation velocity and curvature for different types of explosive.

5:16PM E4.00003 Similarity Solutions of the Compressible Flow Equations for a General Equation of State. ZACHARY BOYD1, SCOTT RAMSEY, ROY BATY, Los Alamos National Laboratory — The Euler compressible flow equations admit discontinuous (e.g. shock) solutions regardless of the equation of state (EOS) used to close them. In addition, certain classes of initial conditions and EOS admit special flows known as similarity solutions, including some containing shocks. These are useful (1) as test problems for hydrocodes, (2) as intermediate asymptotic estimates for non-symmetric problems, and (3) in forecasting experimental results. To date, the vast majority of work pertaining to similarity solutions of the Euler equations has been accomplished in the context of the ideal gas EOS; the case where the material is arbitrary is less well-understood. In this work, we classify using Lie-group analysis those materials which admit similarity solutions. We also indicate how such solutions may be found for a variety of materials of interest, including those characterized by particular forms of the Gruneisen EOS.

1Graduate Student Department of Mathematics, UCLA

5:29PM E4.00004 Effects of Segmented Slot Blowing at the Leading Edge of a Finite Span Cavity in Supersonic Flow. BENJAMIN GEORGE, LAWRENCE UKEILEY, Univ of Florida - Gainesville, LOUIS CATTAFESTA, KUNIHIKO TAIRA, Florida State University — In this investigation, the effects of employing segmented slot blowing at the leading edge of a finite span cavity in Mach 1.4 flow are studied. The rectangular cavity under consideration has a length to depth ratio of 6 and width to depth ratio of 2 with an approaching turbulent boundary layer. Qualitative surface flow visualization results reveal changes in the flow characteristics due to the introduction of the sidewalls and multiple slot blowing configurations, as has been previously shown. Quantitatively, unsteady surface pressure measurements and particle image velocimetry (PIV) were utilized to characterize the mechanisms for suppressing surface pressure fluctuations in a three-dimensional flow field. Joint time-frequency analysis using wavelet transformations highlight changes in the tonal and broadband surface pressure fluctuations as a function of time with the different slot configurations. PIV data results from the baseline finite span case were compared with the slot blowing cases to illustrate their effects on the mean flow field properties in the shear layer and recirculation region. Finally, the finite span cavity experimental results are compared with previously acquired data for the full span cavity case to gain some insight into the flow field modifications as the cavity span is altered.
of such states bifurcating from the fold. These correspond to states with $n$ near the extinction region the burn front breaks up into structures that are localized in the direction along the front, with multiple branches. Temperature states are connected by an S-shaped branch with a fold at low Damköhler number below which extinction takes place. Various bifurcations generate states in which the hotspots are non-identical and separated by unequal distances. All these states are present in the same parameter region corresponding to the negative temperature conditions.

A viscous shear layer may form behind a non-bifurcating reflected shock wave as well and may affect chemical reactions. Under certain conditions; however, a viscous shear layer may form behind a non-bifurcating reflected shock wave as well and may affect chemical kinetics and ignition of certain fuels. The focus of this talk is on the development of the viscous shear layer and the coupling to the ignition in the regime corresponding to the negative temperature conditions.

Session E5 Combustion II 104 - Amy Lang, University of Alabama

4:50PM E5.00001 Localized structures in gaseous combustion , EDGAR KNÖBLOCH, University of California at Berkeley, DAVID LO JACONO, ALAIN BERGEON, IMFT, Université de Toulouse, UPS-INP — We consider a flame between a pair of porous walls at $x = \pm 1$ that allow fuel and oxidizer to diffuse into the burn region from opposite sides. The burn process is described by a binary one-step process of Arrhenius type. The heat released is redistributed via radiation. Convection is ignored. In 1D the low and high temperature states are connected by an S-shaped branch with a fold at low Damköhler number below which extinction takes place. Various instabilities occur on the upper (flame) branch leading to different time-dependent but 1D flames. In 2D the situation is dramatically modified: near the extinction region the burn front breaks up into structures that are localized in the direction along the front, with multiple branches of such states bifurcating from the fold. These correspond to states with $n = 1, 2, \ldots$ identical and equispaced hotspots. Further bifurcations generate states in which the hotspots are non-identical and separated by unequal distances. All these states are present in the same parameter interval, implying great sensitivity of the system to initial conditions.

5:03PM E5.00002 ABSTRACT WITHDRAWN –

5:16PM E5.00003 Experimental and Numerical Investigation of Vortical Structures in Lean Premixed Swirl-Stabilized Combustion , SOUFIEN TAAMALLAH, NADIM CHAKROUN, SANTOSH SHANBHOGUE, GAURAV KEWLANI, AHMED GHONIEM, Massachusetts Institute of Technology — A combined experimental and LES investigation is performed to identify the origin of major flow dynamics and vortical structures in a model gas turbine’s swirl-stabilized turbulent combustor. Swirling flows in combustion lead to the formation of complex flow dynamics and vortical structures that can interact with flames and influence its stabilization. Our experimental results for non-reacting flow show the existence of large scale precession motion. The precessing vortex core (PVC) dynamics disappears with combustion but only above a threshold of equivalence ratio. In addition, large scale vortices along the inner shear layer (ISL) are observed. These structures interact with the ISL stabilized flame and contribute to its wrinkling. Next, the LES setup is validated against the flow field’s low-order statistics and point temperature measurement in relevant areas of the chamber. Finally, we show that LES is capable of predicting the precession motion as well as the ISL vortices in the reacting case: we find that ISL vortices originate from a vortex core that is formed right downstream of the swirler’s centerbody. The vortex core has a conical spiral shape resembling a corkscrew that interacts - as it winds out - with the flame when it reaches the ISL.

5:29PM E5.00004 3-Dimensional Aerospike Nozzle Design¹, BENJAMIN STEVENS, Rose-Hulman Institute of Technology, RICHARD BRANAM, University of Alabama — Our research has developed a computational tool that can characterize the performance of an aerospike nozzle. Performance characteristics that must be analyzed include thrust, nozzle weight, and specific impulse. The program employs an iterative method of characteristics algorithm to solve for the 3-dimensional flow field around a specified aerospike nozzle geometry, and uses the results to compute a better geometry. Compared to a conical aerospike nozzle, where the radius decreases linearly along the nozzle, the aerospike optimal design offers extremely high performance. The optimal design features a radius that decreases very quickly initially, but becomes more gradual along the axial direction. This design increases specific impulse, increases thrust, and decreases weight, giving the aerospike the potential to lower launch costs significantly.

¹This research was funded by NSF REU Site Award 13588197

5:42PM E5.00005 High-Energy X-ray Absorption Diagnostics as an Experimental Combustion Technique¹, JARED DUNNMON, SADAF SOBHANI, WALDO HINSHAW, REBECCA FAHRIG, MATTHIAS IHME, Stanford University — X-ray diagnostics such as X-ray Computed Tomography (XCT) have recently been utilized for measurement of scalar concentration fields in gas-phase flow phenomena. In this study, we apply high-energy X-ray absorption techniques to visualize a laboratory-scale flame via fluoroscopic measurements by using krypton as a radiodense tracer media. Advantages of X-ray absorption diagnostics in a combustion context, including application to optically inaccessible environments and lack of ambient photon interference, are demonstrated. Analysis methods and metrics for extracting physical insights from these data are presented. The accuracy of the diagnostic is assessed via comparison to known results from canonical flame configurations, and the potential for further applications is discussed.

¹Support from the NDSEG fellowship, Bosch, and NASA are gratefully acknowledged.

5:55PM E5.00006 Electric field effects on droplet burning , ADITYA PATYAL, University of Illinois at Urbana-Champaign, DIMITRIOS KYRTIS, Khalifa University, MOSHE MATALON, University of Illinois at Urbana-Champaign — The effects of an externally applied electric field are studied on the burning characteristics of a spherically symmetric fuel drop including the structure, mass burning rate and extinction characteristics of the diffusion flame. A reduced three-step chemical kinetic mechanism that reflects the chemic-ionization process for general hydrocarbon fuels has been proposed to capture the production and destruction of ions inside the flame zone. Due to the imposed symmetry, the effect of the ionic wind is simply to modify the pressure field. Our study thus focuses exclusively on the effects of Ohmic heating and kinetic effects on the burning process. Two distinguished limits of weak and strong field are identified, highlighting the relative strength of the internal charge barrier compared to the externally applied field, and numerically simulated. For both limits, significantly different charged species distributions are observed. An increase in the mass burning rate is noticed with increasing field in either limit with negligible change in the flame temperature. Increasing external voltages pushes the flame away from the droplet and causes a strengthening of the flame with a reduction in the extinction Damkohler number.

Session E6 Reacting Flows: High Speed 105 - Seong-kyun Im, Worcester Polytechnic Institute

Sunday, November 22, 2015 4:50PM - 6:08PM
Challenging test cases that involve both free-streaming and optically thick regions will be presented. Reacting Flows universally-acceptable flamelet procedure for supersonic combustion.

We try to find some justifications for different scaling laws, with the hope of coming up with a more library. This scaling assumption is quite simple and will therefore be very attractive if it has a sound theoretical basis and it works for a large pressure in the finite Mach number flow field relative to the usually atmospheric static pressure field used in the generation of the flamelet. Supersonic combustion, the role ascribed to pressure has not been very convincing. That is, the reaction rate is often scaled on the square of the time-dependence of the energy input and the conversion from fuel to products. To that end, we are developing a procedure optimizing chemical species to be practical for realistic scenarios. The objective of our work is to create the simplest model possible that can reproduce affordable model of the chemical and diffusive properties. Full detailed chemical models of these systems contain too many reactions and thermal nonequilibrium effects are derived from the rovibrational state-specific reaction and scattering rates associated with the chemical mechanism. For continuum-scale applications, thermal nonequilibrium effects are derived from the rovibrational state-specific reaction and scattering rates associated with the chemical mechanism. 

Challenging test cases that involve both free-streaming and optically thick regions will be presented. 

\[ \text{FP}_N \] approximations to the RTE have been developed. The \[ \text{FP}_N \] equations are hyperbolic and, as a result, can be solved using algorithms that are similar Godunovs method for compressible fluid flow. The \[ \text{FP}_N \] model is also valid in optically thick and thin situations provided that the order, \( N \), is high enough. We show that the \[ \text{FP}_N \] equations are a promising alternative to traditional RTE approximations.

Challenging test cases that involve both free-streaming and optically thick regions will be presented.

This work is supported by DOE/NNSA and AFOSR.

High-speed reactive flow applications. Some scenarios, such as dust explosions in coal mines, can have regions that are nearly transparent and other regions with high dust concentration that are optically thick. Most approximations to the radiative transfer equation (RTE) are not valid in both limits simultaneously. Issues also arise when solving approximations to the RTE that can often require the solution of elliptic equations. Many compressible hydrodynamic codes use explicit time-marching and block-structured adaptive-mesh-refinement algorithms. Adapting these codes to solve elliptic equations is not always straightforward. Recently, filtered spherical harmonics (\[ \text{FP}_N \]) approximations of the RTE have been developed. The \[ \text{FP}_N \] equations are hyperbolic and, as a result, can be solved using algorithms that are similar Godunovs method for compressible fluid flow. The \[ \text{FP}_N \] model is also valid in optically thick and thin situations provided that the order, \( N \), is high enough. We show that the \[ \text{FP}_N \] equations are a promising alternative to traditional RTE approximations.

Challenging test cases that involve both free-streaming and optically thick regions will be presented.

Funded by AFOSR FA9550-12-1-0460

In high-speed reactive flows in scramjets, thermal nonequilibrium is introduced in the flow via shock waves. Though rotational and translational energy modes relax back to equilibrium quickly, vibrational relaxation is comparable to the bulk mixing and reaction timescales. The discrepancy between vibration and rotation/translation energy distributions can dramatically alter on the initiation of the fuel oxidation process. For continuum-scale applications, thermal nonequilibrium effects are derived from the rovibrational state-specific reaction and scattering rates associated with the chemical mechanism. In this work, the state-specific reaction rates are calculated for the chain branching reactions in the hydrogen combustion mechanism using a quasi-classical trajectory (QCT) framework. The state-specific rates are incorporated into a multiple temperature continuum-scale model whereby each species is characterized by a Boltzmann distribution parametrized by its own vibrational temperature. The flame ignition rates are implemented in a CFD code to simulate a reactive coflow.

This work is supported by DOE/NNSA and AFOSR.

In search of reaction rate scaling law for supersonic combustion.

In high-speed reactive flows in scramjets, thermal nonequilibrium is introduced in the flow via shock waves. Though rotational and translational energy modes relax back to equilibrium quickly, vibrational relaxation is comparable to the bulk mixing and reaction timescales. The discrepancy between vibration and rotation/translation energy distributions can dramatically alter on the initiation of the fuel oxidation process. For continuum-scale applications, thermal nonequilibrium effects are derived from the rovibrational state-specific reaction and scattering rates associated with the chemical mechanism. In this work, the state-specific reaction rates are calculated for the chain branching reactions in the hydrogen combustion mechanism using a quasi-classical trajectory (QCT) framework. The state-specific rates are incorporated into a multiple temperature continuum-scale model whereby each species is characterized by a Boltzmann distribution parametrized by its own vibrational temperature. The flame ignition rates are implemented in a CFD code to simulate a reactive coflow.

Funded by AFOSR FA9550-12-1-0460

Optimizing simplified one-step chemical models for high speed reacting flows.

The scaling assumption is quite simple and will therefore be very attractive if it has a sound theoretical basis and it works for a large selection of high-speed combustion flows. We try to find some justifications for different scaling laws, with the hope of coming up with a more universally-acceptable flamelet procedure for supersonic combustion.

Optimizing simplified one-step chemical models for high speed reacting flows.

Full detailed chemical models of these systems contain too many reactions and chemical species to be practical for realistic scenarios. The objective of our work is to create the simplest model possible that can reproduce the time-dependence of the energy input and the conversion from fuel to products. To that end, we are developing a procedure optimizing parameters in the most simplified “one-step” model. An important requirement of this model is that it reproduces known flame and detonation properties. Multidimensional numerical simulations using the new model are compared to deflagration-to-detonation experiments in channels containing ethylene and oxygen.

Sunday, November 22, 2015 4:50PM - 5:55PM —
Session E7 High Reynolds Number Experiments 107 - Joseph Katz, Johns Hopkins University
c. FLUIDICS: Turbomachinery

- Session E8 CFD: Turbomachinery
- 108 - Reza Sheikhi, Northeastern University

Sunday, November 22, 2015 4:50PM - 6:08PM
4:50PM E8.00001 Computational study of a High Pressure Turbine Nozzle/Blade Interaction. JAMES KOPRIVA, GREGORY LASKOWSKI, GE Aviation, REZA SHEIKHI, Northeastern University — A downstream high pressure turbine blade has been designed for this study to be coupled with the upstream uncooled nozzle of Arts and Rouvroit [1992]. The computational domain is first held to a pitch-line section that includes no centrifugal forces (linear sliding-mesh). The stage geometry is intended to study the fundamental nozzle/blade interaction in a computationally cost efficient manner. Blade/Nozzle count of 2:1 is designed to maintain computational periodic boundary conditions for the coupled problem. Next the geometry is extended to a fully 3D domain with endwalls to understand the impact of secondary flow structures. A set of systematic computational studies are presented to understand the impact of turbulence on the nozzle and downstream blade boundary layer development, resulting heat transfer, and downstream wake mixing in the absence of cooling. Doing so will provide a much better understanding of stage mixing losses and wall heat transfer which, in turn, can allow for improved engine performance. Computational studies are performed using WALE (Wale Adapted Local Eddy), IDDES (Improved Delayed Detached Eddy Simulation), SST (Shear Stress Transport) models in Fluent.

5:03PM E8.00002 Impact of tip-gap size and periodicity on turbulent transition. ALEXEJ POGORELOV, MATTHIAS MEINKE, WOLFGANG SCHROEDER, RWTH - Aachen — Large-Eddy Simulations of the flow field in an axial fan are performed at a Reynolds number of 936,000 based on the diameter and the rotational speed of the casing wall. A finite-volume flow solver based on a conservative Cartesian cut-cell method is used to solve the unsteady compressible Navier-Stokes equations. Computations are performed at a flow rate coefficient of 0.165 and a tip-gap size of s/D = 0.01, for a 72 degrees fan section resolving only one out of five blades and a full fan resolving all five blades to investigate the impact of the periodic boundary condition. Furthermore, a grid convergence study is performed using four computational grids. Results of the flow field are analyzed for the computational grid with 1 billion cells. An interaction of the turbulent wake, generated by the tip-gap vortex, with the downstream blade, is observed, which leads to a cyclic transition with high pressure fluctuations on the suction side of the blade. Two dominant frequencies are identified which perfectly match with the characteristic frequencies in the experimental sound power level such that their physical origin is explained. A variation of the tip-gap size alters the transition on the suction side, i.e., no cyclic transition is observed.

5:16PM E8.00003 Application of dynamic slip wall modeling to a turbine nozzle guide vanes. SANJEEB BOSE, Cascade Technologies, CHAITANYA TALNIKAR, PATRICK BLONIGAN, QIQI WANG, MIT — Resolution of near-wall turbulent structures is computational prohibitive necessitating the need for wall-modeled large-eddy simulation approaches. Standard wall models are often based on assumptions of equilibrium boundary layers, which do not necessarily account for the dissimilarity of the momentum and thermal boundary layers. We investigate the use of the dynamic slip wall boundary condition (Bose and Moin, 2014) for the prediction of surface heat transfer on a turbine nozzle guide vane (Arts and de Rouvroit, 1992). The heat transfer coefficient is well predicted by the slip wall model, including capturing the transition to turbulence. The sensitivity of the heat transfer coefficient to the incident turbulence intensity will additionally be discussed. Lastly, the behavior of the thermal and momentum slip lengths will be contrasted between regions where the strong Reynolds analogy is invalid (near transition on the suction side) and an isothermal, zero pressure gradient flat plate boundary layer (Wu and Moin, 2010).

5:29PM E8.00004 Turbulence Model Evaluation on a High Pressure Turbine Stage 1 Vane. MICHAL OSUSKY, SARA ROSTAMI, GE Global Research, AAMIR SHABBIR, GE Aviation — The accuracy of turbulence modeling depends heavily on the choice of turbulence model. Many turbulence models are only valid for a narrow range of flow regimes, and can produce numerically converged, but physically inaccurate results when applied outside the scope of their intended use. Additionally, the underlying modeling assumptions, such as the linear Boussinesq approximation, impacts the evolution of turbulence in the flow field. As part of the current work, we will study the impact of using various commonly used RANS turbulence models, such as k-omega, BSL, and SST, with and without transition modeling, on the flow field of realistic engine geometries. Additionally, advanced, non-linear turbulence models, such as the Explicit Algebraic Reynolds Stress Model (EARS), will also be studied for their potential benefits in capturing additional physics in the simulation. Presenting here is that the EARS model has a significant impact on the location on laminar to turbulent transition, versus the SST model. All computational results will be compared against detailed experimental data.

5:42PM E8.00005 Compressible DNS of transitional and turbulent flow in a low pressure turbine cascade. RAJESH RANJAN, SURESH DESHPANDE, RODDAM NARASIMHA, JNCSAR — Direct numerical simulation (DNS) of flow in a low pressure turbine cascade at high incidence is performed using a new in-house code ANUROOP. This code solves compressible Navier-Stokes equations in conservative form using finite volume technique and uses kinetic-energy consistent scheme for the flux calculations. ANUROOP is capable of handling flow past complex geometries using hybrid grid approach (separate grid topologies for the boundary layer and rest of the blade passage). This approach offers much more control in mesh spacing and distribution compared to elliptic grid technique, which is used in many previous studies. Also, in contrast to previous studies, focus of the current work is mainly on the boundary layer flow. The flow remains laminar on the pressure side of the blade, but separates in the aft region of the suction side leading to transition. Separation bubbles formed at this region are transient in nature and we notice multiple bubbles merging and breaking in time. In the mean flow however, only one bubble is seen. Velocity profiles very near to the leading edge of the suction side suggest strong curvature effect. Higher-order boundary layer theory that includes effect of curvature is found to be necessary to characterize the flow in this region. Also, the grid convergence study reveals interesting aspects of numerics vital for accurate simulation of this kind of complex flows.

5:55PM E8.00006 Unsteady adjoint of a gas turbine inlet guide vane. CHAITANYA TALNIKAR, QIQI WANG, Massachusetts Inst of Tech-MIT — Unsteady fluid flow simulations like large eddy simulation have been shown to be crucial in accurately predicting heat transfer in turbomachinery applications like transonic flow over an inlet guide vane. To compute sensitivities of aerothermal objectives for a vane with respect to design parameters an unsteady adjoint is required. In this talk we present unsteady adjoint solutions for a vane from VKI using pressure loss and heat transfer over the vane surface as the objectives. The boundary layer on the suction side near the trailing edge of the vane is turbulent and this poses a challenge for an adjoint solver. The chaotic dynamics cause the adjoint solution to diverge exponentially to infinity from that region when simulated backwards in time. The prospect of adding artificial viscosity to the adjoint equations to dampen the adjoint fields is investigated. Results for the vane from simulations performed on the Titan supercomputer will be shown and the effect of the additional viscosity on the accuracy of the sensitivities will be discussed.

Sunday, November 22, 2015 4:50PM - 6:08PM
Session E9 CFD: Computational Methods and Modeling of Multiphase Flows II
109 - Mathieu Lepilliez, University of Toulouse
Overall framework is demonstrated via an array of test cases. Numerical simulations of the interaction between ocean waves and WECs are in a VOF-PLIC context. A consistent mass and momentum transport approach enables simulations at high density ratios. The accuracy of the using the numerically stable fictitious domain method. A novel three-phase interface reconstruction algorithm is used to resolve three phases the structure, which cannot be captured in many potential flow solvers commonly used for WEC simulations. The full Navier-Stokes equations present an MPI-parallel GPU-accelerated computational framework for studying the interaction between ocean waves and wave energy converters (WECs). The computational framework captures the viscous effects, nonlinear fluid-structure interaction (FSI), and breaking of waves around the tank. One can observe different steps during the maneuvers under microgravity: the first part is dominated by accelerations and volume forces, which flatten the bubble on the hydrophobic tank wall. When the forcing stops, the bubble bounces back, generating sloshing by moving under the influence of inertia and capillary effects. Finally viscous effects damp the sloshing by dissipating the kinetic energy of the bubble. The results are compared to actual in-flight data for different typical maneuvers on forces and torques, allowing us to characterize the period and damping of the sloshing.

**5:03PM E9.00002 Structures in the Oscillatory regime of RLDCC flow**

5:03PM E9.00002 Structures in the Oscillatory regime of RLDCC flow1. NAGANGUDY PANCHAPAKESAN, Department of Aerospace Engineering, Indian Institute of Technology, Madras — Rotating lid driven cubical cavity flow (RLDCC flow) is studied with a view to test structure eduction algorithms. OpenFoam software was used to simulate the RLDCC flow at Reynolds numbers higher than the critical Reynolds number for this geometry. Vortex bubble and other characteristic structures were observed in these simulations. The vector fields of the simulations were further analyzed with LCS and other methodologies to elucidate the structures. The structures were compared with level sets of different dynamical variables. The ability of these algorithms to present a coherent representation of the time evolution and unsteady dynamics of the bubble and other structures is evaluated.

1CNES/ Airbus Defence & Space funding.

**5:16PM E9.00003 Aref’s chaotic orbits tracked by a general ellipsoid using 3D numerical simulations**

5:16PM E9.00003 Aref’s chaotic orbits tracked by a general ellipsoid using 3D numerical simulations, PEI SHUI, School of Engineering, The University of Edinburgh, UK, STÉPHANE POPINET, Institut Jean le Rond d’Alembert Université Pierre et Marie Curie, France, RAMA GOVINDARAJAN, TIFR Centre for Interdisciplinary Sciences, Hyderabad, India, PRASHANT VALLURI, School of Engineering, The University of Edinburgh, UK — The motion of an ellipsoidal solid in an ideal fluid has been shown to be chaotic (Aref, 1993) under the limit of non-integrability of Kirchhoff’s equations (Kozlov & Oniscenko, 1982). On the other hand, the particle could stop moving when the damping viscous force is strong enough. We present numerical evidence using our in-house immersed solid solver for 3D chaotic motion of a general ellipsoidal solid and suggest criteria for triggering such motion. Our immersed solid solver functions under the framework of the Gerris flow package of Popinet et al. (2003). This solver, the Gerris Immersed Solid Solver (GISS), resolves 6 degree-of-freedom motion of immersed solids with arbitrary geometry and number. We validate our results against the solution of Kirchhoff equations. The study also shows that the translational/ rotational energy ratio plays the key role on the motion pattern, while the particle geometry and density ratio between the solid and fluid also have some influence on the chaotic behaviour. Along with several other benchmark cases for viscous flows, we propose prediction of chaotic Aref’s orbits as a key benchmark test case for immersed boundary/solid solvers.

**5:29PM E9.00004 Modeling electrokinetic flow by Lagrangian particle-based method**

5:29PM E9.00004 Modeling electrokinetic flow by Lagrangian particle-based method, WEXNIAO PAN, Pacific Northwest National Laboratory, KYUNCJOO KIM, MAURO PEREGO, Sandia National Laboratory, ALEXANDRE TARTAKOVSKY, Pacific Northwest National Laboratory, MIKE PARKS, Sandia National Laboratory — This work focuses on mathematical models and numerical schemes based on Lagrangian particle-based method that can effectively capture mesoscale multiphysics (hydrodynamics, electrostatics, and advection-diffusion) associated in applications of micro-/nano-transport and technology. The order of accuracy is significantly improved for particle-based method with the presented implicit consistent numerical scheme. Specifically, we show simulation results on electrokinetic flows and microfluidic mixing processes in micro-/nano-channel and through semi-permeable porous structures.

**5:42PM E9.00005 Numerical study on influence of electric Reynolds and Peclet number’s on electrohydrodynamic assisted atomization**

5:42PM E9.00005 Numerical study on influence of electric Reynolds and Peclet number’s on electrohydrodynamic assisted atomization, PATRICK SHEEHY1, MARK OWKES2, Montana State Univ — Electrohydrodynamics (EH) has shown great potential for enhancing the atomization of liquid flows by decreasing droplet size and improving dispersion. For many fluid flows relevant to engineering, such as liquid fuel injection, two important fluid properties are the electric Reynolds and Peclet numbers (Re and Pe), which control how fast electric charges relax to the gas-liquid interface and the thickness of electric charge boundary layers. The effect of the numbers is not well understood due to the difficulty of measuring electric charges experimentally. Additionally, predicting the impact of electric charge distribution on atomization is difficult. For example, a smaller electric Re number causes a weaker electric field, higher charge concentrations at interface, and a non-trivial distribution of the Coulomb force. This work uses a numerical approach to simulate a two-phase EHJ jet in order to examine the effect of these two non-dimensional numbers on atomization quality. The approach employed is second-order, conservative, and consistently transports the discontinuous electric charge density, momentum, and phase interface. Several three-dimensional test cases are simulated using this process for a range of Re and Pe numbers and comparisons are made.

1ME Master’s Student
2Graduate Advisor

**5:55PM E9.00006 A 3D MPI-Parallel GPU-accelerated framework for simulating ocean wave energy converters**

5:55PM E9.00006 A 3D MPI-Parallel GPU-accelerated framework for simulating ocean wave energy converters1, ASHISH PATHAK, MEHDI RAESSI, University of Massachusetts Dartmouth — We present an MPI-parallel GPU-accelerated computational framework for studying the interaction between ocean waves and wave energy converters (WECs). The computational framework captures the viscous effects, nonlinear fluid-structure interaction (FSI), and breaking of waves around the structure, which cannot be captured in many potential flow solvers commonly used for WEC simulations. The full Navier-Stokes equations are solved using the two-step projection method, which is accelerated by porting the pressure Poisson equation to GPUs. The FSI is captured using the numerically stable fictitious domain method. A novel three-phase interface reconstruction algorithm is used to resolve three phases in a VOF-PLIC context. A consistent mass and momentum transport approach enables simulations at high density ratios. The accuracy of the overall framework is demonstrated via an array of test cases. Numerical simulations of the interaction between ocean waves and WECs are presented.

1Funding from the National Science Foundation CBET - 1236462 grant is gratefully acknowledged.
Rearrangement dynamics of concentrated emulsions in a tapered micro-channel

A simple and low-cost 3D-printed emulsion generator

Continuous Microfluidic Fabrication of Synthetic Asymmetric Vesicles for Membrane Biology Studies

Changing Emulsion Dynamics with Heterogeneous Surface Wettability

A scalable platform for functional emulsions

Fluid entrainment in confined colloid-polymer mixtures
4:50PM E11.00001 Measurement of the near-wall velocity profile for a nanofluid flow inside a microchannel1, ANOOP KANJIRAKAT, REZA SADR, Texas A&M University at Qatar — Hydrodynamics and anomalous heat transfer enhancements have been reported in the past for colloidal suspensions of nano-sized particles dispersed in a fluid (nanofluids). However, such augmentations may manifest itself by study of fluid flow characteristics near in the wall region. Present experimental study reports near-wall velocity profile for nanofluids (silicon dioxide nanoparticles in water) measured inside a microchannel. An objective-based nano-Particle Image Velocimetry (nPIV) technique is used to measure fluid velocity within three visible depths, O(100nm), from the wall. The near-wall fluid velocity profile is estimated after implementing the required corrections for optical properties and effects caused by hindered Brownian motion, wall-particle interactions, and non-uniform exponential illumination on the measurement technique. The fluid velocities of nanofluids at each of the three visible depths are observed to be higher than that of the base fluid resulting in a higher shear rate in this region. The relative increase in shear rates for nanofluids is believed to be the result of the near-wall shear-induced particle migration along with the Brownian motion of the nanoparticles.

1This research is funded by NPRP grant # 08-574-2-239 from the Qatar National Research Fund (a member of Qatar Foundation).

5:03PM E11.00002 The deposition of gold nanoparticles in MWCNT forests, FRANCISCO DE JONG, Hamburg University of Technology (Hamburg, Germany), ADELINE BUFFET, German Electron Synchrotron (Hamburg, Germany), MICHAEL SCHLUETER, Hamburg University of Technology — The deposition, i.e. transport and attachment, of small-sized particles is a basic process, on which many applications are based. The innumerable applications range from biology and medicine to engineering. Due to their promising mechanical properties multi-walled carbon nanotubes (MWCNTs) have gained increasing popularity in the past decade. A large number of dense packed vertically aligned MWCNTs form a so-called MWCNT forest. In our study we functionalized the MWCNT forest to filter gold nanoparticles from a colloidal suspension. An experimental investigation was carried out in which the particle deposition kinetics was locally determined with small-angle X-ray scattering (SAXS). Furthermore, inductively coupled plasma atomic emission spectroscopy (ICP-AES) was used to verify the local observations. It was concluded that both, SAXS and ICP-AES investigations shows very good agreement. Furthermore, an analytical deposition model was developed based on the DLVO-theory. The experimental and theoretical investigation presented here give insight in the deposition kinetics within a MWCNT forest. The results open up pathways to optimize MWCNT forests for filtering purposes.

5:16PM E11.00003 Experimental demonstration of scaling behavior for ionic transport and its fluctuations in individual carbon nanotube, LYDERIC BOCUET, ELEONORA SECCHI, ANTOINE NIGUES, ALESSANDRO SIRIA, Department of Physics, Ecole Normale Superieure, Paris — We perform an experimental study of ionic transport and current fluctuations inside individual Carbon Nanotubes (CNT) with a size ranging from 40 down to 7 nanometers in radius. The conductance exhibits a power law behavior dependence on the salinity, with an exponent close to 1/3. This is in contrast to Boron-Nitride nanotubes which exhibits a constant surface conductance. This scaling behavior is rationalized in terms of a model accounting for hydroxide adsorption at the (hydrophobic) carbon surface. This predicts a density dependent surface charge with an exponent 1/3 in full agreement with the experimental observations. Then we measure the low frequency noise of the ionic current in single CNTs. The noise exhibits a robust 1/f characteristic, with an amplitude which scales proportionaly to the surface charge measured independently. Data for the various CNT at a given pH do collapse on a master curve. This behavior is rationalized in terms of the fluctuations of the surface charge based on the adsorption behavior. This suggests that the low frequency noise takes its origin in the process occuring at the surface of the carbon nanotube.

5:29PM E11.00004 The near wall TIRFM measurement of nano-tracer’s statistical intensity distribution (SID) and determining the base intensity I0, XU ZHENG, LNM, Institute of Mechanics, CAS, LNM TEAM — The total internal reflection fluorescence microscopy (TIRFM) is an evanescent-wave-based technique for measuring nanoparticle dynamics very close to wall. The intensity of the evanescent wave decays exponentially (i.e. I(z)=I0*exp(-z/zp)), which can provide information of the tracer particle position not just parallel but also normal to wall. However, considering the z information is encoded in tracer intensity, it is critical to determine the base intensity I0. In this study, we will first establish a model to describe the statistical intensity distribution (SID) of the nano-tracers observed in the evanescent field inspired by the works of Huang et al .. A different function of particle-wall interaction and a term of the influence of the objective focal plane thickness are introduced in the present SID method. Then, TIRFM experiments are performed to measure the histogram of SID. The experimental histogram of SID is then fitted by the theoretical curve to determine I0 which is the only one fit parameter. By near wall velocity measurement, we will show that the SID method has a very high precision in determining I0 and the vertical z position of every nanotracer. Further tests show that the PDF of nano-tracers can reveal more information about how nanoparticles interact with the charged solid wall. This provides a promising method to detect the physical properties near interface.

5:42PM E11.00005 Nanodroplet Depinning Dynamics of from Nanoparticles1, FONG YEW LEONG, A*STAR Institute of High Performance Computing, LIU QI, ZAINUL AABDIN, National University of Singapore, TRAN SI BUI QUANG, A*STAR Institute of High Performance Computing, UTFUR MIRSAIDOV, National University of Singapore, NRF CRP COLLABORATION — Nanoscale defects on substrate affect the sliding motion of water nanodroplets. Using in situ TEM imaging, we visualized the depinning dynamics of water nanodroplets from gold nanoparticles on a flat SiNx surfaces. Our observations showed that nanoscale pinning effects of the gold nanoparticle opposes the lateral forces, resulting in stretching, even breakup, of the water nanodroplet. Using continuum long wave theory, we modelled the dynamics of a nanodroplet depinning from a nanoparticle of comparable length scales, and the model results are consistent with experimental findings. In particular, the critical depinning force for a ten-nanometer particle is found to be on the order of a nano-Newton, and the apparent viscosity of interfacial water is inferred to be several orders of magnitude greater than bulk values. Our findings have important implications on surface cleaning at the nanoscale.

1This work is supported in part by the Singapore National Research Foundation under CRP Award No. NRF-CRP9-2011-04.
Yüatak Kazoe, Takuya Ugaejin, Yutaka Kazoe, Takuya Ugaejin, Ryoichi Ohta, Kazuma Mawatari, Takehiko Kitamori, The University of Tokyo, THE UNIVERSITY OF TOKYO TEAM — Micro chemical systems have realized high-throughput analysis in ultra small volumes. Our group has established unit operations such as extraction, separation and reaction, and a concept of integration of chemical processes using parallel multi-phase flows in microchannels. Recently, the research field has been extended to 10-1000 nm space (extended-nanospace). Exploiting extended-nanospace, we developed ultra high performance chemical operations such as AL-chromatography and single molecule immunoassay. However, formation of parallel multi-phase flow in nanochannels has been difficult. The challenge is to control liquid-liquid/gas-liquid interfaces in 100 nm-scale. For this purpose, this study developed a partial surface modification method of nanochannel and verified formation of parallel two-phase flow. We achieved partial hydrophobic modification using focused ion beam (FIB). Using this method, formation of parallel water/dodecane two-phase flow in a nanochannel of 1500 nm width and 890 nm depth was succeeded. Solvent extraction of lipid, which is a basic separation in bioanalysis, was achieved in 25 fL volume much smaller than single cell. This study will greatly contribute to develop novel nanofluidic devices for chemical analysis and chemical synthesis.

This work was supported by Japan Science and Technology Agency, Core Research for Evolutional Science and Technology.

**Sunday, November 22, 2015 4:50PM - 6:08PM**

**Session E12 Non-Newtonian Flows: Viscoplasticity**

200 - Neil Balmforth, University of British Columbia

**4:50PM E12.00001 Viscoplastic flow in a Hele-Shaw cell**

Neil Balmforth, University of British Columbia, Duncan Hewitt, University of Cambridge — A theoretical study is presented of the flow of viscoplastic fluid through a Hele-Shaw cell that contains various kinds of obstructions. Circular and elliptical blockages of the cell are considered together with step-wise contractions or expansions in slot width, all within the simplifying approximation of a narrow gap. Specific attention is paid to the flow patterns that develop around the obstacles, particularly any stagnant plugged regions, and the asymptotic limits of relatively small or large yield stress. Periodic arrays of circular contractions or expansions are studied to explore the interference between obstructions. Finally, viscoplastic flow through a cell with randomly roughened walls is examined, and it is shown that constructive interferences of local contractions and expansions leads to a pronounced channelization of the flow. An optimization algorithm based on minimisation of the pressure drop is derived to construct the path of the channels in the limit of relatively large yield stress or, equivalently, relatively slow flow.

**5:03PM E12.00002 Channelization of viscoplastic flow in a rough Hele-Shaw cell**

Duncan Hewitt, DAMTP, University of Cambridge, Neil Balmforth, Department of Mathematics, University of British Columbia — The flow of viscoplastic fluid down slender conduits or through porous media has application in a range of industrial and geophysical settings, from the plumbing of mud volcanoes to the transport of proppant slurries in hydraulic fracturing. The yield stress can cause the fluid locally to clog up, which can significantly affect the flow patterns. Flow of a viscoplastic fluid in a Hele-Shaw cell that has randomly "roughened" walls is investigated, both numerically and using analogue laboratory experiments. Fluid injected into the centre of the rough cell, which is initially full of the same fluid, show pronounced channelization: above a critical pressure drop (below which there is no flow and all the fluid is unyielded and stagnant), one or more thin conduits of yielded, flowing fluid develop. At larger pressure drops, more channels of yielded fluid develop. The quantity and width of the channels, and the value of the critical pressure drop, depend on the amplitude of the roughness of the walls of the cell. If this roughness is known, the locations of the first channels to flow and the corresponding pressure drop can be predicted by an optimization algorithm.

**5:16PM E12.00003 Yield stress fluid flow in model porous media**

Johan Paiola, Univ de Paris, Hugues Bodiguel, Laboratory of the Future (LoF), University of Bordeaux, 178, avenue du Dr Schweitzer F-33608 Pessac, France, Harold Auradou, Fluide Automatique et Systèmes Thermiques (FAST), Paris Sud University, 91405 Orsay Cedex, France — Yield stress fluids display interesting flow behavior due to their non-linear flow curve and some applications involve this flow in porous media. Predicting the flow behavior is thus of great interest but is further complicated by the complexity of the geometry and the interplay between the heterogeneities of the medium and the existence of a yield stress. Models developed in order to describe Darcy’s law assume a rheological law applied. Alternatively, micromechanical models predicts that the flow concentrates at low flow rates in preferential paths, which strongly depends on the details of the porous geometry. At this stage, rather few experiments are available at the scale of a few pores, and we propose in this work to study the flow of yield stress fluids in micromodels of porous media to address experimentally the existence and characteristic of these preferential paths. We use Carbopol as a model yield stress fluid. This fluid is injected into various model porous media. The main objective of our experiments is to map the fluid velocity field as a function of the global pressure drop applied. We develop a new experimental method where we can obtain simultaneously to measure local velocities at the scale of one channel (200 um) but on the entire porous geometry (5cm x 5cm).

**5:29PM E12.00004 The transverse mobility of yield-stress fluids in fibrous media**

Setareh Sahsavar, Gareth H. McKinley, MIT — The pressure-drop/flow-rate relationship for fluids that exhibit a yield stress and a shear dependent viscosity flowing through fibrous media is studied numerically. The Cauchy momentum equation along with the Bingham or Herschel-Bulkley constitutive equations are solved for flow transverse to a periodic array of fibers and systematic parametric studies are used to understand the individual roles of geometrical characteristics and fluid rheological properties. We develop a scaling model to predict the fluid mobility as a function of the medium porosity and the Bingham number. In addition, using this scaling model we estimate the width of the unyielded region between two adjacent fibers. Numerical computations are combined with the scaling model to obtain a criterion for the critical pressure gradient required to drive flow. Variations in the size of the yielded zones, the velocity profiles and the resulting stress fields are investigated for the limiting cases of (i) densely packed fiber arrays and (ii) very sparsely distributed fibers, and the hydrodynamic transition between these configurations is investigated. While this work focuses on the flow of inelastic fluids, the methodology can be extended to consider more complex rheology such as flow of elasto-visco-plastic fluids.
5:42PM E12.00005 Does Carbopol Elasticity affect its Yielding Dynamics? A study based on the Settling of a Particle in “Plastic” materials\textsuperscript{1}. DIMITRIS FRAGGEDAKIS, YIANNIS DIMAKOPOULOS, JOHN TSAMOPoulos, Univ of Patras — For several decades, Carbopol is assumed to be the ideal plastic material, exhibiting only yield phenomena without viscoelastic effects in yielded regions. Recently, it has been shown that when stresses do not overcome the yield criterion, it behaves as an ideal Hookean solid, Piau (2007). Also, experiments (Putz et al. (2006); Holtenberg et al. (2012)) reveal phenomena which can be attributed only to elastic properties of the fluidized region, such as the appearance of the so-called “negative wake,” Harlen (2002), downstream the sphere and the loss of fore-aft symmetry of the yield surface around a sedimenting particle. Our study is based on the sedimentation of a confined particle in materials which exhibit elastoviscoplastic behavior and proves that Carbopol cannot be considered as the ideal plastic material anymore. Moreover, when elasticity comes into play, the derived stoppage criterion for a sedimenting sphere by Beris et al. (1985) and experimentally confirmed by Tabuteau et al. (2007) is not satisfied, as a complex stress field is developed around the particle and fluidization near the rigid surface is favored. The existence of the yield surface near the sphere enhances the formation of shear layers, which are responsible for the formation of the negative wake, irrespectively of the position of the confinement in relation to the sphere.

\textsuperscript{1}GSRT Greece-Israel bilateral projects PHARMAMUDS \#3163.

5:55PM E12.00006 Insights on the local dynamics during transient flows of waxy crude oils\textsuperscript{1}. MICHELA GERI, Hatsopoulos Microfluids Laboratory - MIT, BRICE SAINT-MICHEL, Laboratoire de Physique, ENS de Lyon UMR 5672 - Université de Lyon, THIBAUT DIVOUX, CRPP, CNRS UPR8641 - Université de Bordeaux, SEBASTIEN MANNEVILLE, Laboratoire de Physique, ENS de Lyon UMR 5672 - Université de Lyon, GARETH H. MCKINLEY, Hatsopoulos Microfluids Laboratory - MIT — Waxy crude oils are mixtures of hydrocarbons and paraffin wax that behave as a Newtonian liquid at high temperatures and display solid-like properties below the solidification temperature of wax. In the latter case, waxy crude oils exhibit a yield stress and show a pronounced time-dependent behavior. By means of rheometry coupled to time-resolved ultrasonic velocimetry we investigate the local scenario associated with a series of decreasing and increasing ramps of shear rate composed of successive steps during which the stress is left to equilibrate for a fixed time. While being fully fluidized or ‘shear-melted’ at large shear rates, we observe that the sample experiences wall slip despite using rough boundary conditions and an arrested band grows inwards towards the rotor as the shear rate is decreased. This shear banding scenario arises from an underlying non-monotonic time-dependent response in the shear stress. As the shear rate is ramped back up to its initial value, the sample experiences a delayed yielding transition involving shear banding and wall slip over a range of shear rates that differs from the range observed on the decreasing branch. We finally discuss these results in the framework of a thixotropic elasto-visco-plastic model.

\textsuperscript{1}MIT-France MISTI Global Seed Funds

Sunday, November 22, 2015 4:50PM - 6:08PM
Session E13 Free Surface Flows III: Marangoni Flows 201 - Matthieu Roch, Universit Paris Diderot, CNRS

4:50PM E13.00001 Polygonal instability of Marangoni flows, MATTHIEU ROCHE, Laboratoire Matière et Systèmes Complexes, CNRS - Université Paris Diderot, MATTHIEU LABOUSSE, Laboratoire Matériaux et Phénomènes Quantiques, CNRS - Université Paris Diderot, BABAY EL HADJ MAIGA, LOC NYA, Laboratoire Matière et Systèmes Complexes, CNRS - Université Paris Diderot, SEBASTIEN LE ROUX, ISABELLE CANTAT, ARNAUD SAINT-JALMES, Institut de Physique de Rennes, CNRS - Université Rennes 1 — The transport of pepper grains floating at the surface of a bowl of water after the release of a drop of dishwashing liquid is a classical experiment to demonstrate the Marangoni effect, i.e. the flow of a liquid layer induced by interfacial tension gradients at its surface. In this case, the interfacial tension gradient results from a surfactant interfacial concentration gradient. Recently, we showed that continuous injection of an aqueous solution of hydrodurable surfactants at the surface of a cm-thick pure water layer induced finite-size Marangoni flows surrounded by a region characterized by the presence of several pairs of interfacial vortices or the vertices of polygons During this talk, I will show that we can understand the flow structure induced by these Marangoni flows, in particular their tendency to have polygonal shapes.

2M. Labousse and J. W. M. Bush, submitted

5:03PM E13.00002 Vibrational instabilities of a nonisothermal liquid layer with insoluble surfactant, ALEXANDER MIKISHEV, Sam Houston State Univ, ALEXANDER NEPOMNYASHCHY, Technion — We consider an infinite horizontal layer of an incompressible liquid, the deformable upper free surface is covered by insoluble surfactant. The layer is subjected to vertical harmonic oscillations with fixed amplitude and frequency, as well as to a transverse gradient of temperature. We suppose that the surface tension of upper boundary linearly depends on temperature and surfactant concentration. Two types of waves on the surface are possible. The first one is capillary-gravity waves (transverse waves) excited by the usual Faraday instability mechanism, under the influence of the interfacial elasticity. The second type of waves is related to compressions, dilatations of the surface. In this work we study the excitation of Marangoni waves by vibration and determine the existence conditions for each type of waves. The results are connected with our previous research on parametric excitation of Marangoni instability when the gradient of temperature is harmonically changed. The instability thresholds are calculated numerically using the Floquet method for disturbances with arbitrary wave numbers.

5:16PM E13.00003 The Effect of Non-Uniform Wetting Properties on Contact Line Dynamics\textsuperscript{1}, MORGANE GRIVEL, DAVID JEON, MORTEZA GHARIB, Caltech — Surfaces with non-uniform wetting properties have been shown to modify contact line dynamics and induce passive displacements of shallow flows. These surfaces are patterned with alternating hydrophobic and hydrophilic stripes of a certain width, spacing and orientation. A thin rectangular wall jet impinges on the surfaces and Fourier Transform Profilometry is used to reconstruct the 3D profile of the low to medium Reynolds number flows. Our previous work reported the development of intriguing roller structures at the contact line near hydrophobic-hydrophilic interfaces and the effect of varying the stripes' dimensions and orientation on these flows. Our present work extends the study to the effects of flow rate and plate inclination angle (with respect to the horizontal). The current work also studies air entrainment by the roller structures of the modified contact line. We will also discuss potential uses of this technique for modifying contact line dynamics and bow waves near surface-piercing bodies.

\textsuperscript{1}Work is funded by the Office of Naval Research (grant N00014-11-1-0031) and the National Science Foundation’s GRFP
5:29PM E13.00004 Enhancing Liquid Micro-volume Mixing with Wettability-Patterned Surfaces, JARED MORRISSETTE, PALLAB SINHA MAHAPATRA, University of Illinois at Chicago, RANJAN GANGULY, Jadavpur University, CONSTANTINE MEGARIDIS, University of Illinois at Chicago, UIC - MNFTL TEAM — Self-driven surface micromixers (SDSM) based on patterned wettability technology provide an elegant solution for low-cost point-of-care (POC) devices and lab-on-a-chip (LOC) applications. Our SDSMs are fabricated by strategically patterning three wettable wedge-shaped tracks onto a non-wettable surface. Current state-of-the-art micromixers require energy, however, our SDSMs utilize the inherent surface energy of liquids, coupled with wettability contrast to efficiently mix small amounts of liquids (e.g. droplets). Transport and mixing of the SDSMs is accomplished by means of Laplace pressure-driven flow and several mixing approaches, such as splitting-recombining, stretching-folding, and transversal vortices. Mixing is initiated when separate liquid micro-volumes are transported along respective, juxtaposed wettable tracks. As the liquid micro-volumes coalesce, subsequent mixing occurs during transport of the combined volume over a third separate wettable track that also features a non-wettable “island.” The two-dimensional island disrupts the flow of liquids, in a similar manner a three-dimensional obstacle would, thus generating subsequent mixing. Several SDSMs, each having different island geometries, were investigated, giving rise to a greater understanding of efficient mixing on surfaces. The study offers a design basis for developing a low-cost surface microfluidic mixing device on various substrates.

5:42PM E13.00005 A New Contact Line Structure for Surfactant-Driven Super-spreading Phenomenon, HSIENT-HUNG WEI, National Cheng Kung University — We propose a new contact line structure capable of explaining the curious linear spreading law observed in surfactant-driven superspreading. We show that a tiny surfactant leak from the air-liquid interface to the substrate suffices to promote the motion of the contact line. This leak leads to a microscopic surfactant-depletion zone on the interface in the vicinity of the contact line. Together with pressure buildup by the Marangoni shearing, a distinctive capillary nose is then developed over the zone to drive the contact line in a surfactant-free manner at a constant wetting speed, which explains the linear superspreading law. Our study not only captures many features seen in previous experiments and simulations, but also provides renewed insights into the superspreading phenomenon.

5:55PM E13.00006 When Marangoni meets Savart: The distant interaction of a drop with a liquid sheet, BAPTISTE NEEL, EMMANUEL VILLERMAUX, Aix-Marseille University — The interaction of a radially expanding water sheet (Savart) with an ethanol droplet evaporating at a short distance from it is investigated. The millimetric pendant droplet is positioned a few millimeters above a horizontal sheet, whose thickness is typically a few tens of microns. Although the droplet and the sheet are not in contact, the sheet radius is abruptly reduced downstream of the droplet. We infer that the introduction of a few molecules of ethanol vapor emanating from the drop into the water sheet decreases its thickness, via a localized surface tension deficit. The corresponding Marangoni stresses induce a flow which progressively digs the sheet, hardening its rupture. A quantitative mechanism is proposed to represent all these observations, whose relevance to the puzzling problem of thin films (in the micron range) stability is underlined.

Sunday, November 22, 2015 4:50PM - 6:08PM – Session E14 General Fluid Dynamics: Drag Reduction

5:40PM E14.00001 Investigations of Air Perfusion through Porous Media and Super-Hydrophobic Surface Active Gas Replenishment1, MARC PERLIN, JAMES W. GOSE, KEVIN GOLOVIN, STEVEN L. CECCIO, ANISH TUTEJA, Univ of Michigan — Ann Arbor — Super-hydrophobic (SH) materials have been used successfully to generate reduced skin-friction in laminar flows. Success in the laminar regime has led researchers to try SH materials in turbulent flows. More often than not, this has been unsuccessful at providing meaningful skin-friction drag reduction, and has even generated increased drag. This failure is frequently attributed to the wetting of an SH surface or equivalently the transition from the Cassie-Baxter to the Wenzel state. The result is fluid flow over an essentially roughened surface. In this investigation the researchers aim to perfuse small amounts of gas through porous media, including sintered and foam metals, to attain skin-friction drag reduction in a fully-developed turbulent channel flow. As air is perfused through porous media, the solid - liquid interaction at the interface transitions to a solid - liquid - gas interaction. This can result in an interface that functions similarly to SH materials. Controlled air perfusion that provides the necessary replenishment of lost gas at the interface might prevent wetting, and thus eliminate or reduce the effect of roughness on the flow. This latter possibility is investigated by perfusing small amounts of gas through porous media with and without SH coatings. To quantify the effectiveness of this method, pressure drop is used to infer friction drag along the surface in a fully-developed turbulent channel flow.

1The authors recognize the support of ONR.

5:03PM E14.00002 A priori models for predicting drag reduction for flow over heterogeneous slip boundaries, MARGARET HECK, DIMITRIOUS PAPAVASSILIOU, University of Oklahoma — Slip at fluid-fluid/fluid-solid interfaces is a subject of interest for many engineering applications, ranging from porous materials to biomedical devices to separation processes. Despite remarkable effort to include the effects of surface topology and various flow and physical properties in models describing fluid slip, the mathematical description of flow over mixed slip boundaries is still under investigation. Using similarity theory, which is based on the generalized homogeneity of physical laws governing most systems and takes advantage of similarity in the spatial distribution of characteristics of motion, the equivalent slip velocity is shown to be a function of the geometry of a microfluidic system. The results are used to predict the slip velocity for flow over surfaces with periodically repeating no-slip/free-shear boundaries in the shape of rectangles for 16%-50% solid fractions. The equivalent slip velocity for flow over rectangular boundaries can then be related to the those for flow over surfaces with square and circular no-slip boundaries using characteristic length ratios. The models developed using this approach can be directly used to estimate the slip velocity for flow over various free-shear/no-slip boundaries for Couette, laminar flow conditions.

5:15PM E14.00003 Sustained Drag Reduction in Turbulent Taylor-Couette Flows Enabled by Low-temperature Leidenfrost Effect, DHANANJAI SARANADHI, DAYONG CHEN, JUSTIN KLEINGARTNER, MIT, SIDDARTH Srinivasan, Harvard, ROBERT COHEN, GARETH MCKINLEY, MIT — A submerged body can be heated past its Leidenfrost temperature to form a thick, continuous film of steam between itself and the water. Here we employ a superhydrophobic surface to drastically reduce the energy input required to create and sustain such a boiling film, and use the resulting slip boundary condition to achieve skin friction drag reduction on the inner rotor of a bespoke Taylor-Couette apparatus. We find that skin friction can be reduced by over 90% relative to an unheated superhydrophobic surface at Re= 19,200, and derive a boundary layer and slip theory to fit the data to a model that calculates a slip length of 3.12 ± 0.4 mm. This indicates that the boiling film has a thickness of 112 μm, which is consistent with literature.
validate this state-space model through comparison with existing analytic solutions for elliptic wings and an unsteady inviscid panel method. Of particular interest in this study is (1) the impact of surface texture shape and gap size on the resulting surface skin friction, (2) the importance of the viscosity ratios of the two fluids and its relationship to drag reduction, and (3) longevity of effectiveness when comparing liquid- to air-influenced surfaces.

This work was supported by the Office of Naval Research under MURI grant numbers: N000141210875, N000141210962, and N000141310458.

5:42PM E14.00005 Drag Coefficient of Thin Flexible Cylinder . CHELAKARA SUBRAMANIAN, HARIKA GURRAM, Florida Institute of Technology — Measurements of drag coefficients of thin flexible cylindrical wires are described for the Reynolds number range between 250 – 1000. Results indicate that the coefficient values are about 20 to 30 percent lower than the reported laminar flow values for rigid cylinders. Possible fluid dynamics mechanism causing the reduction in drag will be discussed.

5:55PM E14.00006 Geometry Mediated Drag Reduction in Taylor-Couette Flows . SHABNAM RAAYAI, GARETH MCKINLEY, Massachusetts Inst of Tech-MIT — Micro-scale ribbed surfaces have been shown to be able to mediate drag reduction on rough surfaces such as skin friction on both natural and surfaces. Previous experiments have shown that ribbed surfaces can reduce skin friction in turbulent flow by up to 4-8% in the presence of zero or mild pressure gradients [1]. Our previous computations have shown a substantial reduction in skin friction using micro-scaled ribs of sinusoidal form in high Reynolds number laminar boundary layer flow. The mechanism of this reduction is purely viscous, through a geometrically-controlled retardation of the flow in the grooves of the surface. The drag reduction achieved depends on the ratio of the amplitude to the wavelength of the surface features and can be presented as a function of the wavelength expressed in dimensionless wall units. Here we extend this work, both experimentally and numerically, to consider the effect of similar ribs on steady viscous flow between concentric cylinders (Taylor-Couette flow). For the experimental work, the inner rotating cylinder (rotor) is machined with stream-wise V-groove structures and experiments are performed with fluids of different viscosity to compare the measured frictional torques to the corresponding values on a smooth flat rotor as a measure of drag reduction. The numerical work is performed using the OpenFOAM® open source software to compare the results and understand the physical mechanisms underlying this drag reduction phenomenon. [1] D. Bechert et al., Experiments in Fluids 28 (2000)

Sunday, November 22, 2015 4:50PM - 6:08PM – Session E15 Aerodynamics: Unsteady Airfoil and Wing

4:50PM E15.00001 Modeling intermittent leading-edge vortex shedding in unsteady airfoil flows with reduced-count discrete vortices1 . ASHOK GOPALARATHNAM, North Carolina State University, KIRAN RAMESH, University of Glasgow, UK, ARUN VISHNU SURESH BABU, North Carolina State University — A discrete-vortex method for unsteady airfoil flows with intermittent leading-edge vortex (LEV) shedding was proposed by Ramesh et al (JFM, 2014). Two novelties were introduced: (i) LEV shedding is initiated using discrete vortices whenever the Leading Edge Suction Parameter (LESP), which is a measure of leading-edge suction, exceeds a critical value, and (ii) the strength of the discrete vortices is determined such that the LESP maintained at the critical value during the shedding process. Although results from this low-order method agree with CFD and experiments, the increasing vortex count with time increases the computational cost. The large number of shed vortices from the TE can be reduced through traditional techniques such as amalgamation and deletion, as they typically convect away from the airfoil and interact only weakly with the airfoil vorticity. The LEV, on the other hand, interacts strongly with the airfoil, and has a large influence on the forces. An approach to reduce the vortex count is desired. Inspired by Wang and Eldredge (TCFD, 2013), we propose a model that has just a single vortex to model an active LEV. The varying strength of this free vortex is determined using our LESP criterion. Results from the method for unsteady airfoil motions are promising.

1This work was supported by AFOSR grant FA 9550-13-1-0179

5:03PM E15.00002 Assessment of Control Volume Estimation of Thrust for a Sinusoidally Pitching Airfoil at Low Reynolds Number1 , PATRICK HAMMER, AHMED NAGUIB, MANOOCHEHR KHESEHANI, Michigan State University — The propal estimation of thrust is very important for understanding the aerodynamics of oscillating airfoils at low chord Reynolds number Re. Although direct force measurement is possible, force values at low Re are often small, and separation of the test-models inertia forces from the data may not be straightforward. A common alternative is a control-volume (CV) approach, where terms in the integral momentum equation are computed from measured wake velocity profiles. Although it is acceptable to use only the mean streamwise-velocity profile in estimating the streamwise force on stationary airfoils, recent work has highlighted the importance of terms relating the velocity fluctuation and pressure distribution in the wake for unsteady airfoils. The goal of the present work is to capitalize on 2D computational data for a harmonically pitching airfoil at Re in the range 2,000-22,000, where all terms in the momentum-integral equation are accessible, to evaluate the importance of the various terms in the equation and assess the accuracy of the assumptions that are typically made in experiments due to the difficulty in measuring certain terms (such as the wake pressure distribution) by comparing the CV results with the actual computed thrust.

1This work was supported by AFOSR grant number FA9550-10-1-0342.
5:29PM E15.00004 The Development of the Vorticity Field Downstream of a NACA0012 Airfoil Undergoing Small Amplitude Sinusoidal Oscillation. COLIN STUTZ, Clark-son University, PATRICK HAMMER, Michigan State University, DOUGLAS BOHL, Clarkson University, MANOOCHEHR KOOCHEFSAHANI, Michigan State University — Symmetric small amplitude oscillation of an airfoil produces a semi-infinite array of alternating sign vortices. Under some conditions single vortices of alternating sign are produced for each cycle, whereas under other conditions multiple vortices of each sign are produced. This work investigates a reduced frequency range, \( k = 3.5 - 5.1 \), for which at low \( k \) (3.5-4.5) two vortices of each sign are produced, at higher \( k \) (5.0) the two vortices pair, and finally at the highest \( k \) (5.1) a single vortex of each sign is produced. The work utilizes highly resolved (spatially and temporally) computations for \( \alpha_{\text{max}} = 2^\circ \) and \( Re = 12000 \). For low \( k \) the second vortex has the higher vorticity of the two and the two vortices spread vertically away from each other. As the reduced frequency increases the magnitude of the peak vorticity equalizes between the two structures and they move towards each other as the downstream distance is increased. At the highest reduced frequencies these two structures merge downstream for form a single structure in the more typical von Karman vortex array. The range of reduced frequencies over which the flow transitions from multi-structure, and spreading in nature, to that of single structures is narrow, \( k = 5.1 \).

5:42PM E15.00005 Evolution and Control of the Leading Edge Vortex on an Unsteady Wing. JAMES AKKALA, JAMES BUCHHOLZ, University of Iowa — The development of the leading-edge vortex is investigated on a periodically plunging plate within a uniform free stream. Vortex circulation is governed primarily by the strength of the leading edge shear layer, which provides the primary source of circulation, and a substantial opposite-sign contribution due to the pressure-gradient-driven diffusive flux of vorticity from the suction surface of the plate. The latter has been shown to produce a substantial reduction in leading-edge vortex strength, and leads to the development of a secondary vortex whose evolution influences the interaction between the leading edge vortex and the surface, and thus alters the surface pressure gradients. Suction is applied in the vicinity of the secondary vortex in an attempt to regulate the aerodynamic loads in the presence of the leading-edge vortex. The effect on vorticity transport, leading-edge vortex dynamics, and the resulting aerodynamic loads is discussed.

5:55PM E15.00006 Streamwise Oscillation of Airfoils into Reverse Flow. KENNETH GRANLUND, North Carolina State University, ANYA JONES, University of Maryland, MICHAEL OL, Air Force Research Laboratory — An airfoil in freestream is oscillated in streamwise direction to cyclically enter reverse flow. Measured lift is compared to analytical blade element theory. Advance ratio, reduced frequency and angle of attack is varied within those typical for helicopters. Experimental results reveal that lift does not become negative in the flow reversal part, contradicting one theory and supported by another. Flow visualization reveal the leading edge vortex advecting against the freestream to a point in front of the leading edge.

Sunday, November 22, 2015 4:50PM - 6:08PM – Session E16 Aerodynamics: Theory and Vehicles

4:50PM E16.00001 Flow Structures within a Helicopter Rotor Hub Wake. BRIAN ELBING, Oklahoma State Univ, DAVID REICH, SVEN SCHMITZ, Pennsylvania State University — A scaled model of a notional helicopter rotor hub was tested in the 48” Garfield Thomas Water Tunnel at the Applied Research Laboratory Penn State. The measurement suite included total hub drag and wake velocity measurements (LDV, PIV, stereo-PIV) at three downstream locations. The main objective was to understand the spatiotemporal evolution of the unsteady wake between the rotor hub and the nominal location of the empennage (tail). Initial analysis of the data revealed prominent two- and four-per-revolution fluid structures linked to geometric hub features persisting into the wake far-field. In addition, a six-per-revolution fluid structure was observed in the far-field, which is unexpected due to the lack of any hub feature with the corresponding symmetry. A nonlinear interaction is occurring within the wake to generate these structures. This presentation will provide an overview of the experimental data and analysis with particular emphasis on these six-per-revolution structures.

5:03PM E16.00002 A Hybrid Vortex Sheet / Point Vortex Model for Unsteady Separated Flows. DARWIN DARAKANANDA, JEFF D. ELDREDGE, University of California, Los Angeles, TIM COLONIUS, California Institute of Technology, DAVID R. WILLIAMS, Illinois Institute of Technology — The control of separated flow over an airfoil is essential for obtaining lift enhancement, drag reduction, and the overall ability to perform high agility maneuvers. In order to develop reliable flight control systems capable of realizing agile maneuvers, we need a low-order aerodynamics model that can accurately predict the force response of an airfoil to arbitrary disturbances and/or actuation. In the present work, we integrate vortex sheets and variable strength point vortices into a method that is able to capture the formation of coherent vortex structures while remaining computationally tractable for control purposes. The role of the vortex sheet is limited to tracking the dynamics of the shear layer immediately behind the airfoil. When parts of the sheet develop into large scale structures, those sections are replaced by variable strength point vortices. We prevent the vortex sheets from growing indefinitely by truncating the tips of the sheets and transferring their circulation into nearby point vortices whenever the length of sheet exceeds a threshold. We demonstrate the model on a variety of canonical problems, including pitch-up and impulse translation of an airfoil at various angles of attack.

5:16PM E16.00003 On Entropy Trail. SAEED FAROKHI, RAY TAGHAVI, SHAWN KESHMIRI, University of Kansas — Stealth technology is developed for military aircraft to minimize their signatures. The primary attention was focused on radar signature, followed by the thermal and noise signatures of the vehicle. For radar evasion, advanced configuration designs, extensive use of carbon composites and radar-absorbing material, are developed. On thermal signature, mainly in the infra-red (IR) bandwidth, the solution was found in blended rectangular nozzles of high aspect ratio that are shielded from ground detectors. For noise, quiet and calm jets are integrated into vehicles with low-turbulence configuration design. However, these technologies are totally incapable of detecting new generation of revolutionary aircraft. These shall use all electric, distributed, propulsion system that are thermally transparent. In addition, composite skin and non-emitting sensors onboard the aircraft will lead to low signature. However, based on the second-law of thermodynamics, there is no air vehicle that can escape from leaving an entropy trail. Entropy is thus the only inevitable signature of any system, that once measured, can detect the source. By characterizing the entropy field based on its statistical properties, the source may be recognized, akin to face recognition technology. Direct measurement of entropy is cumbersome, however as a derived property, it can be easily measured. The measurement accuracy depends on the probe design and the sensors onboard. One novel air data sensor suite is introduced with promising potential to capture the entropy trail.
5:29PM E16.00004 Performance Evaluation of Leading Edge Slats on Rigid Wing Sail Catamarans
CHELSEA JOHNSON, Union University, CHARLES O’NEILL, University of Alabama — Rigid wing sails have created the fastest catamarans in history; however, with the addition of a leading edge slat higher lift and faster speeds may be achieved. Slats are currently used on airplane wings to increase lift, but have not been implemented on a rigid wing sail catamaran. Using 3D modeling and computational fluid dynamics software, this research investigates the effect that slats have on the performance of rigid wing sail catamarans. Aerodynamics and hydrodynamics form the basis of the research. The preliminary results show an increase in the coefficient of lift for sail models with slats over sail models without slats, allowing the catamaran to perform at higher speeds. The ability of the slat to rotate has also been identified as a key factor in increasing the benefit of the slat.

1This work was supported by NSF site award 1358991.

5:42PM E16.00005 Steady and Unsteady Aerodynamics of Thin Airfoils with Porosity Gradients
ROZHN HAJJIAN, JUSTIN W. JAWORSKI, Lehigh University — Porous treatments have been shown in previous studies to reduce turbulence noise generation from the edges of wings and blades. However, this acoustical benefit can come at the cost of aerodynamic performance that is degraded by seepage flow through the wing. To better understand the trade-off between acoustic stealth and the desired airflow performance, the aerodynamic loads of a thin airfoil in uniform flow with a prescribed porosity distribution are determined analytically in closed form, provided that the distribution is Hölder-continuous. The theoretical model is extended to include unsteady heaving and pitching motions of the airfoil section, which has applications to the performance estimation of biologically-inspired swimmers and fliers and to the future assessment of vortex noise production from porous airfoils.

5:55PM E16.00006 The leading-edge stall of airfoils with various nose shapes
MATTHEW KRALJIC, ZVI RUSAK, Rensselaer Polytechnic Institute, SHIXIAO WANG, U Auckland, NZ — We study the inception of leading-edge stall on stationary, smooth thin airfoils with various nose shapes of the form $x^n$ (where $0 < a < 1/2$) at low to moderately high chord Reynolds number flows. A reduced-order, multi-scale model problem is developed and solved using numerical simulations. The asymptotic theory demonstrates that a subsonic flow about a thin airfoil can be described in terms of a uniform stagnation region, an outer region, and an inner region, around the nose, that asymptotically match each other. The flow in the outer region is dominated by the classical thin airfoil theory. Scaled (magnified) coordinates and a modified (smaller) Reynolds number $Re_M$ are used to correctly account for the nonlinear behavior and extreme velocity changes in the inner region, where both the near-stagnation and high suction areas occur. The inner region problem is solved numerically to determine the inception of leading-edge stall on the nose. It is found that stall is delayed to higher angles of attack with the decrease of nose parameter $a$. Specifically, new airfoil shapes are proposed with increased stall angle at subsonic speeds and higher critical Mach numbers at transonic speeds.

Sunday, November 22, 2015 4:50PM - 6:08PM — Session E17 Flow Control: Instabilities 205 - Matthew Juniper, University of Cambridge

4:50PM E17.00001 Experimental sensitivity analysis and control of thermoacoustic systems
MATTHEW JUNIPER, NICHOLAS JAMIESON, University of Cambridge, LARRY LI, Hong Kong University of Science and Technology, GEORGIOS RIGAS, University of Cambridge — We report the results of an experimental sensitivity analysis on a thermoacoustic system: an electrically heated Rijke tube. We measure accurately the change of the linear stability characteristics of the system, quantified as shifts in the growth rate and oscillation frequency, that is caused by the introduction of a passive control device. In the case presented here, the control device is a mesh, which causes drag in the system. The rate of growth is slow, so the growth rate and frequency can be measured over many hundred cycles in the linear regime with and without control. This means that the shift in growth rate and frequency can be calculated very accurately. These measurements agree well with the theoretical predictions from adjoint-based methods of Magri & Juniper (JFM 2013, 719, 183-202). The results suggest that adjoint-based methods can accurately predict the effect of different passive control devices on the stability of a thermoacoustic system, opening new avenues for the development, implementation and validation of control strategies for more complex thermoacoustic systems.

5:03PM E17.00002 Tollmien-Schlichting wave cancellation by feedback control
HARI VEMURI, JONATHAN MORRISON, ERIC KERRIGAN, Imperial College London — Tollmien-Schlichting (TS) waves are primary instabilities in the boundary layer and, by actively intervening with their growth, the transition process can be delayed. In this study the experimental results of both open-loop and real-time feedback control will be shown for 3D TS waves excited within a flat-plate boundary layer. They are excited at a 0.75mm pin-hole source driven by a speaker. A 0.75 mm thin, dual slot geometry is used for actuation by another speaker and a wall hot-wire sensor manufactured in-house is used as the sensor for feedback control. The spatial transfer function models between the source and sensor ($G_s$) and the actuator and sensor ($G_a$) obtained by classical frequency sweep techniques are used to synthesize various types of robust, stabilizing controllers ($K$). The transfer function $G_s$ determines the unstable range of frequencies whereas $G_a$ together with $K$ determines the stability of the closed-loop. A second traversing hot-wire is used to record the performance of the controller downstream. It is shown that the experimental transfer functions agree remarkably well with numerical calculations as do the predicted results from feedback control. Preliminary experimental feedback control results for various other actuator configurations will also be presented.

1This work is supported by EADS (support agreement number IW203591) and LFC-UK.

5:16PM E17.00003 Controlling Spatiotemporal Chaos in Active Dissipative-Dispersive Nonlinear Systems
SUSANA GOMES, Department of Mathematics, Imperial College London, MARC PRADAS, Department of Mathematics and Statistics, The Open University, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, DEMETRIOS PAPAGEORGIOU, GRIGORIOS PAVLIOTIS, Department of Mathematics, Imperial College London — We present a reduced-order, multi-scale model problem is developed and solved using numerical simulations. The asymptotic theory demonstrates that a subsonic flow about a thin airfoil can be described in terms of a uniform stagnation region, an outer region, and an inner region, around the nose, that asymptotically match each other. The flow in the outer region is dominated by the classical thin airfoil theory. Scaled (magnified) coordinates and a modified (smaller) Reynolds number $Re_M$ are used to correctly account for the nonlinear behavior and extreme velocity changes in the inner region, where both the near-stagnation and high suction areas occur. The inner region problem is solved numerically to determine the inception of leading-edge stall on the nose. It is found that stall is delayed to higher angles of attack with the decrease of nose parameter $a$. Specifically, new airfoil shapes are proposed with increased stall angle at subsonic speeds and higher critical Mach numbers at transonic speeds.

5:29PM E17.00004 ABSTRACT WITHDRAWN —
subsequent vortex rings. The vectoring behaviour is shown to be primarily influenced by pinch-off time of vortex rings generated by the synthetic jets. Beyond a certain experimentally. Phase-locked vorticity fields, measured using Particle Image Velocimetry (PIV), are used to track vortex pairs. The physical actuator lagging in phase. In the present study, the influence of phase difference and Strouhal number on the vectoring behaviour is examined between the two driving signals. The resulting jet can be merged or bifurcated and either vectored towards the actuator leading in phase or the RAMANI, GUILLAUME GOMIT, University of Southampton — A pair of parallel synthetic jets can be vectored by applying a phase difference of 5°. This work seeks to examine control authority of the LAFPAs in the supersonic regime. Experiments conducted with a supersonic cavity demonstrated the LAFPAs retain the ability to suppress or excite resonance. The ability to either excite or suppress resonance, as needed, is required in some applications.

Sponsored by AFRL

5:03PM E18.00002 LES-based characterization of a suction and oscillatory blowing fluidic actuator. JOELONLAE KIM, PARVIZ MOIN, Center for Turbulence Research, Stanford University — Recently, a novel fluidic actuator using steady suction and oscillatory blowing was developed for control of turbulent flows. The suction and oscillatory blowing (SaOB) actuator combines steady suction and pulsed oscillatory blowing into a single device. The actuation is based upon a self-sustained mechanism of confined jets and does not require any moving parts. The control output is determined by a pressure source and the geometric details, and no additional input is needed. While its basic mechanisms have been investigated to some extent, detailed characteristics of internal turbulent flows are not well understood. In this study, internal flows of the SaOB actuator are simulated using large-eddy simulation (LES). Flow characteristics within the actuator are described in detail for a better understanding of the physical mechanisms and improving the actuator design. LES predicts the self-sustained oscillations of the turbulent jet. Switching frequency, maximum velocity at the actuator outlets, and wall pressure distribution are in good agreement with the experimental measurements. The computational results are used to develop simplified boundary conditions for numerical experiments of active flow control.


1Supported by the Boeing company

5:16PM E18.00003 Modeling of a zero-net mass flux actuator for aqueous media. BRADLEY AYERS, California State University, Northridge, CHARLES HENOCH, Naval Undersea Warfare Center, HAMID JOHARI, California State University, Northridge — A zero-net mass flux actuator was designed to maximize the jet thrust with a 3-in size constraint. The actuator was driven by a solenoid moving a piston in a cavity and when the solenoid circuit was opened, a return spring pulled the piston until the de-energized position was reached. Using the solenoid characteristics, a model was developed to assist in determining the optimal design parameters such as the piston diameter and stroke length, orifice diameter, and the spring constant. The model consisted of three separate elements: the solenoid and return spring forces; the fluid inertia within the cavity as well as the mass of moving parts; and the pressure loss associated with the orifice. The actuator model was used to determine the piston motion through one cycle. A piston stroke length of 4 mm and a cylinder bore of 45 mm was chosen resulting in a slug stroke ratio of 3.9. For the shortest possible waveform and the chosen actuator parameters, the model predicted the piston reaching its maximum stroke of 4 mm in 10 ms, and then returning to its resting position in 31 ms. Thus, a maximum frequency of 32 Hz was found for the shortest waveform in an ideal setup. The model predictions were compared with PIV measurements.

1Supported by ONR-NURP

5:29PM E18.00004 Vectoring of parallel synthetic jets. TIM BERK, BHARATHRAM GANAPATHISUBRAMANI, GUILLAUME GOMIT, University of Southampton — A pair of parallel synthetic jets can be vectored by applying a phase difference between the two driving signals. The resulting jet can be merged or bifurcated and either vectored towards the actuator leading in phase or the actuator lagging in phase. In the present study, the influence of phase difference and Strouhal number on the vectoring behaviour is examined experimentally. Phase-locked vorticity fields, measured using Particle Image Velocimetry (PIV), are used to track vortex pairs. The physical mechanisms that explain the diversity in vectoring behaviour are observed based on the vortex trajectories. For a fixed phase difference, the vectoring behaviour is shown to be primarily influenced by pinch-off time of vortex rings generated by the synthetic jets. Beyond a certain formation number, the pinch-off timescale becomes invariant. In this region, the vectoring behaviour is determined by the distance between subsequent vortex rings.

We acknowledge the financial support from the European Research Council (ERC grant agreement no. 277472).
stereo PIV measurements were performed at the University of Toronto Institute for Aerospace Studies (UTIAS), in the spanwise-wall-normal plane. Actuation focuses on the application of active flow control, in the form of synthetic jet actuators, of turbulent boundary layers. An array of 2 synthetic jets are oriented in the spanwise direction and located approximately 2.7 meters downstream from the leading edge of a flat plate. Actuation is applied perpendicular to the surface of the flat plate with varying blowing ratios and reduced frequencies (open-loop). Two-component large jet measurements were performed at the University of Toronto Institute for Aerospace Studies (UTIAS), in the spanwise-wall-normal plane. The freestream Reynolds number is 3x10^6, based on the boundary layer thickness. The skin friction Reynolds number is 1.200 based on the skin friction velocity. The experiments at UTIAS allow for the observation of the control effects as the flow propagates downstream. The experiments at UTIAS allow for the observation of the streamwise vorticity induced from the actuation. Overall the two experiments provide a 3D representation of the flow field with respect to actuation effects. The current work focuses on the comparison of the two experiments, as well as the effects of varying blowing ratios and reduced frequencies on the turbulent boundary layer.

The support of AFOSR under grant number FA 9550-12-1-0469 is gratefully acknowledged.

Funded Supported by Airbus
1 The effects of high and low shear stress. Additionally, conditionally averaged mean velocity profiles will be presented to assess the impact on wall shear stress. Thus, estimation of the wall shear stress from the low-frequency velocity signal will be performed, and is expected to be statistically significant in the outer boundary layer. Experimentally, the large-scales also appear to amplitude and frequency modulate the eddies of large-scales are the dominating eddies having significantly higher energy, than the small-scales across almost the entire boundary layer even at the low to moderate Reynolds numbers under consideration. The small-scale energy has a peak in the near-wall region associated with the near-wall turbulent cycle as in canonical boundary layers. However, eddies of large-scales are the dominating eddies having significantly higher energy, than the small-scales across almost the entire boundary layer even at the low to moderate Reynolds numbers under consideration. The large-scales also appear to amplitude and frequency modulate the smaller scales across the entire boundary layer.

4:50PM E20.00001 Computations of turbulent channel flow using the nested-LES approach

5:03PM E20.00002 A metric for assessing the dynamic content of large-eddy simulations

5:16PM E20.00003 Discrete dynamical system approximation to the Boltzmann equation for eddy-viscosity-free LES modeling of transitional flow

5:29PM E20.00004 Autonomic Closure for Large Eddy Simulation

This material is based upon work supported by the National Science Foundation Graduate Research Fellowship under Grant No. DGE-1315138.
5:42PM E20.00005 Temporal characterization of turbulence and coherent structures in a recirculating flow. JESUS RAMIREZ-PASTRAN, CARLOS DUQUE-DAZA, Department of Mechanical & Mechatronics Engineering, Universidad Nacional de Colombia, Bogota, Colombia, DUNCAN A. LOCKERBY, School of Engineering, University of Warwick, Coventry CV4 7AL, UK — Analysis of the temporal behavior of a recirculating flow is performed by numerical experiments on a lid-driven cavity setup. Simulations at two Reynolds numbers, based on the cavity depth (Re=3200 and Re=12000), were computed for an incompressible turbulent flow using LES. Good agreement was observed with results reported for velocity profiles along vertical and horizontal planes, but discrepancies against experimental data were found for the fluctuating velocity profiles. By using long-term simulations a complex quasi-periodic behaviour is observed and a set of dominant frequencies identified. Such behaviour is also identified by examining the temporal evolution of the TKE production and viscous dissipation terms. Coherent structures based on the Q-criterion are calculated and used to characterize the quasi-periodic behavior. Simple correlations are established between the structures and the TKE terms. Results seem to indicate that the apparently unstable behaviour is promoted by the existence of two internal flow streams, located at the side walls, and which collide at a region around the centre of the cavity. It is concluded that this interaction is responsible for the appearance of the rich set of frequencies observed at the different locations within the recirculating flow.

5:55PM E20.00006 Study of dealiasing schemes in pseudo-spectral methods for Large-Eddy Simulations of incompressible flows. FABIEN MARGAIRAZ, University of Utah, MARCO GIOMETTO, Ecole Polytechnique Federale de Lausanne, MARC PARLANGE, University of British Columbia, MARC CALAF, University of Utah — The performance of dealiasing schemes and their computational cost on a pseudo-spectral code are analyzed. Dealiasing is required to limit the error that occurs when two discretized variables are multiplied, polluting the accuracy of the result. In this work three different dealiasing methods are explored: the 2/3 rule, the 3/2 rule, and a high order Fourier smoothing based method. We compare the cost of the traditionally accepted 3/2 rule (Canuto et al. 1988), where an expansion of the computational domain to a larger grid is required, to the cost of the other two techniques that do not require this expansion. This analysis is performed in the framework of Large-Eddy Simulations (LES) of incompressible flows using the constant Smagorinsky sub-grid model with a wall damping function and a wall model based on the log-law. A highly efficient LES code parallelized using a 2D pencil decomposition has been developed. The code employs the traditional pseudo-spectral approach to integrate the incompressible Navier-Stokes equations. Several simulations of a neutral atmospheric boundary layer using different degrees of numerical resolution are considered. Results show a net difference in computational cost between the different techniques without relevant changes in statistics.

Sunday, November 22, 2015 4:50PM - 6:08PM — Session E21 Turbulence: Planetary Boundary Layers

4:50PM E21.00001 Nonequilibrium Response of the Daytime Atmospheric Boundary Layer to Mesoscale Forcing. JAMES BRASSEUR, BAJALI JAYARAMAN, Penn State, SUE HAUPT, JARED LEE, NCAR — The essential turbulence structure of the daytime atmospheric boundary layer (ABL) is driven by interactions between shear and buoyancy. A relatively strong inversion layer “lid” typically confines the ABL turbulence, whose height grows during the day with increasing surface heat flux ($Q_s$) to ~ 1-2 km before collapsing with $Q_s$ towards the day’s end. The 3D “microscale” ABL turbulence is forced largely in the horizontal by winds above the capping inversion at the “mesoscale” of the O(100) km scale. Whereas the “canonical” ABL is in equilibrium and quasi-stationary, quasi-2D weather dynamics at the mesoscale is typically nonstationary at sub-diurnal time scales. We study the consequences of nonstationarity in the quasi-2D mesoscale forcing in horizontal winds and solar heating on the dynamics of ABL turbulence and especially on the potential for significant deviations from the canonical equilibrium state. We apply high-fidelity LES of the dry cloudless ABL over Kansas in July forced at the mesoscale (WRF) with statistical homogeneity in the horizontal. We find significant deviations from equilibrium that appear in a variety of interesting ways. One of the more interesting results is that the changes in mesoscale wind direction at the diurnal time scale can destabilize the ABL and sometimes cause a transition in ABL eddy structure that are normally associated with increased surface heating. Supported by DOE. Computer resources by the Penn State ICS.

5:03PM E21.00002 Towards a Subgrid Model of Planetary Boundary Layers Based on Direct Statistical Simulation1. JOSEPH SKITKA, BAYLOR FOX-KEMPER, BRAD MARSTON, Brown University — Reliable weather and climate modeling requires the accurate simulation of Earth’s oceanic and atmospheric boundary layers. However, long-time integration of the Navier-Stokes equations for centuries beyond the reach of present day computers, hence subgrid modeling is required. Direct statistical simulation (DSS) that is based upon expansion in equal-time cumulants offers the prospect of building improved subgrid schemes. In contrast to other statistical approaches, DSS makes no unphysical assumptions about the homogeneity, isotropy, or locality of correlations. We investigate the feasibility of a second-order closure (CE2) by performing simulations of the ocean boundary layer in a quasi-linear approximation for which CE2 is exact. Wind-driven Langmuir turbulence and thermal convection are studied by comparison of the quasi-linear and fully nonlinear statistics. We also investigate whether or not basis reduction can be achieved by proper orthogonal decomposition (POD) of the second cumulant.

1Supported in part by NSF DMR-1306806 and NSF CCF-1048701.

5:16PM E21.00003 Investigation of the pressure-strain-rate correlation in the convective atmospheric surface layer. KHUONG NGUYEN, SHUAISHUAI LIU, Clemson University, MARTIN OTTE, US EPA, CHENNING TONG, Clemson University — Recent studies have identified the pressure-strain-rate correlation as the main cause of surface layer anisotropy in the convective atmospheric boundary layer. We decompose the pressure field into the rapid, slow, buoyancy, Coriolis, and harmonic parts using large-eddy simulation to investigate their contributions to the pressure-strain-rate correlation. In a strongly convective surface layer, the buoyancy contribution resulting from large-scale temperature fluctuations dominates. Contributions obtained by solving the free-space Poisson equation show the same trends. The buoyancy contribution is much larger than the harmonic part, indicating that the sources terms, rather than the boundary conditions for the pressure Poisson equation are the main cause of the observed behaviors of the pressure-strain-rate. The results have implications for modeling the pressure-strain-rate correlation.
5:29PM E21.00004 Adjustment of mean velocity and turbulence due to a finite-size wind farm in a neutral ABL - A LES study. VARUN SHARMA, cole Polytechnique Fédrale de Lausanne, MARC B. PARLANGE, University of British Columbia, Vancouver, MARC CALAF, University of Utah, Salt Lake City — Large-eddy simulation (LES) has become a well-established tool to simulate and understand the interaction between wind farms and the atmospheric boundary layer (ABL). A popular simulation technique considers wind turbines as actuator disks and simulates “infinite” wind farms due to periodic boundary conditions in the horizontal directions. These simulations have indicated the presence of a fully developed internal boundary layer (IBL) due to ‘wind farm roughness’, which has been shown to have important implications, especially in stratified flow conditions. However, the relationship between the length of the wind farm and the resulting IBL vis-à-vis the asymptotic IBL and its relevance in real-world wind farms is not well understood at present. To address this issue, simulations of wind farms with different horizontal extents are performed in a neutral ABL using an extremely elongated computational domain. Results focus on identifying length scales defining the adjustment of the ABL to a new equilibrium within the wind farm and comparing it to the infinite wind farm case. Furthermore, analyses shall be extended upstream as well as downstream of the wind farm to determine the ‘impact’ region and the ‘exit’ region of the wind farm.

5:42PM E21.00005 Aerodynamic surface stress intermittency and conditionally averaged turbulence statistics1, WILLIAM ANDERSON, DAVID LANIGAN, UT Dallas — Aeolian erosion is induced by aerodynamic stress imposed by atmospheric winds. Erosion models prescribe that sediment flux, Q, scales with aerodynamic stress raised to an exponent, n, where n > 1. Since stress (in fully rough, inertia-dominated flows) scales with incoming velocity squared, \( u^2 \), it follows that \( q \sim u^n \) (where \( u \) is some characteristic the velocity). Thus, even small (turbulent) deviations of \( u \) from its time-mean may be important for aeolian activity. This rationale is augmented given that surface layer turbulence exhibits maximum Reynolds stresses in the fluid immediately above the landscape. To illustrate the importance of stress intermittency, we have used conditional averaging predicated on stress during large-eddy simulation of atmospheric boundary layer flow over an arid, bare landscape. Conditional averaging provides an ensemble-mean visualization of flow structures responsible for erosion events. Preliminary evidence indicates that surface stress peaks are associated with the passage of inclined, high-momentum regions flanked by adjacent low-momentum regions. We characterize geometric attributes of such structures and explore streamwise and vertical vorticity distribution within the conditionally averaged flow field.

5:55PM E21.00006 Large-eddy simulations of mean and turbulence dynamics in unsteady Ekman boundary layers1. MOSTAFA MOMEN, ELIE BOU-ZEID, Princeton University — Unsteady geostrophic forcing in the atmosphere or ocean not only influences the mean wind, but also affects the turbulent statistics. In order to see when turbulence is in quasi-equilibrium with the mean, one needs to understand how the turbulence decays or develops, and how do the turbulent production, transport and dissipation respond to changes in the imposed forcing. This helps us understand the underlying dynamics of the unsteady boundary layers and develop better turbulence closures for weather/climate models and engineering applications. The present study focuses on the unsteady Ekman boundary layer where pressure gradient, Coriolis, and friction forces interact but are not necessarily in equilibrium. Several cases are simulated using LES to examine how the turbulence and resolved TKE budget terms are modulated by the variability of the mean pressure gradient. We also examine the influence of the forcing variability time-scale on the turbulence equilibrium and TKE budget. It is shown that when the forcing time-scale is in the order of the turbulence characteristic time-scale, the turbulence is no longer in quasi-equilibrium due to highly nonlinear mean-turbulence interactions and hence the conventional log-law and turbulence closures are no longer valid.

Sunday, November 22, 2015 4:50PM - 6:08PM Session E22 Turbulent Boundary Layers: Wall Modeling 210 · Kurt Aikens, Houghton College

4:50PM E22.00001 Inviscid Wall-Modeled Large Eddy Simulations for Improved Efficiency1, KURT AIKENS, KYLIE CRAFT, ANDREW REDMAN, Houghton College — The accuracy of an inviscid flow assumption for wall-modeled large eddy simulations (LES) is examined because of its ability to reduce simulation costs. This assumption is not generally applicable for wall-bounded flows due to the high velocity gradients found near walls. In wall-modeled LES, however, neither the viscous near-wall region or the viscous length scales in the outer flow are resolved. Therefore, the viscous terms in the Navier-Stokes equations have little impact on the resolved flowfield. Zero pressure gradient flat plate boundary layer results are presented for both viscous and inviscid simulations using a wall model developed previously. The results are very similar and compare favorably to those from another wall model methodology and experimental data. Furthermore, the inviscid model reduces simulation costs by about 25% and 39% for supersonic and subsonic flows, respectively. Future research directions are discussed as are preliminary efforts to extend the wall model to include the effects of unresolved wall roughness.

1This work used the Extreme Science and Engineering Discovery Environment (XSEDE), which is supported by National Science Foundation grant number ACI-1053575. Computational resources were provided by the Texas Advanced Computing Center at the University of Texas.

5:03PM E22.00002 Application of the Integral Length-Scale Approximation to Wall Modelled LES, AMIRREZA ROUHI, UGO PIOMELLI, Queen’s University, Canada, ALEXANDRE SILVA-LOPES, University of Porto, Portugal — A new length-scale for modelling the unresolved stresses in LES was proposed [Piomelli et al., J. Fluid Mech., 766, 2015] in which the filter width is related to the turbulence statistics instead of the grid. The model constant is assigned by requiring that the unresolved subfilter scales (SFS) support some percentage of the total stress. This model gave very good results in wall-resolved LES channel flow. When the same model is applied to wall-modelled simulations, however, significant errors result due to the requirement that the contribution of SFS to the transport be constant through the channel. Near the wall, the grid becomes larger than the mixing length of the flow, and the resolved eddies are not able to support the desired contribution to the transport. Better agreement is obtained by requiring that, as the wall is approached, the SFS contribute an increasing percentage of the momentum transport, reaching 100% at the wall. The model was tested on channel flows at \( Re \sim 5 \times 10^4 \) and \( 5 \times 10^5 \), 256 \( \times \) 80 \( \times \) 128 and 256 \( \times \) 160 \( \times \) 128 grid points. The simulations predict the universal log-law very well. The model has the same cost as the Smagorinsky model and the robustness of the dynamic eddy-viscosity model.
5:16PM E22.00003 Multiple-relaxation-time lattice Boltzmann simulations of turbulent channel and pipe flows. , HARISHANKAR MANIKANTAN, CHENG PENG, LIAN-PING WANG, University of Delaware — The mesoscopic Lattice Boltzmann method (LBM) has become a reliable alternative for solving incompressible turbulent flows. However, the statistics of a simulated turbulent flow near a curved boundary may deviate from the physical rotational invariance (RI) of lattice coordinates. The main objective of this study is to compare the effects of different lattice models on the simulation results of turbulent flows, and explore ways to restore RI near a curved boundary. We will apply D3Q19 and D3Q27 multiple-relaxation-time LBM models to simulate turbulent pipe and channel flows. The statistics of the simulated flows are examined to quantify the nature of departures from RI. To help understand whether the departure is originated from the bounce-back scheme at the solid wall, we will perform simulations of a turbulent channel flow with walls oriented at an angle from the lattice grid, and test the use of an overset lattice grid near a pipe wall. The Chapman-Enskog analysis of these models will be performed to probe RI errors near a boundary. Our goal is to eventually perform an accurate direct numerical simulation of a turbulent pipe flow, and compare the results to previous simulations based on the Navier-Stokes equations.

5:29PM E22.00004 Assessment of turbulence models for boundary layers with pressure gradient and roughness , RABIJIT DUTTA, UGO PIOMELLI, Queens University — The performance of sand-grain-based roughness corrections for the SA, SST $k - \omega$ and $k - c_\epsilon$ models has been evaluated by comparing the model results with large eddy simulation (LES) data. Computations are performed for a turbulent boundary layer with both smooth and rough walls subjected to two different pressure-gradient conditions, namely, an adverse pressure gradient (APG) with separation and a realistic pressure-gradient situation encountered in a hydraulic turbine blade. A new roughness correction was developed for the SST $k - \omega$ model that gave improved results near separation. For the cases with smooth wall, RANS models give reasonable agreement in predicting skin friction coefficient $(c_f)$ at the wall. RANS models predict too high Reynolds stresses in the separated region, which lead to earlier reattachment. For the rough wall computations, the RANS models predict that $c_f$ changes sign much later than the LES data. In the LES, however, the wall stress becomes negative inside the roughness sublayer, and the flow reversal does not correspond to the separation, which occurs much later, where the separation leaves the body, and the total stress above the roughness crest changes sign. The RANS models predict the position of this point more accurately.

5:42PM E22.00005 Investigation of pressure gradient aware wall modeling in LES , OLIVIER THIRY, GREGOIRE WINKELMANS, MATHIEU DUPONCHEEL, Université catholique de Louvain (UCL) - Institute of Mechanics, Materials and Civil Engineering (IMMC) — This work focuses on the investigation of various wall modeling strategies for the simulation of high Reynolds number wall-bounded turbulent flows with acceleration and/or deceleration. Our code is based on fourth order finite differences, is momentum conserving, and is energy conserving up to fourth order. We here use a “channel flow” set-up, with no slip and wall modeling at the bottom, with slip at the top, and with blowing and/or suction at the top in order to generate the desired acceleration-deceleration profile. Two strategies are investigated and compared. Pressure gradient corrected algebraic models are first considered, and we investigate various local roughness corrections so as to avoid imposing mean profile laws pointwise. RANS sub-layer models are then also considered, where the turbulent viscosity is corrected to account for pressure gradient effects and for resolved LES fluctuations effects. A well-resolved LES was also performed to provide a reference solution.

5:55PM E22.00006 Improved engineering models for turbulent wall flows , ZHEN-SU SHE, Peking University, XI CHEN, Texas Tech University, HONG-YUE ZOU, Peking University, FAZLE HUSSAIN, Texas Tech University — We propose a new approach, called structural ensemble dynamics (SED), involving new concepts to describe the mean quantities in wall-bounded flows, and its application to improving the existing engineering turbulence models, as well as its physical interpretation. First, a revised $k - \omega$ model for pipe flows is obtained, which accurately predicts, for the first time, both mean velocity and (streamwise) kinetic energy $\langle u' u' \rangle$ for a wide range of the Reynolds number $(Re)$, validated by Princeton experimental data. In particular, a multiplicative factor is introduced in the dissipation term to model an anomaly in the energy cascade in a meso-layer, predicting the outer peak of $\langle \omega \rangle$ at the wall. RANS models agree with data. Secondly, a new one-equation model is obtained for compressible turbulent boundary layers (CTBL), building on a multi-layer formula of the stress length function and a generalized temperature-velocity relation. The former refines the multi-layer description - viscous sublayer, buffer layer, logarithmic layer and a newly defined bulk zone - while the latter characterizes a parabolic relation between the mean velocity and temperature. DNS data show our predictions to have a 99% accuracy for several Mach numbers $Ma=2.25, 4.5$, improving, up to 10%, a previous similar one-equation model (Baldwin & Lomax, 1978). Our results promise notable improvements in engineering models.


5:03PM E23.00001 Stretch-coil transition of a semiflexible filament in extensional flow , HARISHANKAR MANIKANTAN, DAVID SAINTILLAN, University of California San Diego — We present a theoretical model for the fluctuation-rounded buckling transition of a semiflexible polymer placed in extensional flow. The competition between elastic rigidity and line tension developed in the polymer backbone can trigger a buckling instability, and the effect of thermal fluctuations on this bifurcation has recently gathered significant attention. While this problem has been studied experimentally and computationally before, the exact nature of the stochastic transition is yet to receive a full quantitative treatment. Motivated by the findings of recent experiments and our own numerical simulations, we approach this analytically by expanding a slender-body equation for the polymer around the first deterministic buckled mode at the onset of the instability. This leads us to a Ginzburg-Landau model for the amplitude of the buckled shape, solving which reveals an expression for a stochastic supercritical bifurcation. This solution captures the smooth transition from a stretched state to a buckled state as the extensional flow strength is increased. It matches excellently with full numerical simulations, and corroborates the conclusions drawn from recent microfluidic experiments.

5:03PM E23.00002 Elastohydrodynamics of contact in adherent sheets , ANDREAS CARLSON, Harvard University, SHREYAS MANDRE, Brown University, L. MAHADEVAN, Harvard University — The dynamics of contact between a thin elastic film and a solid arises in many scientific and engineering applications, from the simple saran-wrap to cellular adhesion to grounding lines in ice sheets. Here, we use a mathematical description of the multi-scale processes associated with microscopic adhesion, fluid flow and elastic thin film deformation to deduce the dynamics of the onset of adhesion, as well as the speed and the shape of the adhesion zone. Our analysis is consistent with prior experimental observations, provides new testable predictions for the shape, size and dynamics of adherent contact in thin sheets and in addition provides a broadly applicable prescription for the boundary conditions at elastic contact lines.
5:16PM E23.00003 Dynamics and topology of a flexible chain: knots in steady shear flow , AGNIESZKA SŁOWICKA, Institute of Fundamental Technological Research, Polish Academy of Sciences, Pawinskiego 5b, 02-106 Warsaw, Poland, STEVE KUEI, Rice University, Department of Chemical and Biomolecular Engineering, Houston, TX 77005, USA, MARIA EKIEL-JEZEWSKA, ELGIUSZ WAJNRYB, Institute of Fundamental Technological Research, Polish Academy of Sciences, Pawinskiego 5b, 02-106 Warsaw, Poland, HOWARD STONE, Department of Mechanical and Aerospace Engineering, Princeton University, Princeton NJ 08544, USA — Dynamics of particles in a water-base liquid is a very important subject of research from the point of view of biological, medical and industrial applications. Motion of microorganisms, biopolymers, proteins or artificial particles immersed in a flowing liquid is complex and such systems have numerous applications but, on the other hand, the dynamics has not been yet very well understood. In our paper we performed numerical simulations of a bead-spring model chain to investigate the dynamics of long and flexible elastic fibers in a steady shear flow. For a class of rather open conformations and different parameters of flexibility, we identify two district topological transitions with different final size, shape, and orientation. Through further analysis we identify slipknots in the chain. We also analyzed evolution of the fibers which initially form ”open” trefoils for different chain flexibilities and initial orientations with respect to the flow direction. We found examples, which illustrate that the shear flow can unknot a flexible chain and then knot it again; this phenomenon sometimes repeats several times.

5:29PM E23.00004 Flow-Induced Stiffness Enables Torsional Oscillations in a Two-Degree-of-Freedom, Flexibly-Mounted and Free-to-Rotate Rigid Plate, PARIYA POURAZRARM, YAHYA MODARRES-SADEGHI, University of Massachusetts Amherst — We study flow-induced oscillations of a flexibly-mounted rigid flat plate placed in water, for a plate with two degrees of freedom in the torsional and transverse directions, with no torsional spring, i.e., no structural stiffness in the torsional direction. At low flow velocities, the plate rotates in the clockwise or counterclockwise direction several times while oscillating in the transverse direction. The frequency of these full rotations converges to a constant number for the majority of flow velocities. At higher flow velocities, the full rotations stop and the plate starts to oscillate in the torsional direction as well, as if there existed a torsional spring. It is concluded that these oscillations in the torsional direction are made possible due to the flow-induced stiffness, since there is no structural stiffness in that direction.

1 The support provided by the National Science Foundation, grant CBET-1437988 is acknowledged.

5:42PM E23.00005 Self-propulsion of a heaving and pitching flexible flag, BOYOUNG KIM, SUNG GOON PARK, HYUNG JIN SUNG, KAIST — Flapping motions of flexible flags are widespread in nature. Birds, fish, and insects use their wings, fins, or bodies to stay afloat and to advance forward in the surrounding fluids. In the present study, a self-propelled flexible flag with heaving and pitching motions in a quiescent flow has been simulated by using the immersed boundary method. The flexible flag can move freely in the horizontal direction and the body of the flexible flag moves passively along the head. The motion of the head of the flag was described as a harmonic oscillation in the vertical direction. The motion of the angle of the head was described as a harmonic oscillation with a moving clamped condition for the heaving and pitching flag. The cruising speed and the swimming efficiency of the self-propelled flag were determined as functions of the bending coefficient (\( \gamma \)), the heaving amplitude (\( A_h \)), the pitching amplitude (\( A_p \)), the heaving frequency (\( St \)), and the phase difference (\( \Delta \phi \)) between the oscillations. We conducted a parametric study on the optimized the cruising speed and the swimming efficiency with respect to \( \gamma \), \( St, A_h, A_p, \) and \( \Delta \phi \).

5:55PM E23.00006 Fluid-Structure Interaction Study on a Pre-Buckled Deformable Flat Ribbon, LAUREN FOVARGUE, EHSAN SHAMS, AMY WATTERSON, Rensselaer Polytech Inst, DAVE CORSON, Altair Engineering, Inc., BENJAMIN FILARDO, DANIEL ZIMMERMAN, BOB SHAN, Pliant Energy Systems LLC, ASSAD OBERAI, Rensselaer Polytech Inst — A Fluid-Structure Interaction study is conducted for the flow over a deformable flat ribbon. This mechanism, which is called ribbon frond, maybe used as a device for pumping water and/or harvesting energy in rivers. We use a lower dimensional mathematical model, which represents the ribbon as a pre-buckled structure. The surface forces from the fluid flow, dictate the deformation of the ribbon, and the ribbon in turn imposes boundary conditions for the incompressible Navier-Stokes equations. The mesh model is handled using an Arbitrary Lagrangian-Eulerian (ALE) scheme and the fluid-structure coupling is handled by iterating over the staggered governing equations for the structure, the fluid and the mesh. Simulations are conducted at three different free stream velocities. The results, including the frequency of oscillations, show agreement with experimental data. The vertical structures near the surface of the ribbon and its deformation are highly correlated. It is observed that the ribbon motion exhibits deviation from a harmonic motion, especially at lower free stream velocities. The behavior of the ribbon is compared to swimming animals, such as eels, in order to better understand its performance.

1 The authors acknowledge support from ONR SBIR Phase II, contract No. N0001412C0064 and USDA, NIFA SBIR Phase I, contract No. 2013-33610-20836 and NYSERDA PON 2569, contract No. 30364.

Sunday, November 22, 2015 4:50PM - 6:08PM — Session E24 Biofluids: Vesicles and Micelles 302 - Tsorg-Whay Pan, University of Houston

4:50PM E24.00001 The dynamics of inextensible capsules in shear flow under the effect of the natural state, TSORG-WHAY PAN, Univ of Houston, XITING NIU, None, ROLAND GLOWINSKI, Univ of Houston — The effect of the natural state on the motion of an inextensible capsule in two-dimensional shear flow has been studied numerically. The energy barrier based on such natural state plays a role for having the transition between two well-known motions, tumbling and tank-treading (TT) with the long axis oscillating about a fixed inclination angle (a swinging mode), when varying the shear rate. Between tumbling and TT with a swinging mode, the intermittent region has been obtained for the capsule with a biconcave rest shape. The estimated critical value of the swelling ratio for having the intermittent region is \( \frac{\gamma}{\gamma_0} < 0.7 \), i.e., the capsule with the rest shape closer to a full disk has no intermittent behavior. The capsule intermittent behavior is a mixture of tumbling and TT. Just like the TT with a swinging mode, which can be viewed as TT with an inextensible tumbling, the membrane tank-treads backward and forward within a small range while tumbling. The transition between tumbling and TT with a swinging mode has been studied.

1 This work is supported by an NSF grant DMS-0914788.
5:03PM E24.00002 Off-plane motion of an oblate capsule in simple shear flow. ANNE-VIRGINIE SALSC, CLAIRE DUPONT, FABIEN DELAHAYE, DOMINIQUE BARTHES-BIESEL, CNRS – UTC University (Compiegne, France), BIOMECHANICS & BIOENGINEERING LAB TEAM — As biomimetic models of red blood cells, non-spherical liquid-core capsules have received great attention to understand their dynamics in simple shear flow. They are also of interest for drug delivery applications having higher diffusion properties than spherical ones. Most studies have modeled the capsule motion placing the revolution axis in the shear plane, which is an equilibrium configuration in Stokes flow conditions and thus a special case. The present objectives is to determine the stability of the equilibrium configurations of oblate capsules and investigate the effects of the capillary number \( \text{Ca} \), inner-to outer viscosity ratio \( \lambda \) and initial orientation. To solve the fluid-structure interaction problem, we use a numerical model coupling a finite element method for the capsule deformation with a boundary integral method for the internal and external flows. The equilibrium motions are found to be independent of the capsule initial inclination and to depend only on \( \text{Ca} \). The tumbling and swinging regimes (characterized by the revolution axis in the shear plane) are found to be stable only until \( \text{Ca} \sim 0.9 \). Above, the capsule takes a rolling motion with its revolution axis normal to the shear plane. For \( \lambda > 4 \), only tumbling is stable at low \( \text{Ca} \) and rolling at higher \( \text{Ca} \).

5:16PM E24.00003 Quantifying mixing in vesicle suspensions using numerical simulations in two dimensions. GOKBERK KABACAOGLU, GEORGE BIROS, BRYAN QUAIPE, Univ of Texas, Austin — Vesicles, which resist bending and are locally inextensible, serve as an experimental and numerical proxy for red blood cells. In this work, we study the effect of the presence of vesicles to mixing. The motivating application is the study of transport phenomena in microcirculation. We investigate transport specifically in a Couette apparatus, which is governed by an advection-diffusion equation, and we consider mixing in the absence and presence of vesicles using numerical simulations in two dimensions. The advection-diffusion equation is discretized spectrally in space, and with a second-order L-stable Strang splitting in time. To our knowledge, there are no universally accepted measures of mixing. Here, we study two measures: the “mix-norm” defined by a Sobolev norm of negative index and a standard moment fluctuation of the transported species. We compare experimentally in microchannels under flow condition, the pure hydrodynamic cluster formation of RBCs and the cluster formation of RBCs in the presence of macromolecules inducing aggregation. The results reveal strong differences in the cluster morphology. Emphasizing on the case of clusters formed by two cells, the surface to surface interdistances between the cells in the different solutions shows a bimodal distribution. Numerical simulations based on the boundary integral method showed a good agreement with the experimental findings.

5:29PM E24.00004 Hydrodynamic and macromolecules induced clusters of red blood cells in microcapillary flow. VIVIANA CLAVEIRA, OTHMANE AOUANE, Saarland University, GWENNOU COUPIER, CHAOUQI MISBAH, University Grenoble Alpes, MANOUK ABKARIAN, University Montpellier 2, CHRISTIAN WAGNER, Saarland University — Recent studies have shown that fibrinogen and other fibrinogen-like synthetic polymers are able to aggregate red blood cells (RBCs) in microcapillaries, leading to the formation of large vesicle clusters. The mechanisms of interactions induced by the presence of macromolecules in the cluster formation has not been established. In order to elucidate this mechanism, we compare experimentally in microchannels under flow condition, the pure hydrodynamic cluster formation of RBCs and the cluster formation of RBCs in the presence of macromolecules inducing aggregation. The results reveal strong differences in the cluster morphology. Emphasizing on the case of clusters formed by two cells, the surface to surface interdistances between the cells in the different solutions shows a bimodal distribution. Numerical simulations based on the boundary integral method showed a good agreement with the experimental findings.

5:42PM E24.00005 Multiscale modeling of mechanosensing channels on vesicles and cell membranes in 3D constricted flows and shear flows. ZHANGLI PENG, University of Notre Dame, ON SHUN PAK, Santa Clara University, YUAN-NAN YOUNG, New Jersey Institute of Technology, ALLEN LIU, University of Michigan, HOWARD STONE, Princeton University — We investigate the gating of mechanosensing channels (Mscl) on vesicles and cell membranes under different flow conditions using a multiscale approach. At the cell level (microns), the membrane tension is calculated using a 3D two-component whole-cell membrane model based on dissipative particle dynamics (DPD), including the cortex cytoskeleton and its interactions with the lipid bilayer. At the Mscl level (nanometers), we predict the relation between channel gating and the membrane tension obtained from a cell-level model using a semi-analytical model based on the bilayer hydrophobic mismatch energy. We systematically study the gating of Mscl channels on vesicles and cell membranes in constricted channel flows and shear flows, and explore the dependence of the gating on flow rate, cell shape and size. The results provide guidance for future experiments in inducing Mscl opening for various purposes such as drug delivery.

5:55PM E24.00006 Resolving lubrication layers in immersed boundary method simulations of vesicular transport in dendritic spines. THOMAS FAI, Harvard University, REMY KUSTERS, Eindhoven University of Technology, CHRIS RYCROFT, Harvard University — Our understanding of how neuronal connections in the brain are maintained and reorganized is being revolutionized by new experimental and computational techniques. Existing high-resolution 3D images show that neuronal axons often terminate onto micron-sized structures known as dendritic spines, which are characterized by their thin necks and bulbous heads. Vesicles containing membrane receptors must deform significantly to squeeze into the bulbous heads of the spines, but more quantitative estimates of the force and energy required are still lacking. We have used three-dimensional immersed boundary method simulations to capture the fluid dynamics of vesicle transport into spines. We vary the applied force and neck geometry to identify the region in phase space in which the vesicle can squeeze into the spine. These results are compared to pass-stuck diagrams computed previously in the case of vesicles squeezing through open channels with rigid walls. The resulting force estimates are found to be consistent with the physiological density of motor proteins. Resolving the thin lubricating layers between the vesicles and spine poses significant numerical challenges, and we have used elements from lubrication theory to help resolve these boundary layers.

---

Session E25 Superfluids 304 - Scott Strong, Colorado School of Mines

4:50PM E25.00001 Nonlinear Binormal Flow of Vortex Filaments. SCOTT STRONG, LINCOLN CARR, Colorado School of Mines, Department of Applied Physics — With the current advances in vortex imaging of Bose-Einstein condensates occurring at the Universities of Arizona, São Paolo and Cambridge, interest in vortex filament dynamics is experiencing a resurgence. Recent simulations, Salmon (2013), depict dissipative mechanisms resulting from vortex ring emissions and Kelvin wave generation associated with vortex tangles. As the local isles induction approximation fails to capture vortices on long timescales, it lacks a similar dissipative mechanism. On the other hand, Strong&Carr (2012) showed that the exact representation of the velocity field induced by a curved segment of vortex contains higher-order corrections expressed in powers of curvature. This nonlinear binormal flow can be transformed, Hasimoto (1972), into a fully nonlinear equation of Schrödinger type. Continued transformation, Madelung (1926), reveals that the filament’s square curvature obeys a quasilinear scalar conservation law with source term. This implies a broader range of filament dynamics than is possible with the integrable linear binormal flow. In this talk we show the affect higher-order corrections have on filament dynamics and discuss physical scales for which they may be witnessed in future experiments.

---

1Partially supported by NSF
2Partially supported by NSF
ment of skin friction in fluid flow

of existing pressure sensors. This will present new observations of reconnection events and analysis comparing vortex reconnection behavior in three dimensions to previous work that observed such events in two-dimensional projection. In particular, we discuss the power law scaling of vortex separation as a function of time and the effect of the initial angle of separation between the vortex filaments.

5:16PM E25.00003 Vortex knottiness in superfluids, HRIDESH KEDIA, The University of Chicago, DUSTIN KLECKNER, University of California, Merced, DAVIDE PROMENTI, University of East Anglia, WILLIAM IRVINE, The University of Chicago — Recent work has demonstrated that linked and knotted vortices will spontaneously unknot or untie in both classical fluids and superfluids. This effect would appear to jeopardize any notion of conservation of fluid topology (helicity), but this need not be the case: vortices can transfer their knottedness to helical coils, preserving some measure of the original topology. We ask how this notion of topology preservation behaves in the context of collections of vortices with topology. We address this question by numerical simulations of superfluid vortices in the Gross-Pitaevskii equation.

5:29PM E25.00004 Hydrodynamic Decay of Decorated Quantum Vortex Rings, LUCA MORICONI, Instituto de Física - Universidade Federal do Rio de Janeiro — The decay of quantum vortex rings in HeII, visualized with the help of solid hydrogen particles trapped in their cores, has been a problematic issue within the two-fluid model of superfluidity: the large drag exerted on the vortex rings by the flow of normal fluid past the hydrogen particles would ultimately lead to decay times that mismatch the ones observed in the laboratory. We discuss a phenomenological solution of this puzzle, which is based on the fact that the vortex ring energy loss is accounted for not only by mutual friction, but also by the viscous dissipation and sweeping of the flow structures produced from the vortex ring backreaction on the normal component of the surrounding superfluid.

1Work partially supported by CNPq and FAPERJ

Sunday, November 22, 2015 4:50PM - 6:08PM —
Session E26 Experiments: Sensing and Field Measurements 306 - Jianzhong Zhu, University of Virginia

4:50PM E26.00001 Expandable and retractable self-rolled structures based on metal/polymer thin film for flow sensing, JIANZHONG ZHU, CARL WHITE, MEHDI SAADAT, HILARY BART-SMITH, University of Virginia — Most aquatic animals such as fish rely heavily on their ability to detect and respond to ambient flows in order to explore and inhabit various habitats or survive predator-prey encounters. Fish utilize neuromasts in their skin surface and lateral lines in their bodies to align themselves while swimming upstream for migration, avoid obstacles, reduce locomotion cost, and detect flow variations caused by local environmental peculiarities. Neuromasts, also known as lateral-line systems, are sensitive to small frictions along the flow direction but thick (e.g., 5 mm) to be robust along all other directions. This compact, low profile, and highly dynamic Sensing Applications, JEFF DUSEK, MICHAEL TRIANTAFYLOU, JEFFREY LANG, Massachusetts Institute of Technology — Shallow, turbid, and highly dynamic coastal waters provide a challenging environment for safe and reliable operation of marine vehicles faced with a distinct environmentally driven perceptual deficit. In nature, fish have solved this perplexing sensory problem and exhibit an analogy and assumption but tend to suffer from instrumentation challenges, such as low sensing resolution or misalignments. Recently, silicon micromachined floating plates showed good resolution and perfect alignment but were too small for general purposes and too fragile to attach other surface samples repeatedly. In this work, we report a skin friction sensor consisting of a monolithic floating plate and a high-resolution optical encoder to measure its displacement. The key for the high resolution is in the suspension beams, which are very narrow (e.g., 0.25 mm) to sense small frictions along the flow direction but thick (e.g., 5 mm) to be robust along all other directions. This compact, low profile, and complete sensor is easy to use and allows repeated attachment and detachment of surface samples. The shear-stress sensor has been tested in water tunnel and towing tank at different flow conditions, showing high sensing resolution for skin friction measurement.

1Supported by National Science Foundation (NSF) (No. 1330060) and Defense Advanced Research Projects Agency (DARPA) (No. HR0011-15-2-0021)

5:03PM E26.00002 High-resolution compact shear stress sensor for direct measurement of skin friction in fluid flow, MUCHEN XU, CHANG-JIN "CJ" KIM, University of California, Los Angeles — The high-resolution measurement of skin friction in complex flows has long been of great interest but also a challenge in fluid mechanics. Compared with indirect measurement methods (e.g., laser Doppler velocimetry), direct measurement methods (e.g., floating element) do not involve any analogy and assumption but tend to suffer from instrumentation challenges, such as low sensing resolution or misalignments. Recently, silicon micromachined floating plates showed good resolution and perfect alignment but were too small for general purposes and too fragile to attach other surface samples repeatedly. In this work, we report a skin friction sensor consisting of a monolithic floating plate and a high-resolution optical encoder to measure its displacement. The key for the high resolution is in the suspension beams, which are very narrow (e.g., 0.25 mm) to sense small frictions along the flow direction but thick (e.g., 5 mm) to be robust along all other directions. This compact, low profile, and complete sensor is easy to use and allows repeated attachment and detachment of surface samples. The shear-stress sensor has been tested in water tunnel and towing tank at different flow conditions, showing high sensing resolution for skin friction measurement.

1Supported by National Science Foundation (NSF) (No. 1330060) and Defense Advanced Research Projects Agency (DARPA) (No. HR0011-15-2-0021)

5:16PM E26.00003 Bio-Inspired Pressure Sensitive Foam Arrays for use in Hydrodynamic Sensing Applications, JEFF DUSEK, MICHAEL TRIANTAFYLOU, JEFFREY LANG, Massachusetts Institute of Technology — Shallow, turbid, and highly dynamic coastal waters provide a challenging environment for safe and reliable operation of marine vehicles faced with a distinct environmentally driven perceptual deficit. In nature, fish have solved this perplexing sensory problem and exhibit an intimate knowledge of the near-body flow field. This enhanced perception is mediated by the ability to discern and interpret hydrodynamic flow structures through the velocity and pressure sensing capabilities of the fish’s lateral line. Taking cues from biological sensory principles, highly conformal pressure sensor arrays have been developed utilizing a novel piezoresistive carbon black-PDMS foam active material. By leveraging the low Young’s modulus and watertight structure of closed-cell PDMS (silicone) foam, the sensor arrays are well suited for hydrodynamic sensing applications and prolonged exposure to fluid environments. Prototype arrays were characterized experimentally using hydrodynamic stimuli inspired by biological flows, and were found to exhibit a high degree of sensitivity while improving on the flexibility, robustness, and cost of existing pressure sensors.
5:29PM E26.00004 Daylight Operable PIV for Use in the Field. LARRY BROCK, JIAN SHENG, Texas Tech Univ — Particle Image Velocimetry (PIV) is widely used in laboratory scale studies, however, has considerable difficulties for application in the field. The issue mainly arise due to the presence of background sunlight and undesirable environmental conditions. To overcome the strong ambient light during the double exposure PIV operation, one must reduce substantially the total ambient illuminations to the tracer particle scattering. To achieve the above mentioned objective, we increase the scattering by using a pulsed laser with short pulse width (<7ns) at the same time shortening the image exposure. The laser light is introduced via fiber optic cable where the laser is located in a remote location and delivered to the encapsulated pod to form a thin collimated 50 x 1 mm sheet, latter being the thickness. The sheet is then reflected between a series of mirrors to create a light-in-flight. The light sheet is overlapped slightly between the reflections and illuminates the entire field of view in the time of camera exposure (e.g. 1us). The DOPIV system is capable of measuring 2D velocity in a .5 m X .5 m field of view with 0.2 mm spatial resolution and 7.6 mm vector spacing. The bench-top and fields experiments are performed to demonstrate the feasibility of the systems in understanding near surface transport phenomenon such as wake in a wind farm, atmospheric/oceanic boundary layer, etc.

5:42PM E26.00005 Adapting unmanned aerial vehicles for turbulence measurement. BRANDON WITTE, JACOB HELVEY, JON MULLEN, MICHAEL THAMANN, SEAN BAILEY, University of Kentucky — We describe the approach of using highly instrumented and autonomous unmanned aerial vehicles (UAVs) to spatially interrogate the atmospheric boundary layer’s turbulent flow structure. This approach introduces new capabilities not available in contemporary micro-meteorological measurement techniques such as instrumented towers, balloons, and manned aircraft. A key advantage in utilizing UAVs as an atmospheric turbulence research tool is that it reduces the reliance on assumptions regarding temporal evolution of the turbulence inherent within Taylor’s frozen flow hypothesis by facilitating the ability to spatially sample the flow field over a wide range of spatial scales. In addition, UAVs offer the ability to measure in a wide range of boundary conditions and distance from the earth’s surface, the ability to gather many boundary layer thicknesses of data during brief periods of statistical quasi-stationary, and the ability to acquire data where and when it is needed. We describe recent progress made in manufacturing purpose-built airframes and adapting pre-fabricated airframes for these measurements by integrating sensors into those airframes and developing data analysis techniques to isolate the atmospheric turbulence from the measured velocity signal.

5:55PM E26.00006 Characterization of floating element balance for field panel testing. J. TRAVIS HUNSUCKER, HARRISON GARDNER, GEOFFREY SWAIN, Center for Corrosion and Biofouling Control, Florida Institute of Technology — Multiple experiments were performed to investigate and characterize the uncertainty and bias of a through-hull flush mounted floating element balance designed to measure the hydrodynamic drag forces of biofouling and marine coatings on 25 x 30 cm test panels. The instrument is located in a wet well on the aft portion of a 27’ Chris Craft Commander. Testing occurs over a series of speeds ranging from a Froude number of 0.50-2.20 on calm days (force 3 or less) in waters along the central east coast of Florida. Recent modifications have been made to the instrumentation in an effort to improve the overall accuracy of the system. This study compares frictional drag measurements of the floating element balance to those obtained using the Clauser chart and Preston tube methods for a smooth surface. Boundary layer velocity profiles are examined to understand the nature of the flow over the testing section. Roughness function values for 60 and 220 grit sandpaper were calculated from data obtained using the floating element balance. These values were compared with previous work to examine the overall bias of the methodology. Repeat measurements for a smooth panel were analyzed to characterize the overall uncertainty in the system.

Sunday, November 22, 2015 4:50PM - 6:08PM Session E27 Experiments: Temperature and Velocity Measurements 308 - Shahram Pouya, Michigan State University

4:50PM E27.00001 Multi-photon Molecular Tagging Thermometry with Femtosecond Excitation (FemtoMTT). SHAHRAM POUYA, Michigan State University, ALEXANDER VAN RHIJN, NTS Optical B.V., ALIREZA SAFARIPOUR, MARCOS DANTUS, MANOOCHEHR KOOCHEFSAHANI, Michigan State University — Following our earlier report of first Molecular Tagging Velocimetry (MTV) measurement under nonlinear resonant femtosecond excitation in an aqueous flow, we present results of Molecular Tagging Thermometry (MTT) in a simple jet flow using femtosecond excitation. The two-photon absorption process of a phosphorescent supramolecule allows for simultaneous velocity and temperature measurement using a pair of images obtained during the lifetime of the tracer. Results reproduce the tracer temperature response under typical single photon excitation, while providing potential for high rep-rate capabilities for simultaneous velocimetry and thermometry in aqueous flows and eliminating the need for short wavelength UV excitation source and UV optical access in flow facilities.

5:03PM E27.00002 Development of a time-resolved luminescent imaging technique for unsteady temperature measurement in thermal fluid phenomena. KAZUNOBU KOBAYASHI, Osaka Gas Co., Ltd./University of Notre Dame, HIROTAKA SAKAUE, University of Notre Dame — This study presents a time-resolved luminescent imaging that uses two-luminescent outputs to extract the temperature information from an acquired image. This imaging technique is applied to measure the temperature distribution for unsteady thermal fluid phenomena. The thermographic phosphors are seeded into the flow and are excited by a laser sheet as an illumination source. The luminescent images from phosphors are captured to obtain the time-resolved temperature profile by using a fast frame-rate camera as an image acquisition unit. In this study, this technique have been carried out for measurements of two-dimensional gas-phase or boiling water temperature. In the presentation, a current status of this measurement will be presented.
5:16PM E27.00003 Development of a 3-wire probe for the simultaneous measurement of turbulent velocity, concentration and temperature fields. ALAIS HEYES, LAURENT MYDLARSKI, McGill University — The present work focuses on the design and optimization of a probe used to simultaneously measure the velocity, concentration and temperature fields in a turbulent jet. The underlying principles of this sensor are based in thermal-anemometry techniques, and the design of this 3-wire probe builds off the previous work of Sirivat and Warhaft, *J. Fluid Mech.*, 1982. In the first part of this study, the effect of different overheat ratios in the first two wires (called the “interference” or “Way-Libby” probe – used to infer velocity and concentration) are investigated. Of particular interest is their effect on the quality of the resulting calibration, as well as the measured velocity and concentration data. Four different overheat ratio pairs for the two wires comprising the interference probe are studied. In the second part of this work, a third wire, capable of detecting temperature fluctuations, is added to the 3-wire probe. The optimal configuration of this probe, including wire type and overheat ratio for the third wire, is studied and the simultaneously-measured velocity, concentration, and temperature data (e.g. spectra, PDFs) for different probe configurations are presented.

1Supported by the Natural Sciences and Engineering Research Council of Canada (Grant 217184)

5:29PM E27.00004 IV as a temperature measurement tool. GHANEM F. OWEIS, Department of Mechanical Engineering, American University of Beirut, PO Box 11-0236, Beirut, Lebanon — In particle image velocimetry (PIV), a camera records time-lapse snapshot images of the positions of particles embedded in a fluid, which faithfully trace the flow path. Cross correlating sequential particle image pairs results in 2D maps of the particle displacement and velocity fields. Here, the same PIV method is extended to temperature measurements in viscoelastic material. The motivation originates in a need for tissue temperature measurements in hyperthermia therapies such as laser ablation eye surgery and high intensity focused ultrasound (HIFU) tumor ablation. Micron sized particles are embedded in an optically clear tissue mimicking phantom, illuminated with a laser sheet, and imaged with a CCD camera. When the phantom is subjected to heating from a focused ultrasound beam, the particles remain stationary, but not their spatial distribution in the recorded images. The images manifest particle displacements commensurate with alterations in the temperature distribution from heating. The underlying principle behind the thermometric capability of IV is discussed. Temperature changes can be detected with high sensitivity, and the method works best with spatially localized temperature distributions.

5:42PM E27.00005 A Composition-Independent Thermometry Technique for Gaseous Mixtures in Reacting Environments. DOMINIC ZELENAK, Graduate Student, WILLIAM SEALY, Undergraduate Student, TURBULENT SHEAR FLOW LABORATORY TEAM — Temperature is an important thermochemical property that holds the key to uncovering several combustion phenomena such as pollutant formation, flame extinction, and heat release. In a practical combusting environment, the local composition is unknown, hindering the effectiveness of established non-intrusive thermometry techniques. This study aims to offset this limitation by developing a laser-based thermometry technique that does not require prior knowledge of the local composition. The Turbulent Shear Flow Laboratory (TSFL) at North Carolina State University is currently using a combination of krypton planar-laser induced fluorescence (Kr-PLIF) and Rayleigh scattering to measure temperature in reacting environments. Initial work by TSFL has studied Kr-PLIF lineshape properties of several combustion species to obtain scaling for the collisional broadening parameters based upon the properties of Kr and the surrounding environment. This information will be used to demonstrate the mean temperature profile of a 1D lean premixed CH,"flammable gas, flame exhaust at different downstream distances with multiple equivalence ratios for a wide range of temperatures and local compositions. Validation of the proposed technique will be made using Rayleigh scattering temperature measurements.

1 NSF Grant CBET 1511216

5:55PM E27.00006 Preparation and Application of Temperature Sensitive Paintings. CHI LI, Peking University — Temperature sensitive painting (TSP) is a rapidly developing surface optical measurement technology, which uses temperature sensitive fluorescent probe molecules to obtain the temperature distribution on the surface of the model. Two different types of TSP material are prepared to apply in fluid mechanical experiments. Rhodamine is used as fluorescent and acetone as solvent for the first recipe, while rare earth material as fluorescent and zirconia as solvent for the second recipe. With proper calibration, surface temperature nephogram and temperature gradient nephogram is obtained based on the measured light intensity data, and transition location and heat flux is analyzed. Double layer - multi component TSP measurement technology and more strict calibration will be developed in the near future to get more precise heat flux distribution.

Sunday, November 22, 2015 4:50PM - 6:08PM –
Session E28 Wind Turbines: Actuator Lines/Discs 309 - Ralf Deiterding, University of Southampton

4:50PM E28.00001 Predictive simulation of wind turbine wake interaction with an adaptive lattice Boltzmann method for moving boundaries. RALF DEITERDING, University of Southamp-ton – Aerodynamics and Flight Mechanics Group, STEPHEN L. WOOD, University of Tennessee - Knoxville, The Bredesen Center — Operating horizontal axis wind turbines create large-scale turbulent wake structures that affect the power output of downwind turbines considerably. The computational prediction of this phenomenon is challenging as efficient low dissipation schemes are necessary that represent the vorticity production by the moving structures accurately and are able to transport wakes without significant artificial decay over distances of several rotor diameters. We have developed the first version of a parallel adaptive lattice Boltzmann method for large eddy simulation of turbulent weakly compressible flows with embedded moving structures that considers these requirements rather naturally and enables first principle simulations of wake-turbine interaction phenomena at reasonable computational costs. The presentation will describe the employed algorithms and present relevant verification and validation computations. For instance, power and thrust coefficients of a Vestas V27 turbine are predicted within 5% of the manufacturers specifications. Simulations of three Vestas V27-225kW turbines in triangular arrangement analyze the reduction in power production due to upstream wake generation for different inflow conditions.
5:03PM E28.00002 A new class of actuator surface models incorporating wind turbine blade and nacelle geometry effects

It was shown by Kang, Yang, and Sotiropoulos (Journal of Fluid Mechanics 744 (2014): 376-403) that the nacelle has significant effects on the turbine wake even in the far wake region, which the standard actuator line model is not able to predict. We develop a new class of actuator surface models for the blades and nacelle, which is able to resolve the effects of both tip vortices and nacelle vortex. The new nacelle model, which is based on distributing forces from the actual nacelle geometry as in the diffused interface immersed boundary methods, is first tested by carrying out LES of the flow past a sphere and demonstrating good agreement with available in the literature DNS results. The proposed model is subsequently validated by simulating the flow past the hydrokinetic turbine used in the simulations of Kang et al. and good agreement with the measurements is demonstrated. Finally, the proposed model is applied to utility scale wind turbines to elucidate the role of nacelle vortex dynamics on turbine wake meandering.

1This work was supported by Department of Energy DOE (DE-EE0002980, DE-EE0005482 and DE-AC04-94AL85000), and Sandia National Laboratories. Computational resources were provided by SNL and MSI.

5:05PM E28.00003 Simulating wind and marine hydrokinetic turbines with actuator lines in RANS and LES

As wind and marine hydrokinetic (MHK) turbine designs mature, focus is shifting towards improving turbine array layouts for maximizing overall power output, i.e., minimizing wake interference for axial-flow or horizontal-axis turbines, or taking advantage of constructive wake interaction for cross-flow or vertical-axis turbines. Towards this goal, an actuator line model (ALM) was developed to provide a computationally feasible method for simulating full turbine arrays inside Navier–Stokes models. The ALM predicts turbine loading with the blade element method combined with sub-models for dynamic stall and flow curvature. These forces are computed using the Blade Element theory to estimate the normal and tangential components (based on the local simulated flow and the blade characteristics). The local velocities are modified using the Glauert term acting over the regularised disk swept by the rotor. Results are presented for the simulation of performance and wake dynamics of axial- and cross-flow turbines and compared with moderate Reynolds number experiments and body-fitted mesh, blade-resolving CFD.

1Work supported by NSF-CBET grant 1150797.

5:29PM E28.00004 Determining the optimal smoothing length scale for actuator line models of wind turbine blades

The actuator line model (ALM) is a widely used tool for simulating wind turbines when performing Large-Eddy Simulations. The ALM uses a smearing kernel $\eta = 1 / \left( \pi^3 / 2 \exp \left( -r^2 / \epsilon^2 \right) \right)$, where $r$ is the distance to an actuator point, and $\epsilon$ is the smoothing length scale which establishes the kernel width, to project the lift and drag forces onto the grid. In this work, we develop formulations to establish the optimum value of the smoothing length scale $\epsilon$, based on physical arguments, instead of purely numerical constraints. This parameter has a very important role in the ALM, to smooth the lift, which may, for example, be related to the chord of the airfoil being studied. In the proposed approach, we compare features (such as vertical pressure gradient) of a potential flow solution for flow over a lifting surface with features of the solution of the Euler equations with a body force term. The potential flow solution over a lifting surface is used as a general representation of an airfoil. The method presented aims to minimize the difference between these features of the flow fields as a function of the smoothing length scale ($\epsilon$), in order to obtain the optimum value.

1This work is supported by NSF (IGERT and IIA-1243482) and computations use XSEDE resources.

5:42PM E28.00005 Enhanced Actuator Line Simulation of a Wind Turbine by including the Conservative Load at the Blade Tip

At the tip of wind turbine blades, the radial bound circulation is transformed into chordwise circulation just before being released as trailing vortex, giving rise to the tip vortex. The force acting on the chordwise circulation contains a radial and a normal component with respect to the blade axis. This load does not contribute to the torque, so it is a conservative load. Due to this, it is disregarded in the engineering tools used for the design of wind turbines. However, as we demonstrated in a previous work, the conservative load might influence the trajectory of the tip vortex. In order to see how this affects the blade loads, in this research work we transform the chordwise circulations with a conservative load model where the conservative load has been included. The conservative load reduces the angle of attack in the tip region as a consequence of the modified tip vortex trajectory. This has a negative influence on the lift and the power output. We conclude that the accuracy of engineering design tools of wind turbines can be improved if the conservative load acting at the tip is considered.

5:55PM E28.00006 Development of an advanced actuator disk model for Large-Eddy Simulation of wind farms

This work aims at improving the fidelity of the wind turbine modelling for Large-Eddy Simulation (LES) of wind farms, in order to accurately predict the loads, the production, and the wake dynamics. In these simulations, the wind turbines are accounted for through actuator disks, i.e. a body-force term acting over the regularised disk swept by the rotor. These forces are computed using the Blade Element theory to estimate the normal and tangential components (based on the local simulated flow and the blade characteristics). The local velocities are modified using the Glauert tip-loss factor in order to account for the finite number of blades; the computation of this correction is here improved thanks to a local estimation of the free stream velocity at every point of the disk. These advanced actuator disks are implemented in a 4th order finite difference LES solver and are compared to a classical Blade Element Momentum method and to high fidelity wake simulations performed using a Vortex Particle-Mesh method in uniform and turbulent flows.

Sunday, November 22, 2015 4:50PM - 5:42PM — Session E29 Nonlinear Dynamics and Waves I — 310 - Oscar Velasco Fuentes, CICESE
within the volume from the development of an inviscid secondary shear instability. Rotation, which can be responsible for the observed eruptions of jets from the lateral walls of the cylinder leading to the cyclones formation.

We show that when ε is increased from low values the forced mode m = 1 grows with the number of vortices (N) but decreases as the vortices’ radius and pitch (a and τ, respectively) increase; in contrast, the rotation velocity Ω grows with N but has a local minimum around τ = 1 for fixed values of N and a.

5:03PM E29.00002 The Method of Decomposition in Invariant Structures: Exact Solutions for N Internal Waves in Three Dimensions, VICTOR MIROSHNIKOV, CMSV, Department of Mathematics, New York — The Navier-Stokes system of PDEs is reduced to a system of the vorticity, continuity, Helmholtz, and Lamb-Helmholtz PDEs. The periodic Dirichlet problems are formulated for conservative internal waves vanishing at infinity in upper and lower domains. Stationary kinematic Fourier (SKF) structures, stationary kinematic Euler-Fourier (SKEF) structures, stationary dynamic Euler-Fourier (SDEF) structures, and SKEF-SDEF structures of three spatial variables and time are constructed to consider kinematic and dynamic problems of the three-dimensional theory of the Newtonian flows with harmonic velocity. Exact solutions for propagation and interaction of N internal waves in the upper and lower domains are developed by the method of decomposition in invariant structures and implemented through experimental and theoretical programming in Maple. Main results are summarized in a global existence theorem for the strong solutions. The SKEF, SDEF, and SKEF-SDEF structures of the cumulative flows are visualized by two-parametric surface plots for six fluid-dynamic variables.

5:16PM E29.00003 Spectral analysis of approximations of Dirichlet-Neumann operators and nonlocal shallow water wave models, ROSA VARGAS-MAGAA, Universidad Autónoma de Mexico, PANAYOTIS PANAYOTAROS, Departamento de Matemáticas y Mecánica, IIMAS-UNAM, Mexico — We study the problem of wave propagation in a long-wave asymptotic regime over variable bottom of an ideal irrotational fluid in the framework of the Hamiltonian formulation in which the non-local Dirichlet-Neumann (DN) operator appears explicitly in the Hamiltonian. We propose a non-local Hamiltonian model for bidirectional wave propagation in shallow water that involves pseudodifferential operators that approximate the DN operator for variable depth. These models generalize the Boussinesq system as they include the exact dispersion relation in the case of constant depth. We present results for the normal modes and eigenfrequencies of the linearized problem. We see that variable topography introduces effects such as steepening of normal modes with increasing variation of depth, as well as amplitude modulation of the normal modes in certain wavelength ranges. Numerical integration shows that the constant depth nonlocal Boussinesq model with quadratic nonlinearity can capture the evolution obtained with higher order approximations of the DN operator. In the case of variable depth we observe certain oscillations in width of the crest and also some interesting textures in the evolution of wave crests during the passage from obstacles.

5:29PM E29.00004 Pattern formation in thin film evolution equations for complex fluids, MARKUS WIECZ, SVETLANA V. GUREVICH, UWIE THIELE, Institute for Theoretical Physics, University of Muenster, Germany — The description of thin layers of complex fluids like suspensions and solutions is often based on so-called thin film evolution equations which are derived from the hydrodynamics of a large system. In this talk we present a systematic approach to construct such models in a gradient dynamics formulation for a free energy accounting for wettability and capillarity. We propose extensions in this framework and apply it to dewetting and dip-coating problems. Using these models, we study pattern formation phenomena in Langmuir-Blodgett transfer experiments.

Sunday, November 22, 2015 4:50PM - 5:55PM
Session E30 DFD: Geophysical Fluid Dynamics: Rotating Flows 311 - Eckart Meiberg, University of California, Santa Barbara

4:50PM E30.00001 Zonal Flow Velocimetry in Spherical Couette Flow using Acoustic Modes, MATTHEW M. ADAMS, ANTHONY R. MAUTINO, DOUGLAS R. STONE, University of Maryland, College Park, SANTIAGO A. TRIANA, Institute of Astronomy, KU Leuven, VEDRAN LEKIC, DANIEL P. LATHROP, University of Maryland, College Park — We present studies of spherical Couette flows using the technique of acoustic mode Doppler velocimetry. This technique uses rotational splittings of acoustic modes to infer the azimuthal velocity profile of a rotating flow, and is of special interest in experiments where direct flow visualization is impractical. We present results comparing observed splittings with those predicted by theory are presented. While the majority of these studies were performed in the 60 cm diameter device using nitrogen gas, some work has also been done looking at acoustic modes in the 3 m diameter liquid sodium spherical Couette experiment. Prospects for measuring zonal velocity profiles in a wide variety of experiments are discussed.

5:03PM E30.00002 Shear secondary instability in a precessing cylinder flow, WALEED MOUHALI, ECE Paris, THIERRY LEHNER, Luth, Observatoire de Meudon, ATER COLLABORATION — For a certain value of the forcing parameter, cyclones regime has been observed in our experiment involving water in a precessing cylinder. They result from an instability. We propose here to study the nature of this so-called instability. We consider first the mode coupling of two inertial waves with azimuthal wavenumber m=0 and m=1 (mode forced by the precession) in the inviscid regime (at high Re number limit) creates a differential rotation regime which has been observed in the same experiment at small enough Poincaré number ε (ratio of the precession to the rotation frequencies). Secondly, the radial profile of the corresponding axial mean flow vorticity shows an inflexion point leading to a localized inflectional secondary instability. We show that when ε is increased from low values the forced mode m=0 becomes the most unstable in this induced differential rotation, which can be responsible for the observed eruptions of jets from the lateral walls of the cylinder leading to the cyclones formation within the volume from the development of an inviscid secondary shear instability.
and astrophysical flows. At asymptotically small Rossby numbers features of fully developed turbulence. How this scaling is affected by a background rotation is still a controversial issue with importance for geo

dense, CEA Saclay, CNR, Gif-sur-Yvette, France, FREDERIC MOISY, Laboratoire FAST, CNRS, Université Paris-Sud, Orsay, France — The

5:29PM E30.00004 Influence of the multipole order of the source on the decay of an inertial wave beam in a rotating fluid, NATHANAEL MACHICOANE, PIERRE-PHILIPPE CORTET, Laboratoire FAST, CNRS, Université Paris-Sud, Orsay, France, BRUNO VOISIN, Laboratoire LEGI, CNRS, Université Grenoble Alpes, Grenoble, France, FREDERIC MOISY, Laboratoire FAST, CNRS, Université Paris-Sud, Orsay, France — Inertial wave beams emitted from localized sources are relevant to a broad range of geo and astrophysical flows. These beams are excited at critical lines, where the local slope of solid boundaries equals the propagation angle of the wave, in rotating fluid domains affected by a global harmonic forcing (e.g. precession, libration, tidal motion). We show here theoretically and experimentally that the decay of the amplitude of such wave beams depends on the multipole order of the source. We analyze the far-field viscous decay of a two-dimensional inertial wave beam emitted by a harmonic line source in a rotating fluid. By identifying the relevant conserved quantities along the wave beam, we show how the beam structure and decay exponent are governed by the multipole order of the source. Two wavemakers are considered experimentally, a pulsating and an oscillating cylinder, aiming to produce a monopole and a dipole source, respectively. The relevant conserved quantity which discriminates between these two sources is the instantaneous flow rate along the wave beam, which is non-zero for the monopole and zero for the dipole. For each source, the beam structure and decay exponent, measured using particle image velocimetry, are found in good agreement with the predictions.

5:42PM E30.00005 Do inertial wave interactions control the rate of energy dissipation of rotating turbulence? , PIERRE-PHILIPPE CORTET, ANTOINE CAMPAGNE, NATHANAEL MACHICOANE, Laboratoire FAST, CNRS, Université Paris-Sud, Orsay, France, BASILE GALLET, Laboratoire SPHYNX, Service de Physique de l’Etat Condense, CEA Saclay, CNR, Gif-sur-Yvette, France, FREDERIC MOISY, Laboratoire FAST, CNRS, Université Paris-Sud, Orsay, France — The scaling law of the energy dissipation rate, \( \epsilon \propto U^3/L \) (with \( U \) and \( L \) the characteristic velocity and lengthscale), is one of the most robust features of fully developed turbulence. How this scaling is affected by a background rotation is still a controversial issue with importance for geo and astrophysical flows. At asymptotically small Rossby numbers \( \text{Ro} = U/L \Omega \), i.e. in the weakly nonlinear limit, wave-turbulence arguments suggest that \( \epsilon \) should be reduced by a factor \( \text{Ro} \). Such scaling has however never been evidenced directly, neither experimentally nor numerically. We report here direct measurements of the injected power, and therefore of \( \epsilon \), in an experiment where a propeller is rotating at a constant rate in a large volume of fluid rotating at \( \Omega \). In co-rotation, we find a transition between the wave-turbulence scaling at small \( \text{Ro} \) and the classical Kolmogorov law at large \( \text{Ro} \). The transition between these two regimes is characterized from experiments varying the propeller and tank dimensions. In counter-rotation, the scenario is much richer with the observation of an additional peak of dissipation, similar to the one found in Taylor-Couette experiments.

Sunday, November 22, 2015 4:50PM - 6:08PM —

4:50PM E31.00001 Direct Statistical Simulation of Geophysical Flows\(^1\), BRAD MARSTON, Brown University, GREG CHINI, University of New Hampshire, STEVE TOBIAS, University of Leeds — Statistics of models of geophysical and astrophysical fluids may be directly accessed by solving the equations of motion for the statistics themselves as proposed by Lorenz nearly 50 years ago. Motivated by the desire to capture seamlessly multiscale physics we introduce a new approach to such Direct Statistical Simulation (DSS) based upon separating eddies by length scale. Discarding triads that involve only small-scale waves, the equations of motion generalize the quasi-linear approximation (QGL) and are able to accurately reproduce the low-order statistics of a stochastically-driven barotropic jet. Furthermore the two-point statistics of high wavenumber modes close and thus generalize second-order cumulant expansions (CE2) that employ zonal averaging. This GCE2 approach is tested on two-layer primitive equations. Comparison to statistics accumulated from numerical simulation finds GCE2 to be quantitatively accurate. DSS thus leads to new insight into important processes in geophysical and astrophysical flows.

\(^1\)Supported in part by NSF DMR-1306806 and NSF CCF-1048701.

5:03PM E31.00002 Gravity wave emission in an atmosphere-like configuration of the differentially heated rotating annulus experiment, ULRICH ACHATZ, Goethe University Frankfurt, Frankfurt am Main, SEBASTIAN BORCHERT, Deutscher Wetterdienst, Offenbach, Germany, MARK FRUMAN, None, STEFFEHN HIEN, JORAN ROLLAND, Goethe University Frankfurt, Frankfurt am Main, Germany — A finite-volume model of the classic differentially heated rotating annulus experiment is used to study the spontaneous emission of gravity waves (GWs) from jet stream imbalances, which may be an important source of these waves in the atmosphere and for which no satisfactory parameterisation in ion exists. Experiments were performed using a classic laboratory configuration as well as using a much wider and shallower annulus with a much larger temperature difference between the inner and outer cylinder walls. The latter configuration is more atmosphere-like, in particular since the Brunt–Väisälä frequency is larger than the inertial frequency, resulting in more realistic GW dispersion properties. In both experiments, the model is initialised with a baroclinically unstable atmosphere established using a two-dimensional version of the code, and a low-azimuthal-mode baroclinic wave featuring a meandering jet is allowed to develop. Possible regions of GW activity are identified by the horizontal velocity divergence and a modal decomposition of the small-scale structures of the flow. Results indicate GW activity in both annulus configurations close to the inner cylinder wall and within the baroclinic wave. The former is attributable to boundary layer instabilities, while the latter seems to originate in part from spontaneous GW emission from the baroclinic wave.

5:16PM E31.00003 ABSTRACT WITHDRAWN —
waves; an impulsive regime that may share correspondence with the locomotion of water striders. That when the drop bounces off the film, there is a momentum transfer leading to vortex dipole shedding, along with the generation of capillary regime for smaller inclination angles. However, at higher impact angles, puncturing of the film becomes a more common occurrence. We show techniques employed include sodium lamp interferometry to measure film thickness fluctuations and particle tracking velocimetry to measure detailed experimental study of droplet impacts on soap film flow, for a number of film inclination angles and falling heights of the drop. Imaging the impact of a quasi one-dimensional object on a two-dimensional fluid, much like a comet impacting on a thin atmosphere. We present a for static films it gets assimilated within the film, and (c) it pierces through the film. The interaction presents a unique opportunity to explore three distinct types of impact regimes: (a) the drop bounces off the film surface, (b) it coalesces with the downstream flow for a moving film and (c) it pierces through the film. During impact, the drop deforms along with a simultaneous, almost elastic deformation of the film exhibit three fundamental regimes of post-impact dynamics: (a) the drop bounces off the film surface, (b) it coalesces with the downstream flow, and (c) it pierces through the film. During impact, the drop deforms along with a simultaneous, almost elastic deformation of the film transverse to the stream direction. Hence, the governing dynamics for this interaction present the rare opportunity to explore the in-tandem effects of elasticity and hydrodynamics alike. In this talk, we outline the analytical framework to study the drop impact dynamics. The model assumes a deformable drop and a deformable three-dimensional soap film and invokes a parametric study to qualify the three mentioned impact types. The physical parameters include the impact angle, drop impact speed, and the diameters of the drop prior to and during impact when it assumes a deformable drop and a deformable three-dimensional soap film and invokes a parametric study to qualify the three mentioned impact regimes: (a) the drop bounces off the film surface, (b) it coalesces with the downstream flow for a moving film and for static films it gets assimilated within the film, and (c) it pierces through the film. The interaction presents a unique opportunity to explore the impact of a quasi one-dimensional object on a two-dimensional fluid, much like a comet impacting on a thin atmosphere. We present a detailed experimental study of droplet impacts on soap film flow, for a number of film inclination angles and falling heights of the drop. Imaging techniques employed include sodium lamp interferometry to measure film thickness fluctuations and particle tracking velocimetry to measure the velocity field. Film thickness measures approximately 10 microns and the drop diameter is 1 mm. We mostly observe the bouncing-off regime for smaller inclination angles. However, at higher impact angles, puncturing of the film becomes a more common occurrence. We show that when the drop bounces off the film, there is a momentum transfer leading to vortex dipole shedding, along with the generation of capillary waves; an impulsive regime that may share correspondence with the locomotion of water striders.
5:16PM E32.00003 Large bubble entrainment in drop impact. MARIE-JEAN THORAVAL1, King Abdullah University of Science and Technology, Thuwal, Saudi Arabia. YANGFAN LI, National University of Singapore, 9 Engineering Drive 1, Singapore 117576, SIGURDUR T. THORODDSEN, King Abdullah University of Science and Technology (KAUST), Thuwal, 23955-6900, Saudi Arabia. — A drop impacting on a pool of the same liquid can entrap air bubbles in many different ways. A peculiar entrainment was observed by Pumphrey and Elmore (1990) and remained unexplained until now. For a small range of parameters, the cavity produced by the impacting drop spreads radially in a dish-shape and then closes to entrap a bubble larger than the drop. We demonstrate that the large bubble is caused by a vortex ring produced in the liquid during the impact of the drop. We combine experiments and numerical simulations to show that the vortex ring pulls on the interface on the side of the cavity to stretch it radially, explaining the shape of the cavity. Only prolate drops are able to generate large bubbles. This is due to the self-destruction of the vortex earlier during the impact for flatter drops.

1(2) Physics of Fluids Group, University of Twente, Enschede, The Netherlands (3) International Center for Applied Mechanics, Xian Jiaotong University, Xian, China

5:29PM E32.00004 Droplet impact on a liquid pool and bubble entrainment for low Bond numbers. PASCAL SLEUTEL, University of Twente, PEI HSUN TSAL, National Taiwan University, WILCO BOUWHUIS, MARIE-JEAN THORAVAL, CLAAS-WILLEM VISSER, University of Twente, AN-BANG WANG, National Taiwan University, MICHEL VER-SLUIJS, DETLEF LOHSE, University of Twente. — Droplets impacting on a pool of liquid and the subsequent bubble entrainment has been well studied for high Bond numbers where the droplets size is large and velocities are low. Here we study for the first time the droplet impact and bubble entrainment in an entirely new parameter regime (Bo \(\sim 10^{-2} - 10^{-3}\), \(U \sim 6-20\) m/s, \(D \sim 0.08-0.4\) mm). We follow up on the pioneering work of Oguz & Prosperetti, now in the surface tension dominated regime. We predict the bubble entrainment zone by balancing movement of the cavity bottom and droplet inertia with capillary waves enclosing the bubble. Both high-speed imaging experiments and numerical simulations in Gerris validate the model and show the importance of air for smaller droplet sizes.

5:42PM E32.00005 Drainage of the air film during drop impact on flowing liquid films1. ZHIZHAO CHE, OMAR MATAR, Imperial College London. — Immediately upon the impact of a droplet on a liquid or a solid, a thin air cushion is formed by trapping air beneath the droplet. The drainage of the air film is critical in determining the eventual outcome of the impact. Here we propose a model to study the drainage of the gas film between a droplet and a flowing liquid film. The effects of a wide range of parameters influencing the drainage process are studied, such as the fluid viscosities, the surface tension, the velocity of the droplet, the velocity of the liquid film. The results show that the tangential movement of the liquid film can delay the drainage of the air film and promote the bouncing of droplets. This confirms our previous experimental results, which show that during the impact of droplets on low fluid films, the probability of bouncing increases with the Reynolds number of the liquid film.

1EPSRC Programme Grant, MEMPHIS, EP/K0039761/1

5:55PM E32.00006 Ripple Dynamics of Water Entry after Pinch Off. AUSTIN MITUNIEWICZ, BRIAN CHANG, MATT CROSSON, SUNCHWAN JUNG, Virginia Tech, BIOnSPINE FLUID LAB TEAM. — Most research concerning water entry of a projectile focuses on splash during impact and air entrainment during descent. Following pinch off, the air cavity shortens and interfacial rippling develops. In this study, we examine ripple formation induced by projectiles of different shapes under varying kinematic conditions. The amplitude and wavelength of these ripples is determined by the geometry and kinematics of the projectile as well as the cavity pressure. Observations of ripple dynamics demonstrate a close in-phase relationship between the force acting on the projectile and the pressure within the air cavity itself.

Sunday, November 22, 2015 4:50PM - 6:08PM –
Session E33 Drops: Wetting and Spreading II Ballroom A - Sungyon Lee, Texas AM University

4:50PM E33.00001 Drop stability in wind: theory. SUNGYON LEE, Texas A&M University. — Water drops may remain pinned on a solid substrate against external forcing due to contact angle hysteresis. Schmucker and White investigated this phenomenon experimentally in a high Reynolds number regime, by measuring the critical wind velocity at which partially wetting water drops depin inside a wind tunnel. Due to the unsteady turbulent boundary layer, droplets are observed to undergo vortex-shedding induced oscillations. By contrast, the overall elongation of the drop prior to depinning occurs on a much slower timescale with self-similar droplet shapes at the onset. Based on these observations, a simple, quasi-static model of depinning droplet is developed by implementing the phenomenological description of the boundary layer. The resultant model successfully captures the critical onset of droplet motion and is the first of on-going studies that connect the classical boundary layer theory with droplet dynamics.

5:03PM E33.00002 Drop stability in wind: effect of solid protrusion. ALIREZA HOOSHANGINEJAD, BENJAMIN WILCOX, EDWARD WHITE, SUNGYON LEE, Texas A&M University. — We experimentally investigate the inertia-driven onset of droplet depinning behind a solid protrusion inside a wind tunnel. In the high Reynolds number regime, the separation and reattachment of the boundary layer in the presence of the solid protrusion directly lead to complex behavior of the partially wetting water drop as a function of its position. For varying droplet volumes and droplet positions from the protrusion, we measure the critical wind velocity at which the droplet starts to depin. In particular, drops in a certain volume range are observed to reverse their depinning direction at a critical distance from the solid. By coupling the boundary layer characteristics with droplet dynamics, we explain the physical mechanism of the resultant droplet behavior.

5:16PM E33.00003 Simulations of contact angle induced pearling for sliding drops. SCOTT MCCUE, LISA MAYO, TIMOTHY MORONEY, Queensland University of Technology. — Droplets sliding down an incline can develop a corner or a cusp at their rear, or undergo a pearling transition whereby the tail breaks up into a number of smaller satellite droplets. These phenomena have been of interest since the experimental work of Podgorski et al. (2001) Phys Rev Lett 87, 036102. It appears that the critical onset of droplet motion is limited due to the inherent difficulty of minimising contact angle hysteresis, whereby physical or chemical heterogeneities of the substrate cause pinning and distortion of the droplet. By applying a lubrication model with a disjoining pressure term, we investigate these flows numerically in order to further shed light on how certain conditions (such as contact angle) affect the corner-cusp pearling transition.

1We acknowledge support from the ARC Linkage Project LP100200476
where one component has both higher surface tension and higher vapor pressure on a variety of high energy surfaces. We now show how this phenomenon works.

CIRA, ADRIEN BENUSIGLIO, MANU PRAKASH, Stanford University, Dept of Bioengineering — Previously we showed that droplets of aqueous electrolytes can go from super-hydrophilic to super-hydrophobic and back to super-hydrophilic in a matter of seconds, a phenomenon known as superspreading. In this study, we explore the molecular mechanisms that drive superspreading in a way that is applicable to a range of two-component mixtures of miscible liquids. We showed that these mechanisms apply to a range of two-component mixtures of miscible liquids. For larger contact angles, the velocity of the droplet and the eventual occurrence of breakup into smaller droplets are analyzed.

Homsy, JFM, 2010) one of us studied the flow of a droplet on a rigid substrate under the effect of a constant temperature gradient, and under partial wetting and zero-gravity conditions. Three different regimes of flow, that depends on the contact angle and volume of the droplets were reported. Here we introduced the gravity and different substrate-liquid molecular interactions to study its effect on the flow. We observe that for small contact angles, the asymptotic behaviour of the droplet is similar to the one observed for none gravity conditions, no matter the molecular interaction modeled. For larger contact angles, the velocity of the droplet and the eventual occurrence of breakup into smaller droplets are analyzed.

5:55PM E33.00006 Dancing droplets: Chemical space, substrates, and control. NATE CIRA, ADRIEN BENUSIGLIO, MANU PRAKASH, Stanford University, Dept of Bioengineering — Previously we showed that droplets of propylene glycol and water display remarkable properties when placed on clean glass due to an interplay between surface tension and evaporation. (Cira, Benusiglio, Prakash: Nature, 2015). We showed that these mechanisms apply to a range of two-component mixtures of miscible liquids where one component has both higher surface tension and higher vapor pressure on a variety of high energy surfaces. We now show how this rule can be cheated using a simple trick. We go on to demonstrate applications for cleaning, and show how this system works on substrates prepared only with sunlight. We finish by demonstrating active control of droplets, allowing access to a host of new possibilities.

Sunday, November 22, 2015 4:50PM - 6:08PM – Session E35 Drops: Complex Fluids Ballroom B - Jan Guzowski, Princeton University

4:50PM E35.00001 Gel-like double-emulsion droplets1. JAN GUZOWSKI, Princeton University, PIOTR KORCZYK, PIOTR GARSTECKI, Polish Academy of Sciences, HOWARD STONE, Princeton University — We experimentally study the problem of packing of micro-droplets inside a droplet of another immiscible liquid phase. We use microfluidics to encapsulate multiple monodisperse aqueous segments inside a drop of oil. For small numbers N (N~10) of the aqueous droplets and at their volume fraction in oil exceeding the close-packging threshold we observe multiple metastable structures with well-defined point-group symmetries. We attribute the observed metastability to the deformability of the droplets which leads to effective many-body interactions and energy barriers for rearrangement. By changing the composition of the oil phase we find that when the surface tensions of the droplets and of the encapsulating phase are comparable, the energy barriers are high enough to trap elongated structures or even linear chains, independently of N. However, when the surface tension of the encapsulating phase is much larger than that of the droplets, non-spherical morphologies are stable only at sufficiently high N. In such a case multiple internal interfaces can hold stresses and prevent relaxation of the global deformations which leads to a plastic, gel-like behavior. Our findings can serve as guidelines for synthesis of functional particles as well as for designing biomimetic materials, e.g. for tissue engineering.

1J.G. acknowledges financial support from Polish Ministry of Science provided within the framework Mobility Plus.

5:03PM E35.00002 Yield-stress fluid drop impact on heated surfaces. BRENDA BLACKWELL, ALEX WU, RANDY EWOLDT, Univ of Illinois - Urbana — Yield-stress fluids, including gels and pastes, are effectively fluid at high stress and solid at low stress. In liquid-solid impacts, these fluids can stick and accumulate where they impact, motivating several applications of these rheologically-complex materials. Here we use high-speed imaging to experimentally study liquid-solid impact of yield-stress fluids on heated surfaces. At low temperatures yield-stress fluids tend to stick to surfaces and leave a coating layer. At sufficiently high temperatures the Leidenfrost effect can be observed, wherein a layer of vapor is created between the material and the surface due to rapid boiling, which can prevent a droplet of yield-stress fluid from sticking to the surface. In this study rheological material properties, drop size, drop velocity, and temperature surface are varied to characterize behavioral regimes. Material sticking to and releasing from the surface is observed as a function of the input parameters.

5:16PM E35.00003 A Computational Study of the Rheology and Structure of Surfactant Covered Droplets. JOAO MAIA, ARMAN BOROMAND, SAFA JAMALI, Case Western Reserve University — The use of different types of surface-active agents is ubiquitous practice in different industrial applications ranging from cosmetic and food industries to polymeric nano-composite and blends. This allows stable multiphasic systems like foams and emulsions to be produced. Stability and shelf-life of those products are directly determined by the efficiency of the surfactant molecules. Although the effect of molecular configuration of the surface-active molecules on the planar interfaces has been studied both experimentally and computationally, it remains challenging to track the efficiency and effectiveness of different surfactant molecules on curved interfaces. In this study we address this gap by using Dissipative Particle Dynamics, to study the effectiveness and efficiency of different surfactant molecules (linear vs. branched) on a curved interface in equilibrium and far from equilibrium. In particular, we are interested to relate interfacial properties of the surface covered droplets and its dynamics to the molecular configuration of the surface active molecules under equilibrium and far from equilibrium condition.
5:29PM E35.00004 Simulation of Droplet Generation in a Non-Newtonian Dense Granular Suspension1. GUSTAF MRTENSSON, Chalmers University of Technology, Mycron AB, MARTIN SVENSSON, ANDREAS MARK, FREDRIK EDELVIK, Fraunhofer-Chalmers Research Centre for Industrial Mathematics — As with the jet printing of dyes and other low-viscosity fluids, the jetting of dense fluid suspensions is dependent on the repeatable break-off of the fluid filament into well-formed droplets. It is well known that the break-off of dense suspensions is dependent on the volume fraction of the solid phase, particle size and morphology, fluid phase viscosity et cetera, see for example van Deen et al. (2013). The purpose of this study is to propose a novel simulation framework and to show that it captures the main effects such as droplet shape, volume and speed in a cylindrical duct test configuration. The granular suspension is modelled as a mixed single phase suspension, where the local thermodynamic properties are determined by the mixture level.

The simulations are performed with IBOFLOW, a multiphase flow solver, coupled with LaStFEM, a large strain FEM solver. To study how the droplet generation is affected by the acceleration of the fluid, simulations are performed for a series of actuation profiles. The simulation results were compared to experimental data obtained from an industrial jetting head. The simulations exhibit qualitative agreement with the experimental data. A sensitivity to the inlet boundary condition with respect to the resulting droplet speed was observed.

1Thanks to Swedish Research Council (Grant 2010-4334)

5:42PM E35.00005 Multiple phenomena triggered by surfactant solutions on liquid pools1. XIANG WANG, Max Planck Institute for Polymer Research — When a drop of aqueous surfactant solution is placed on a deep subphase of water, multiple phenomena occur. The contact line of the drop spreads until the drop merges with the subphase. A capillary wave train is initiated by the disturbance caused by the drop touching the subphase surface. Marangoni stresses cause the formation and propagation of a localized distortion of the subphase surface (subsequently called the Marangoni ridge). And particles pre-deposited on the subphase surface are propelled by the flow induced by Marangoni stresses. We examine all these phenomena simultaneously at early times. The drop contact line is initiated by the disturbance caused by the drop touching the subphase surface. Marangoni stresses cause the formation and propagation of a localized distortion of the subphase surface (subsequently called the Marangoni ridge). And particles pre-deposited on the subphase surface are propelled by the flow induced by Marangoni stresses. We examine all these phenomena simultaneously at early times. The drop contact line is initiated by the disturbance caused by the drop touching the subphase surface.

The Marangoni ridge propagates slower than the slowest crest of the capillary wave train and the capillary waves are not affected by the presence of the surfactant. Particle motion is not induced by the capillary waves but is initiated by the passing Marangoni ridge. The particles are rapidly accelerated by the force from surface tension gradient acting on the contact line of the particles and viscous forces acting on the submerged surface of the particles.

1This research was supported by the German Research Foundation (DFG) within the Cluster of Excellence 259 Smart InterfacesUnderstanding and Designing Fluid Boundaries and by the National Science Foundation CBET-1159369.

5:55PM E35.00006 Coalescence avalanches in 2D emulsions: a stochastic approach, DANNY RAJ MASILA, RAGHUNATHAN RENGASWAMY, Indian Institute of Technology Madras — One coalescence event in a 2D concentrated emulsion can trigger an avalanche resulting in the rapid destabilization of the drop assembly. The sensitive dependence of this phenomenon on various factors that include surfactant concentration and viscosities of the fluid phases makes the avalanching problem appear probabilistic. We propose a stochastic framework that utilizes a probability function to explain local coalescence events to study the dynamics of the coalescence avalanches. A function that accounts for the local coalescence mechanism is used to fit the experimentally (from literature) measured probability data. A continuation parameter is introduced along with this function to account for the effect of system properties on the avalanche dynamics. Our analysis reveals that this behavior is a result of the inherent autocatalytic nature of the process. We discover that the avalanche dynamics shows critical behavior where two outcomes are favored: no avalanche and large avalanches that lead to destabilization. We study the effect of system size and fluid properties on the avalanche dynamics. A sharp transition from non-autocatalytic (stable emulsions) to autocatalytic (unstable behavior) is observed as parameters are varied.

Sunday, November 22, 2015 4:50PM - 6:08PM

Session E36 Drops: Condensation and Freezing Ballroom C - Alexandre Ponomarenko, Harvard University

4:50PM E36.00001 Instant freezing of impacting wax drops, ALEXANDRE PONOMARENKO, EMMANUEL VIROT, SHMUEL RUBINSTEIN, Harvard University — We present the impact of hot liquid drops of wax on surfaces whose temperature is below the solidifying temperature of the drops. During the fall the drops remain mostly liquid, but upon impact, their temperature quickly decreases resulting in the solidification of the drop. Depending on the impact energy, drop size and the temperature difference between the drop and the surface the solidification results in plethora of solid shapes: simple lenses, triangular drops, spherical caps and popped popcorn shapes.

5:03PM E36.00002 Spatial Control of Condensation using Chemical Micropatterns, KEVIN MURPHY, Virginia Tech, RYAN HANSEN, Oak Ridge National Laboratory, SAURABH NATH, Virginia Tech, SCOTT RETTERER, PATRICK COLLIER, Oak Ridge National Laboratory, JONATHAN BOREYKO, Virginia Tech, NATURE-INSPIRED FLUIDS AND INTERFACES TEAM, CENTER FOR NANOPHASE MATERIALS SCIENCES TEAM — Surfaces exhibiting wettability patterns can spatially control the nucleation of condensation to enable enhanced fog harvesting and phase-change heat transfer. To date, studies of patterned condensation have utilized a combination of chemical and topographical features, making it difficult to isolate the effects of intrinsic wettability versus surface roughness on spatially controlling the condensation phenomenon. Here, we fabricate chemically micropatterned consisting of hydrophilic silicon oxide and a smooth hydrophilic silane monolayer to isolate the effects of changes in intrinsic wettability on the spatial control of condensation. Complete spatial control, defined as every nucleation and growth event occurring exclusively on the hydrophilic features, was observed even for supercooled droplets at high water vapor supersaturation. However, this complete spatial control was found to break down beyond a critical spacing that depended upon the extent of supersaturation. The average diameter of condensate was found to be smaller for the chemically micropatterned surfaces compared to a uniformly hydrophobic surface. Control of inter-droplet spacing between supercooled condensate through chemical patterning can be employed to minimize the growth of inter-droplet frost on cold surfaces.

5:16PM E36.00003 Can Ice Prevent Frost Growth? SAURABH NATH, Virginia Tech, RYAN HANSEN, Oak Ridge National Laboratory, KEVIN R. MURPHY, Virginia Tech, SCOTT RETTERER, PATRICK COLLIER, Oak Ridge National Laboratory, JONATHAN BOREYKO, Virginia Tech, NATURE-INSPIRED FLUIDS & INTERFACES TEAM, CENTER FOR NANOPHASE MATERIALS SCIENCES TEAM — So-called icedrophobic surfaces that exhibit special wettability characteristics can delay the onset of ice nucleation in supercooled water. However, to date no icedrophobic surface has been able to passively prevent frost growth once ice nucleates. Here, we demonstrate that the growth rate of frost can be tuned and even halted with a chemically patterned surface that controls the spatial distribution of supercooled condensation. The success and speed of inter-droplet frost growth is found to depend upon two primary factors: the extent of spacing between hydrophilic regions where liquid nucleation occurs and the time allowed for condensation growth prior to the initial freezing event. Instead of delaying the onset of freezing, we initiate freezing as early as possible. This creates a “dry zone” where no frost and condensation can occur. The underlying mechanism behind the “dry zone” involves the saturation vapor pressure over ice that is lower than that over water at the same temperature, causing ice to behave like a passive humidity sink. Thus, quite remarkably it appears that ice itself may be the solution to the frosting problem.
5:29PM E36.00004 Freezing Behavior of a Supercooled Water Droplet Impacting on Surface Using Dual-Luminescent Imaging Technique, MIO TANAKA, Tokyo University of Science, KATSUAKI MORITA, Japan Aerospace Exploration Agency, MAKOTO YAMAMOTO, Tokyo University of Science, HIROTAKA SAKAIE, University of Notre Dame — A collision of a supercooled-water droplet on an object creates ice accretion on its surface. These icing problems can be seen in any cold environments and may lead to severe damages on aircrafts, ships, power cables, trees, road signs, and architectures. To solve these problems, various studies on ice-prevention and ice-prediction techniques have been conducted. It is very important to know the detail freezing mechanism of supercooled water droplets to propose or improve those techniques. The icing mechanism of a single supercooled-water droplet impacting on object surface would give us great insights for constructing those techniques. In the present study, we use a dual-luminescent imaging technique to measure the time-resolved temperatures of a supercooled water droplet impacting with different speed. The technique we applied consists of high-speed color camera and two luminescent probes. We will report the current status of this experiment in the presentation.

5:42PM E36.00005 Dry Zones Around Frozen Droplets, CAITLIN BISBANO, SAURABH NATH, JONATHAN BOREYKO, Virginia Tech, NATURE-INSPIRED FLUIDS AND INTERFACES TEAM — The saturation pressure of water vapor above supercooled water exceeds that above ice at the same temperature. A frozen droplet will therefore grow by harvesting water vapor from neighboring supercooled condensate, which has recently been demonstrated to be a primary mechanism of in-plane frost growth on hydrophobic surfaces. The underlying physics of this source-sink interaction is still poorly understood. In this work, a deposited water droplet is frozen on a dry hydrophobic surface initially held above the dew point. We demonstrate that when the surface is then cooled beneath the dew point, the frozen droplet harvests nearby water vapor in the air. This results in an annular dry zone that forms between the frozen droplet and the forming supercooled condensation. For a given ambient temperature and humidity, the length of the dry zone varied strongly with surface temperature and weakly with droplet volume. The dependence of the dry zone on surface temperature is due to the fact that the vapor pressure gradients between the ambient and the surface and between the liquid and frozen water are both functions of temperature.

5:55PM E36.00006 Ice Formation Delay on Penguin Feathers, ELAHEH ALIZADEHBIRJANDI, FARYAR TAVAKOLI-DASTJERDI, Department of Mechanical and Aerospace Engineering, UCLA, JUDY ST. Leger, SeaWorld Parks and Entertainment, STEPHEN H. DAVIS, Department of Engineering Sciences and Applied Mathematics Northwestern University, JONATHAN P. ROTHSTEIN, Department of Mechanical and Industrial Engineering University of Massachusetts, Amherst, H. PIROUZ KAVEHPOUR, Department of Mechanical and Aerospace Engineering UCLA — Antarctic penguins reside in a harsh environment where air temperature may reach -40 °C with wind speed of 40 m/s and water temperature remains around -2.2 °C. Penguins are constantly in and out of the water and splashed by waves, yet even in sub-freezing conditions, the formation of macroscopic ice is not observed on their feathers. Bird feathers are naturally hydrophobic; however, penguins have an additional hydrophobic coating on their feathers to reinforce their non-wetting properties. This coating consists of preen oil which is applied to the feathers from the gland near the base of the tail. The combination of the feather’s hydrophobicity and surface texture is known to increase the contact angle of water drops on penguin feathers to over 140 ° and classify them as superhydrophobic. We here develop an in-depth analysis of ice formation mechanism on superhydrophobic surfaces through careful experimentations and development of a theory to address how ice formation is delayed on these surfaces. Furthermore, we investigate the anti-icing properties of warm and cold weather penguins with and without preen oil to further design a surface minimizing the frost formation which is of practical interest especially in aircraft industry.
5:03PM E37.00002 Computational algorithms for vesicle electrohydrodynamics\textsuperscript{1}. SHRAN VEERAPANENI, University of Michigan — In this talk, we discuss a new integral equation method for simulating the electrohydrodynamics of a suspension of vesicles. The classical Taylor-Melcher leaky-dielectric model is employed for the electric response of each vesicle and the Helfrich energy model combined with local inextensibility is employed for its elastic response. The coupled governing equations for the vesicle position and its transmembrane electric potential are solved using a numerical method that is spectrally accurate in space and first-order in time. The method uses a semi-implicit time-stepping scheme to overcome the numerical stiffness associated with the governing equations. We will present new results on the suspension rheology, two-body interactions and pattern formation. This is joint work with Bowei Wu.

\textsuperscript{1}This work was sponsored by NSF under grants DMS-1224656 and DMS-1418964

Sunday, November 22, 2015 4:50PM - 6:08PM —
Session E38 Flow Instability: General I Sheraton Back Bay B - M.J. Philipp Hack, Stanford University

5:03PM E38.00002 Effect of Prandtl number on the linear stability of compressible Couette flow . KRISHNENDU SINHA, ASHWIN RAMACHANDRAN, BIJAYLAKSHMI SAIKIA, Department of Aerospace Engineering, IIT Bombay, RAMA GOVINDARAJAN, TIFR Centre for Interdisciplinary Sciences, Hyderabad — Accurate prediction of laminar to turbulent transition in high speed flows is a challenging task. Compressibility, and the resultant large variations in transport properties can affect this transition significantly. Prandtl number (ratio of momentum and thermal diffusivities) is an important parameter which affects the linear stability of high Mach number wall-bounded flows. A two-dimensional compressible plane Couette flow having uniform viscosity and thermal conductivity with varying Prandtl numbers is our model problem. A temporal stability analysis shows that the variation of phase speed with Prandtl number leads to synchronization between acoustic modes, with peaks in growth rate at the synchronization points. Two types of branching patterns are observed, depending on the Prandtl number. The stability diagrams for varying Mach and Reynolds numbers show a destabilizing role of decreasing Prandtl number, both in terms of increased disturbance growth rates, and of larger regions of instability in the parameter space. It also results in a significant reduction in the critical Reynolds number of the flow, especially at high Mach numbers.

5:16PM E38.00003 Criteria for instability of helical disturbances in inviscid, swirling flows . CHRISTOPHER DOUGLAS, BENJAMIN EMERSON, TIMOTHY LIEUWEN, Georgia Institute of Technology — This work considers the linear inviscid instability of columnar vortices with axial flow in unbounded domains subjected to 3D perturbations. The base flow parameters have a general dependence on the radial distance from the swirl axis. Following Howard and Gupta’s approach, we develop two stability conditions in terms of an infinite set of helical disturbances via a normal modes expansion. We develop a generalization of Fjørtoft’s necessary criterion which states that a wave-like disturbance may be unstable if the base shear velocity has an inflection point in the binormal direction of the helix which is also a vorticity maximum. A necessary condition for instability is that

\[ (W' - W_0')d(\kappa')/dr < 0 \]

must be satisfied somewhere for any real constant \( W_0' \) where \( \kappa \) is the curvature of the helix, \( W' \) is the binormal base velocity, and \( \gamma' \) is the binormal base shear rate. The second condition leads to a generalization of Rayleigh’s criterion for centrifugal instability for helical disturbances. We find that a necessary and sufficient condition for instability is that

\[ Vd\gamma'/dr < 0 \]

be satisfied somewhere, where \( V \) is the base azimuthal velocity and \( \Gamma' \) is the base circulation due to the flux of vorticity tangent to the helical vortex tube.

Sunday, November 22, 2015 4:50PM - 6:08PM —
Session E37 Focus Session: Electro-Hydro-Dynamics of Drops, Vesicles and Membranes II Sheraton Back Bay A - David Saintillan, University of California - San Diego

5:16PM E37.00003 Electrohydrodynamics Of Multicomponent Vesicles\textsuperscript{1}. PRERNA GERA, DAVID SALAC, University at Buffalo SUNY — The addition of cholesterol into a lipid membrane induces the formation of distinct domains. These domains try to minimize the overall energy of the system by coalescence and migration. The application of electric fields will induce flow of these membrane domains and influence the rate at which they coarsen. In this work the electrohydrodynamics of multicomponent vesicles is numerically modelled. The method uses a Cahn-Hilliard-Cook model of the lipid domains restricted to a deforming three-dimensional vesicle and will be briefly discussed. Sample results will be presented and compared to experimental observations.

\textsuperscript{1}This work supported by NSF Grant #1253739

5:29PM E37.00004 Magnetohydrodynamics of Vesicles . DAVID SALAC, University at Buffalo SUNY — Lipid molecules are known to have an anisotropic magnetic susceptibility. When a lipid vesicle is exposed to a magnetic field, this anisotropy induces forces which drag the vesicle and the surrounding fluid into motion. Here a new model of a three-dimensional vesicle in the presence of magnetic fields is presented. The model is based on a novel level-set/projection method which enforces volume and surface area conservation simultaneously. The force on the vesicle membrane due to the applied magnetic field will be shown. The simulated dynamics will be compared to experimental results and future possibilities of combining electric and magnetic fields will be discussed.

Sunday, November 22, 2015 4:50PM - 6:08PM —
Session E38 Flow Instability: General I Sheraton Back Bay B - M.J. Philipp Hack, Stanford University
Three-dimensional instabilities in a rapidly counter-rotating split cylinder.

The three-dimensional flow in a counter-rotating cylinder that is split at its mid-plane is studied numerically via spectral methods. The cylinder of radius $a$ and length $h$ is completely filled with fluid of kinematic viscosity $\nu$. The top half rotates with angular speed $\omega$ and the bottom half with angular speed $-\omega$. There are two nondimensional parameters governing the flow, $Re = \omega a^2/\nu$ and $\Gamma = h/a$. For small values of $Re$ and $\Gamma$ the flow is steady, axisymmetric and reflection symmetric about the mid-height (with appropriate changes of sign for some flow components). In this regime the interior flow in each half of the cylinder rotate as solid-body rotation of opposite senses. Apart from the boundary layers on the cylinder walls, there is also an internal shear layer separating the two counter-rotating halves. Above a critical $Re$ that depends on $\Gamma$, this internal shear layer becomes unstable to low frequency instabilities that break both the axisymmetry and the reflection symmetry. For these cases there exist rotating waves associated with the shear-layer instability. The variation of the critical $Re$ and the azimuthal wavenumbers of the instability as a function of $\Gamma$ is studied, along with the nonlinear dynamics.

This work is supported by NSF (CBET-1150389)

**Sunday, November 22, 2015 4:50PM - 6:08PM**

**Session E37 Focus Session: Electro-Hydro-Dynamics of Drops, Vesicles and Membranes II** Sheraton Back Bay A - David Saintillan, University of California - San Diego

**5:42PM E38.00004 Inertial instability of miscible fluid stratifications in square microchannels.** XIAOYI HU, THOMAS CUBAUD, Stony Brook University — The stability of stratifications made between miscible fluids having large differences in viscosity is experimentally investigated in square microchannels. Parallel fluid layers with a fast central stream and a slow sheath flow are produced by focusing low-viscosity fluid with high-viscosity fluid in a straight microchannel. We examine in particular the formation and evolution of periodic wave trains at each fluid interface over a range of fluid viscosities and flow rates. This study shows that miscible fluid arrangements can be destabilized for moderate Reynolds numbers. Several relationships are developed for the propagating velocity, size, and frequency of generated waves. In the unstable regime, minute amount of high-viscosity fluid is entrained and blended into the low-viscosity fluid recirculating plumes formed by the traveling waves. This phenomenon provides new insights into the development of microfluidic methods for continuously mixing high- and low-viscosity fluids.

The three-dimensional flow in a counter-rotating split cylinder is studied, along with the nonlinear dynamics.

**Sunday, November 22, 2015 5:55PM - 6:08PM**

**Session E39 Flow Instability: Richtmyer-Meshkov II** Sheraton Back Bay C - Peter Vorobieff, University of New Mexico

**5:55PM E38.00006 Optimal Free-Stream Vortical Disturbances.** M.J. PHILIPP HACK, Center for Turbulence Research, Stanford University — In boundary layers exposed to moderate levels of free-stream disturbances, natural transition via the exponential amplification of Tollmien-Schlichting waves is bypassed by a more rapid breakdown process. The external disturbances interact with the mean shear and induce the growth of highly energetic streaks, which cause transition to turbulence by virtue of the growth of inviscid secondary instabilities. The relationship between external vortices and boundary-layer perturbations is, however, not entirely understood. The present study provides a rigorous link between dynamics in the free-stream and inside the boundary layer by computing the optimal free-stream vortical disturbances, i.e. the external disturbances which maximize the energy content of the resulting boundary-layer perturbations. The mathematical framework is based on a semi-norm formulation of the adjoint linearized compressible Navier-Stokes equations in curvilinear coordinates and enables the global analysis of disturbance sensitivity as well as the computation of optimal disturbances in flows with variable density and miscellaneous geometries.
4:50PM E39.00001 Flow morphologies after oblique shock acceleration of a cylindrical density interface\textsuperscript{1}. PATRICK WAYNE, DYLAN SIMONS, DELL OLMSTEAD, C. RANDALL TRUMAN, PETER VOROBIEFF, University of New Mexico, SANJAY KUMAR, IIT Kanpur — We present an experimental study of instabilities developing after an oblique shock interaction with a heavy gas column. The heavy gas in our experiments is sulfur hexafluoride infused with 11% acetone by mass. A misalignment of the pressure and density gradients results in three-dimensional vorticity deposition on the gaseous interface, triggering the onset of Richtmyer-Meshkov instability (RMI). Shortly thereafter, other instabilities develop along the interface, including a shear-driven instability that presents itself on the leading (with respect to the shock) and trailing edges of the column. This leads to the development of rows of co-rotating “cat’s eye” vortices, characteristic of Kelvin-Helmholtz instability (KHI). Characteristics of the KHI, such as growth rate and wavelength, depend on several factors including the Mach number of the shock, the shock tube angle of inclination $\alpha$ (equal to the angle between the axis of the column and the plane of the shock), and the Atwood number.

\textsuperscript{1}This work is supported by the US National Nuclear Security Agency (NNSA) via grant DE-NA0002913.

5:03PM E39.00002 Structure functions of passive scalar: evolution in fully 3D shock-driven transition to turbulence\textsuperscript{1}. PETER VOROBIEFF, DELL OLMSTEAD, DYLAN SIMONS, PATRICK WAYNE, C. RANDALL TRUMAN, University of New Mexico, SANJAY KUMAR, IIT Kanpur — Oblique interaction between a planar shock and a cylindrical density interface results in baroclinic vorticity deposition. The character of the evolving flow is thus different from a similar flow produced by planar normal shock acceleration of the same density interface. In the latter case (planar normal shock), vorticity deposited by the shock is predominantly two-dimensional (directed along the axis of the cylinder), while in the case we consider the shock-induced vorticity field is fully three-dimensional. This results in a complex interplay of vortical structures with different orientations. The statistical properties of the flow are analyzed based on images from two orthogonal visualization planes, using second-order structure functions of the intensity maps of fluorescent tracer pre-mixed with the heavy gas. Scalings consistent with fully developed turbulence are observed at late times. The character of the emergence of these scalings is affected by the flow Mach number, Atwood number, and initial geometry.

\textsuperscript{1}This work is supported by the US National Nuclear Security Agency (NNSA) via grant DE-NA0002913.

5:16PM E39.00003 Progress on Simultaneous PLIF/PIV Measurements for a Turbulent Complex Fluid Interface \textsuperscript{1}. DAVID REILLY, MOHAMMAD MOHAGHAR, JOHN CARTER, Georgia Institute of Technology, JACOB MCFARLAND, University of Missouri, DEVESH RANJAN, Georgia Institute of Technology — Experiments were performed at the inclined shock tube facility at Georgia Institute of Technology to study a Richtmyer-Meshkov unstable complex interface. The complex density stratification was achieved by counter flowing $N_2$ and CO$_2$ in order to create shear and buoyancy effects. The resulting Atwood number is 0.23 with an incident shock strength of Mach 1.55 and an angle of inclination of 80$^\circ$. High-resolution, full-field simultaneous Planar Laser-Induced Fluorescence (PLIF) and Particle Image Velocimetry (PIV) was employed to measure density and velocity statistics, respectively. For the first time with the inclined interface, mixing parameters from the BHR (Besnard-Harlow-Rauenzahn) model, including the density self-correlation and turbulent mass flux, are determined from experiments. Secondary modes added to the interface result in markedly greater mixing compared to the simple inclined interface as measured by mixedness and mixed mass.

5:29PM E39.00004 Evaluation of a Two-Length Scale Turbulence Model with Experiments on Shock-Driven Turbulent Mixing \textsuperscript{1}. JOHN CARTER, Georgia Inst of Tech, ROB GORE, Los Alamos National Laboratory, DEVESH RANJAN, Georgia Inst of Tech — A new second moment turbulence model which uses separate transport and decay length scales is used to model the shock-driven instability. The ability of the model to capture the evolution of turbulence statistics and mixing is discussed. Evaluation is based on comparison to the Georgia Tech shock tube experiments. In the experiments a membraneless light-over-heavy interface is created. There is a long-wavelength perturbation which exists due to inclination of the entire shock tube. By limiting calculations to one dimension, there is not a geometric description of the incline, and the ability of the transport length scale alone to capture the effect of the long-wavelength perturbation is tested.

5:42PM E39.00005 Shock Wave Interactions in Multi-Phase Particle Systems Characterized by Various Interfaces \textsuperscript{1}. WOLFGANG BLACK, Univ of Missouri - Columbia, NICHOLAS DENISSEN, Los Alamos National Laboratory, JACOB MCFARLAND, Univ of Missouri - Columbia — Multi-phase systems have been of interest since the 1800s with Stokes studying flow over a particle, and are still an important field of study today with various applications in propulsion design, astrophysics, refrigeration, fluid instabilities, as well as fusion. Many multi-phase systems experience complex accelerations, such as shock waves, which may drive shear dominated instabilities, increase or dampen mixing between the phases, and even affect a phase change phenomena within the flow. The parameter space to study within these systems is extensive and provides a rich field for research, with hydrodynamic codes allowing new insight into old and recent experiments alike. This talk will discuss early efforts to tap into this parameter space by using high density high energy hydrodynamics codes to investigate simulations of multi-phase systems that experience a shock wave interaction across an interface, for example a particle laden gas cylinder within an unseeded shocked tube environment, and the evolution of these systems. This particular interface will be compared with recent experiments within literature while other turbulent interfaces will be discussed as future experiments to be performed by the University of Missouri Fluid Mixing Shock Tube Laboratory.

5:55PM E39.00006 Comparison of hydrodynamic simulations with two-shockwave drive target experiments\textsuperscript{1}. VARAD KARKHANIS, PRAVEEN RAMAPRABHU, University of North Carolina at Charlotte, WILLIAM BUTTLER, Los Alamos National Laboratory, USA — We consider hydrodynamic continuum simulations to mimic ejecta generation in two-shockwave target experiments [1], where metallic surface is loaded by two successive shock waves. Time of second shock in simulations is determined to match experimental amplitudes at the arrival of the second shock. The negative Atwood number ($A \rightarrow -1$) of ejecta simulations leads to two successive phase inversions of the interface corresponding to the passage of the shocks from heavy to light media in each instance. Metallic phase of ejecta (solid/liquid) depends on shock loading pressure in the experiment, and we find that hydrodynamic simulations quantify the liquid phase ejecta physics with a fair degree of accuracy, where RM instability is not suppressed by the strength effect. In particular, we find that our results of free surface velocity, maximum ejecta velocity, and maximum ejecta areal density are in excellent agreement with their experimental counterparts, as well as ejecta models [2,3]. We also comment on the parametric space for hydrodynamic simulations in which they can be used to compare with the target experiments. [1] W. T. Buttler et al., J. Appl. Phys., 116 (2014). [2] Guy Dimonte et al., J. Appl. Phys., 113 (2013). [3] W. T. Buttler et al., J. Fluid Mech., 703 (2012).

\textsuperscript{1}This work was supported in part by the (U.S.) Department of Energy (DOE) under Contract No. DE-AC52-06NA2-5396.
Towards an understanding of vortex shedding frequency in quasi-two-dimensional flows.

Paul W. Fontana

Seattle University

I investigate mean flows and the role played by surface friction and surface tension in generating them in a quasi-two-dimensional vortex shedding experiment, thereby elucidating the connection between quasi-two-dimensional effects and shedding frequency. We have previously shown that quasi-two-dimensional effects in a vertical soap film channel produce anomalously low frequencies compared with conventional observations, and that the Strouhal number \( St = fD/U_{∞} \), where \( f \) is the shedding frequency, \( D \) the cylinder diameter, \( U_{∞} \) the upstream flow speed, is not uniquely determined by the Reynolds number \( Re = DU_{∞}/ν \), where \( ν \) is the kinematic viscosity. Surface tension and surface friction play dominant roles in the experiment. We observe that the flow breaks its symmetry with a single wavenumber \( \pi A/c \), where \( A \) is the amplitude of oscillation, \( f \) the frequency of oscillation, \( c \) the diameter of the disk, and \( ν \) the kinematic viscosity of the fluid. We observe two distinctive flow regions in the \((KC, β)\) parameter space. First, in the low \( β \) region, the flow breaks its symmetry with a single wavenumber mode getting a positive growth rate. Second, in the high \( β \) region, high-order unstable modes emerge, with the highest mode number \( m = 9 \) recorded. Furthermore, we carry out Direct Numerical Simulations (DNS) on the fully three-dimensional Navier-Stokes equations. The results reproduce the main features of the high-order unstable modes predicted by the Floquet analysis, exhibiting the highest mode number \( m = 6 \). We conjecture that the inconsistency in the highest mode number between the Floquet linear stability analysis and the DNS implies the non-linear characteristic of the current problem.

Supported by the National Natural Science Foundation of China (Grant No: 11272283) and Zhejiang Provincial Natural Science Foundation of China (Grant No: LY12A020006)
8:00AM G1.00001 Reactive mixing in heterogeneous porous media flows: scalar gradient distribution, spatial intermittency and temporal scaling of effective reaction kinetics, TANGUY LE BORGNE, Université de Rennes 1, MARCO DENTZ, IDAE-CSIC Barcelona, EMMANUEL VILLERMAUX, Université Aix-Marseille — Reactive mixing processes play a central role in a range of porous media systems, including CO2 sequestration operations, reactive geothermal dipoles, biofilms, or flow-through reactors. Many of these reactions are limited by fluid mixing processes that bring the reactants into contact. Hence, the temporal dynamics of effective global reactivity is determined by the creation of concentration gradients by fluid stretching and their dissipation by diffusion. From the analysis of the elongation and aggregation of lamellar structures formed in the transported scalar fields, we derive analytical predictions for the probability density functions of scalar gradients in heterogeneous Darcy flows over a large range of Péclet numbers and permeability field variances. In this framework, we show that heterogeneous Darcy fields generate highly intermittent concentration fields, as manifested by the spatial scaling of structure functions. The resulting effective reaction rates display a range of temporal behaviors that depend on the degree of heterogeneity. In the large Damköhler limit, we derive analytical expressions for these temporal scalings in the different regimes that arise when exploring the Péclet number space. We generalize these results for different random flows.

8:13AM G1.00002 Predicting anomalous diffusion rates of Stokes flow in porous media, BRYAN QUAIFE, Florida State University, PIETRO DE ANNA, MIT, GEORGE BIROS, University of Texas, RUBEN JUANES, MIT — Stokes flow in porous media finds many applications in hydrology, filtration, and groundwater flow. I will first describe an experimental setup that simulates two-dimensional Stokes flow in a porous media, and compare experimental and numerical results. Then, I will describe a technique where statistics of the geometry are used to predict statistics of the flow, including anomalous diffusion rates. This technique will be tested on several geometries.

8:26AM G1.00003 Simulating Anomalous Dispersion and Multiphase Segregation in Porous Media with the Lattice Boltzmann Method, RASTIN MATIN, MAREK K. MISZTAL, ANIER HERNANDEZ-GARCIA, JOACHIM MATHIESEN, Niels Bohr Institute — Many hydrodynamic phenomena such as flows at micron scale in porous media, large Reynolds numbers flows, non-Newtonian and multiphase flows have been simulated numerically using the lattice Boltzmann method. By solving the Lattice Boltzmann Equation on three-dimensional unstructured meshes, we efficiently model single-phase fluid flow in real rock samples. We use the flow field to estimate the permeability and further investigate the anomalous dispersion of passive tracers in porous media. By extending our single-phase model with a free-energy based method, we are able to simulate binary systems with moderate density ratios in a thermodynamically consistent way. In this presentation we will present our recent results on both anomalous transport and multiphase segregation.

8:39AM G1.00004 Dispersion properties in porous media: application to Redox Flow Battery electrodes, FRANCESCO PICANO, DARIO MAGGIOLIO, ANDREA MARION, MASSIMO GUARNIERI, Department of Industrial Engineering, University of Padova — Redox Flow Batteries (RFBs) represent a promising technology as a way to store energy. However, in order to improve RFBs performance, some conceptual and technological issues are still open. In particular, a properly designed geometry of flow channels and porous medium is still under investigation in order to uniformly distribute the reacting species all along the electrode. The ideal configuration aims to minimize the drag maximizing the mixing so to increase the overall performance and efficiency. In the present work a Lattice Boltzmann 3D model (LBM) has been used to better understand the dependence of mass and momentum transports on the porosity and carbon fiber preferential orientation. The LBM has been coupled with a Lagrangian particle tracking algorithm in order to investigate the dispersion mechanism on the scale flowing in a typical RFB. Results show that the drag is considerably reduced when the medium fibers are preferentially oriented along the streamwise direction. Surprisingly, this configuration shows also the highest transversal dispersion rate characterized by a super-diffusive behavior. Actually, the dispersion features are found to strongly depend on the porous media microstructure showing either anomalous or regular diffusion.

8:52AM G1.00005 Diffusion in random networks, JUAN C. PADRINO, DUAN Z. ZHANG, Los Alamos National Laboratory — The ensemble phase averaging technique is applied to model mass transport in a porous medium. The porous material is idealized as an ensemble of random networks, where each network consists of a set of junction points representing the pores and tortuous channels connecting them. Inside a channel, fluid transport is assumed to be governed by the one-dimensional diffusion equation. Mass balance leads to an integro-differential equation for the pores mass density. Instead of attempting to solve this equation, and equivalent set of partial differential equations is derived whose solution is sought numerically. As a test problem, we consider the one-dimensional diffusion of a substance from one end to the other in a bounded domain. For a statistically homogeneous and isotropic material, results show that for relatively large times the pore mass density evolution from the new theory is significantly delayed in comparison with the solution from the classical diffusion equation. In the short-time case, when the solution evolves with time as if the domain were semi-infinite, numerical results indicate that the pore mass density becomes a function of the similarity variable $x t^{-1/2}$ rather than $x t^{-1/2}$ characteristic of classical diffusion. This result was verified analytically. Possible applications of this framework include flow in gas shales.

9:05AM G1.00006 Coupling micro-CT with computer simulations to analyze dispersion in porous media, SADAF SOBHANI, JARED DUNNMON, MICHAEL WERER, Stanford University — In recent years, table-top X-ray Computed Tomography (XCT) systems have been utilized to analyze various samples with a resolution on the order of 1μm-100μm. In this study, we explore the use of these systems both in extracting high-resolution topologies of porous structures for use as inputs in computational simulations and in directly characterizing gas dispersion within such structures using fluorescent imaging of dense gaseous tracers. The opaque-solid environment and small pore-scale effects in porous media restrict the use of conventional imaging techniques, thereby making XCT a potentially useful diagnostic technique for understanding internal flows in porous and optically inaccessible structures. In the present work, we extract the topology of various reticulated porous foams from 3D XCT data and perform numerical simulations of the flow in a range of Péclet numbers and permeability field variances. Permeability and tortuosity, which are key parameters in volume-averaged models are evaluated from the resulting flow fields and knowledge of the solid structure.
9:18AM G1.00007 Particle dispersion and deposition in porous media: a computational perspective. GIANLUCA BOCCARDO, ELEONORA CREVACORE, RAJANDREA SETHI, DANIELE MARCHISIO, Politecnico di Torino — This work investigates particle dispersion in porous media, which is of central relevance in a number of applications ranging from groundwater remediation to chemical engineering. The challenge lies in studying the complex fluid dynamics behavior arising at the microscale (very difficult to observe experimentally) and obtaining transport models to be employed at the macroscopic scale of interest. While a wealth of studies have approached this problem, the case of particle transport with a concurrent heterogeneous chemical reaction (e.g.: particle deposition) still lacks a satisfactory description, especially when considering a polydisperse population of solid particles. Moreover, the oft-used simplified descriptions of the porous medium (via array of spheres or similar strategies) fail to fully take into account the effect of the packing structure. Our novel approach relies on an “in-silico” procedure where many 3-D realistic porous media models are constructed via rigid-body simulations and fluid flow. We then investigate transport and deposition. Our approach thus affords a deeper look, resulted by these methodology, into the influence of the features of realistic porous media on particle transport and packing structure. This work investigates particle dispersion in porous media, which is of central relevance in a number of applications ranging from groundwater remediation to chemical engineering. The challenge lies in studying the complex fluid dynamics behavior arising at the microscale (very difficult to observe experimentally) and obtaining transport models to be employed at the macroscopic scale of interest. While a wealth of studies have approached this problem, the case of particle transport with a concurrent heterogeneous chemical reaction (e.g.: particle deposition) still lacks a satisfactory description, especially when considering a polydisperse population of solid particles. Moreover, the oft-used simplified descriptions of the porous medium (via array of spheres or similar strategies) fail to fully take into account the effect of the packing structure. Our novel approach relies on an “in-silico” procedure where many 3-D realistic porous media models are constructed via rigid-body simulations and fluid flow. We then investigate transport and deposition. Our approach thus affords a deeper look, afforded by these methodology, into the influence of the features of realistic porous media on particle transport and packing structure.

9:31AM G1.00008 Effective reaction rates for transport of particles to heterogeneous reactive (or porous) surfaces under shear. PREYAS SHAH, ERIC S. G. SHAQFEH, Stanford Univ — Mass transfer to heterogeneous reactive (or porous) surfaces is common in applications like heterogeneous catalysis, and biological porous media transport like drug delivery. This is modeled as advection-diffusion in a shear flow to an inert surface with first-order reactive patches. We study transport of point particles using boundary element simulations. We show that the heterogeneous surface can be replaced with a uniform flux boundary condition related to the Sherwood number (S), aka, the dimensionless flux to the reactive region. In the dilute limit of reactive regions, large-scale interaction between the reactive patches is important. In the dilute limit of inert regions, [S] grows as the reciprocal of the inert area fraction. Based on the method of resistances and numerical results, we provide correlations for [S] for general reactive surfaces and flow conditions. We model finite sized particles as general spheroids, specifically for biological applications. We do Brownian Dynamics simulations to account for hydrodynamic and steric interactions with the flow field and the domain geometry, and compare to the point particle results. We observe that anisotropic particles gave a higher pore transport flux compared to spherical particles at all flow conditions.

9:44AM G1.00009 Targeted delivery by smart capsules for controlling two-phase flow in porous media. JING FAN, Harvard University, ALIREZA ABBASPOURRAD, Cornell University, DAVID WEITZ, Harvard University, HARVARD WEITZGROUP TEAM — Two-phase flow in porous media is significantly influenced by the physical properties of the fluids and the geometry of the medium. We develop a variety of smart microcapsules that can deliver and release specific substances to the target location in the porous medium, and therefore change the fluid property or medium geometry at certain locations. In this talk, we will present two types of smart capsules for targeted surfactant delivery to the vicinity of oil-water interface and targeted microgel delivery for improving the homogeneity of the porous medium, respectively. We further prove the conjecture by monitoring the capsule location and the fluid structure in the porous media by micro-CT and confocal microscopy. This technique not only is of particular importance to the relevant industry applications especially in the oil industy but also opens a new window to study the mechanism of two-phase flow in porous media.

9:57AM G1.0010 Transport and Aggregation of Nanoparticles in Packed Beds: Effects of Pore Velocity and Initially-Fed Particle Size on Transient Particle Size Distributions. NGOC PHAM, DIMITRIOS PAPAVASSILIOU, The University of Oklahoma — Aggregation of colloidal particles in flow through porous media has received careful consideration, as it reduces particle breakthrough due to pore clogging and sedimentation. Additionally, in unstable colloidal systems, deposition of colloidal aggregates on the pore surfaces can create sub-surfaces for further colloidal attachment. This phenomenon is known as ripening effect. In this study, transient particle size distributions of nano-particle systems, propagating in a bed packed with spheres are numerically investigated. In our simulation, only pair interactions are considered, and the aggregation rate is varied with the relative position of two particles in a pair. The particles in a pack consist of spheres of known size, randomly packed in a simulation box. To generate the velocity field of water inside the porous medium, the lattice Boltzmann method (LBM) is used. In conjunction with that, the trajectories of thousands of massless particles moving with the flow under convection and diffusion are recorded employing a Lagrangian framework [1, 2]. While pore clogging is neglected, we draw attention to the change of the distribution of particle size under different pore velocities and different initially-fed particle sizes.


Monday, November 23, 2015 8:00AM - 10:10AM – Session G2 Suspensions: Gels and Soft Particles 101 - James Swan, MIT

8:00AM G2.00001 A parsimonious hydrodynamic model for colloidal gelation. ZSIGMOND VARGA, JAMES W. SWAN, Department of Chemical Engineering, Massachusetts Institute of Technology — Colloidal gels are formed during arrested phase separation. Models for microstructural evolution during gelation have often struggled to match experimental results with long standing questions regarding the role of hydrodynamics. We hypothesize that long-ranged hydrodynamic interactions between the suspended particles are key for colloidal gelation. A simplified hydrodynamic model tests this hypothesis by including only long-ranged interactions via the Rotne-Prager-Yamakawa tensor. We show simulations of gelation with and without hydrodynamic interactions between the suspended particles executed in HOOMD-blue. The disparities between these simulations are striking. The hydrodynamic simulations agree with experimental observations, however. These results suggest that long-ranged hydrodynamic interactions are sufficient for establishing the gel boundary, structure and coarsening kinetics observed in experiments and more sophisticated simulation methods. Near the gel boundary, there exists a competition between compaction of individual aggregates which suppresses gelation and coagulation of aggregates which enhances it. The time scale for coagulation is greatly accelerated, leading to a shift in the gel boundary when compared to models that neglect hydrodynamic interactions.
8:13AM G2.00002 Delayed yield in reversible colloidal gels: a micro-mechanical perspective, ROSEANNA ZIA, BENJAMIN LANDRUM, Cornell University, WILLIAM RUSSEL, Princeton University — We study via dynamic simulation the nonlinear response of a reversible colloidal gel undergoing deformation under fixed stress, with a view toward elucidating mechanisms of macroscopic yield at the level of particle dynamics. Under shear, such gels may flow but then regain solid-like behavior upon removal of the stress. The transition from solid-like to liquid-like behavior is a yielding process that is not instantaneous but rather occurs after a finite delay. The delay duration decreases as stress increases, but the underlying microstructural origins are not understood. Recent experiments reveal two regimes, suggesting multiple yield mechanisms. Theories advanced to link these two regimes to the linear theory. We also explain why other numerical and experimental studies lead to contradicting results. Furthermore, by performing simulations, we recover this effect and demonstrate that it persists for a wide range of parameters such as the concentration, membrane deformability and the swelling degree. We also explain why other numerical and experimental studies lead to contradicting results. Furthermore, our simulations show that this effect even persists in non-dilute and confined suspensions, but that it becomes less pronounced at higher concentrations and for more swollen particles [Kaoui et al., Soft Matter 10, 4735 (2014)]. The interplay of inertia and deformability has also a substantial impact on rheological properties. When a suspension of soft particles is subjected to Poiseuille flow, at finite Reynolds numbers, the Segre-Silberberg effect is suppressed and a flow focusing effect emerges, which is accompanied by a non-monotonic behavior of the suspension viscosity [Krueger et al., J. Fluid Mech. 751, 725 (2014)].

8:26AM G2.00003 ABSTRACT WITHDRAWN —

8:39AM G2.00004 Transient yield in reversible colloidal gels: a micro-mechanical perspective, LILIAN JOHNSON, BENJAMIN LANDRUM, ROSEANNA ZIA, School of Chemical and Biomolecular Engineering, Cornell University — We study the nonlinear rheology of colloidal gels via large-scale dynamic simulation, with a view toward understanding the micro-mechanical origins of the transition from solid-like to liquid-like behavior during flow startup, and post-cessation relaxation, and its connection to energy storage and viscous dissipation. Such materials often exhibit an overshoot in the stress during startup, but the underlying microstructural origins of this behavior remain unclear. To understand this behavior, a fixed strain rate is imposed on a reversible colloidal gel, where thermal fluctuations in the viscoelasticity have been suppressed. The model gel is comprised of soft uncharged particles that can adapt their conformation at the interface. In this study, we compute the shapes of soft elastic particles using molecular dynamics simulations of a cross-linked polymer gel, complemented by continuum calculations based on the linear elasticity. It is shown that the particle shape is not only affected by the Young’s modulus of the particle, but also strongly depends on whether the gel is partially or completely wetting the fluid interface. We find that the molecular simulations for the partially wetting case are very accurately described by the continuum theory. By contrast, when the gel is completely wetting the fluid interface the linear theory breaks down and we reveal that molecular details have a strong influence on the equilibrium shape.

8:52AM G2.00005 Deformation of ovalbumin-alginate capsules in a T-Junction, EDGAR HÄNER, ANNE JUEL, University of Manchester — We study experimentally the flow-induced deformation of liquid-filled ovalbumin-alginate capsules in a T-junction. In applications, capsules/cells often negotiate branched networks with junctions thus experiencing large deformations. We investigate the constant volume-flux viscous flow of buoyancy-neutral thin-walled capsules close to the centreline of rectangular channels, by comparison to near-rigid gelled beads. The motion of the capsules in straight channels with the capillary number the ratio of viscous to elastic forces. However, the effect of elastic deformation on the motion is sufficiently weak that a rigid sphere model predicts the velocity of capsules with diameters of up to 70% of that of the channel to within 5%. In the T-junction, systematic selection of daughter channel (right-left) occurs outside a finite region around the channel centreline, by contrast with near-rigid gelled beads, where the actual centreline is the separator. We quantify the behaviour of capsules in terms of their longitudinal stretching (up to a factor of three without rupture). We show the large range of deformations encountered can be applied to the measurement of the elastic properties of capsules as well as to the geometric-induced sorting and manipulation of capsules.

9:05AM G2.00006 Soft particles at a fluid interface, HADI MEHRABIAN, University of Twente, JENS HARTING, Eindhoven University of Technology, JACCO H. SNOEIJER, University of Twente — Particles added to a fluid interface can be used as a surface stabilizer in the food, oil and cosmetic industries. As an alternative to rigid particles, it is promising to consider highly deformable particles that can adapt their conformation at the interface. In this study, we compute the shapes of soft elastic particles using molecular dynamics simulations of a cross-linked polymer gel, complemented by continuum calculations based on the linear elasticity. It is shown that the particle shape is not only affected by the Young’s modulus of the particle, but also strongly depends on whether the gel is partially or completely wetting the fluid interface. We find that the molecular simulations for the partially wetting case are very accurately described by the continuum theory. By contrast, when the gel is completely wetting the fluid interface the linear theory breaks down and we reveal that molecular details have a strong influence on the equilibrium shape.

9:18AM G2.00007 Interplay of microdynamics and macro rheology in a suspension of fluid-filled soft particles, BADR KAOUUI, Bayreuth University, Germany — The microscopic dynamics of objects suspended in a fluid determines the macroscopic rheology of a suspension. As shown theoretically, the viscosity of a dilute suspension of vesicles is a non-monotonic function of the viscosity contrast (ratio between the viscosities of the encapsulated and the suspending fluids). By performing simulations, we recover this effect and demonstrate that it persists for a wide range of parameters such as the concentration, membrane deformability and the swelling degree. We also explain why other numerical and experimental studies lead to contradicting results. Furthermore, our simulations show that this effect even persists in non-dilute and confined suspensions, but that it becomes less pronounced at higher concentrations and for more swollen particles [Kaoui et al., Soft Matter 10, 4735 (2014)]. The interplay of inertia and deformability has also a substantial impact on rheological properties. When a suspension of soft particles is subjected to Poiseuille flow, at finite Reynolds numbers, the Segre-Silberberg effect is suppressed and a flow focusing effect emerges, which is accompanied by a non-monotonic behavior of the suspension viscosity [Krueger et al., J. Fluid Mech. 751, 725 (2014)].

9:31AM G2.00008 Buckling and its effect on the confined flow of a model capsule suspension, SPENCER BRYNGELSON, JONATHAN FREUND, University of Illinois at Urbana-Champaign — The rheology of confined flowing suspensions, such as blood, depend upon the dynamics of the components, which can be particularly rich when they are elastic capsules. Using boundary integral methods, we simulate a two-dimensional model channel through which flows a dense suspension of fluid-filled capsules. A parameter of principal interest is the equilibrium membrane perimeter, which ranges from round capsules to capsules with an elongated dog-bone-like equilibrium shape. It is shown that the minimum effective viscosity occurs for capsules with a biconcave equilibrium shape, similar to that of a red blood cell. The rheological behavior changes significantly over this range; transitions are linked to specific changes in the capsule dynamics. Most noteworthy is an abrupt change in behavior when capsules transition to a dog-bone-like equilibrium shape, which correlates with the onset of capsule buckling. The buckled capsules have a more varied orientation and make significant rotational (rotlet) contributions to the capsule–capsule interactions.

1Supported under NSF Grant No. CBET 13-30972.
The mean inward RV vs. particle separation distance increases with mass loading ratio on the particle velocities is analyzed. These will increase transport of particles away from the walls and raise turbulence levels in the central region. While the effect of particles on the gas flow is negligible, the square channel flow contains mean secondary flows not present in high aspect ratio geometries. Low volume and mass loading ratios are considered. Under these conditions, preferential concentration is expected to be strong and particle clustering is confirmed by our analysis.

Number DE-NA0002373-1.

The behavior of suspensions of different shapes is found in several natural and industrial systems, examples including biological cells, elastic capsules, and microgels. When suspended in a flowing liquid, such deformable particles can exhibit complicated dynamics in response to the hydrodynamic forces exerted by the suspending fluid and, in turn, have a significant impact on the overall mechanical properties of the multiphase material. The behavior of suspensions of soft particles, both of spherical and non-spherical shape at rest, in Newtonian and viscoelastic fluids subjected to shear flow is studied through direct numerical simulations. In both Newtonian and viscoelastic matrices, initially spherical particles are found to deform and eventually migrate orthogonally to the flow, the direction and velocity of such migration being determined by the interplay of the geometrical and the rheological parameters of the system. Non-spherical particles have even more complex dynamics due to their non-trivial undeformed shape, which introduces additional parameters to the system.

### Monday, November 23, 2015 8:00AM - 10:10AM –
Session G3 Particle-Laden Flows: Clustering and Dispersion I

#### 8:00AM G3.00001 Inertial Particle Relative Velocity in a High-Reynolds-Number Homogeneous and Isotropic Turbulence Chamber

ZHONGWANG DOU, ZACHARY PECENAK, ZACH LIANG, University at Buffalo - SUNY, LUIJIE CAO, Ocean University of China, PETER IRELAND, LANCE COLLINS, Cornell University, HUI MENG, University at Buffalo - SUNY — Particle-pair radial relative velocity (RV) in turbulence plays a critical role in droplet collision and cloud formation. Both simulations and experiments are performed to better understand RV of inertial particles in homogeneous and isotropic turbulence (HIT). However, past experimental measurement of particle RV statistics exhibited large deviations from DNS results (de Jong et al., 2010). In the current study, we identified intrinsic limitations in our previous study and devised a 4-frame particle tracking velocimetry technique to measure particle RV. In a second-generation, enclosed, fan-driven HIT chamber, both tracer and inertial particles were studied at Re,λ of 366. The experimentally measured RV statistics were compared with DNS with excellent agreement. Additionally, for both kinds of particles, the mean inward RV vs. particle separation distance also matched very well with DNS, but at near-zero r, experimental values were slightly higher. To investigate the cause of this discrepancy, we compared DNS of both mono- and tri-dispersed particles. We found that the tridispersed particles exhibited higher mean inward RV at small r than any mono-dispersed RV particles. This suggests that the increase of mean inward RV in the experiment could be due to the Stokes number (St) distribution present in the particles, while DNS employed single St values.

This work was supported by the National Science Foundation through a Collaborative Research Grant CBET-0967407.

#### 8:13AM G3.00002 Turbulent particle clustering in a fully developed square channel flow

LAURA VILLAFANE, ANDREW BANKO, CHRIS ELKINS, JOHN EATON, Stanford University — Particle-turbulence interactions are investigated in a fully developed turbulent channel air flow to determine the gas phase effect on the particle concentration and velocity fields. The experimental apparatus is a vertical channel with square cross section. The Reynolds number based on channel width is about 10^4. The 12 um nominal diameter nickel particles are smaller than the Kolmogorov length scale and the corresponding Stokes number is of the order of one. Low volume and mass loading ratios are considered. Under these conditions, preferential concentration is expected to be strong while the effect of particles on the gas flow is negligible. The square channel flow contains mean secondary flows not present in high aspect ratio channels studied previously. These will increase transport of particles away from the walls and raise turbulence levels in the central region. Current experiments are focused on the statistics of the particle phase including particle concentration inhomogeneities and particle velocities. The particle concentration field is analyzed from high resolution laser illuminated planar images. Particle velocity distributions are measured by increasing mass loading ratio on the particle velocities is analyzed.

This work was supported by the National Science Foundation through a Collaborative Research Grant CBET-0967407.

#### 8:26AM G3.00003 An asymptotic analysis of particle clustering in turbulent flows

MAHDI ESMAILY MOGHADAM, ALI MANI, Stanford Univ — Interaction of dense inertial particles with turbulent flow is analysed. An asymptotic solution is obtained by quantifying the particle clustering on a wide range of Stokes numbers and flow conditions. In a simplified form, particle clustering is predicted to be St/(St^2 + 1), in which St is the Stokes number based on the Kolmogorov time scale, hence predicting maximum clustering at St = 1 and first order decay of clustering as St → 0 and ∞. These results are validated against numerical simulation of inertial particles in a homogeneous isotropic turbulent flow. This comparison shows excellent prediction of our analysis at all Reynolds and Stokes numbers with a slight under-prediction when both Reynolds and Stokes numbers are high. The important role of Kolmogorov scale on particle clustering is confirmed by our analysis.

This is based upon work supported by the Department of Energy National Nuclear Security Administration under Award Number DE-NA0002373-1.

### Notes

1. This work was supported by NSF grant CBET-1436082.

2. This work was supported by the National Science Foundation through a Collaborative Research Grant CBET-0967407.

3. This is based upon work supported by the Department of Energy National Nuclear Security Administration under Award Number DE-NA0002373-1.
stability of Particle Laden Homogeneous Shear, MOHAMED KASBAOUI, Sibley School for Mechanical and Aerospace Engineering, Cornell University, DONALD KOCH, School of Chemical and Biomolecular Engineering, Cornell University, OLIVIER DESJARDINS, Sibley School for Mechanical and Aerospace Engineering, Cornell University — In a previous study (Kasbaoui et al, J. Fluid Mech. 2015), particle laden homogeneous shear was shown to be subject to an algebraic instability. Initially randomly distributed particles are entrained by wave-like perturbations in the fluid velocity and segregate in a similar wave-like pattern while they sediment under gravity. The preferential concentration mechanism, which is the tendency of particles to exit vortical regions and gather in straining regions, causes the two waves to amplify each other resulting in an algebraic instability. By means of simulations, we compare the perturbations growth to the one yielded by the theory in the limit of small Stokes number particles. The simulations are conducted with an Eulerian model of the particles as well as a Lagrangian model. The two are compared. A secondary Rayleigh-Taylor instability caused by the periodic stacking of heavy layers of concentrated particles on top of depleted lighter layers is analyzed.

DESJARDINS, Sibley School for Mechanical and Aerospace Engineering, Cornell University — Computational capabilities have matured sufficiently to render possible the dynamic simulation of thousands of resolved particles in fluid flows, generating an unprecedented amount of data. In this work we present a simulation of 2,000 fluidized particles generated with the Physalis method, and focus on probing the data with tools from statistical physics. In particular, the study of particle triads and tetrads has been used to study the dispersion of passive scalars in turbulence. Knowledge of the average shape and size of these structures over time provides insight into particle diffusion and the persistence of clusters.

We will present measurements of turbulent flow and particle dynamics using flow visualization techniques to understand the multiphase fluid behavior. Glass spheres with a mean diameter of 35-45 microns are used in experiments as a surrogate for airborne particulates encountered during flight. Particle Separators (IPS) have been one common effective method but they experience complex internal particle-laden flows that are challenging to understand and model. This research employs an IPS test rig to simulate dust particle separation under different flow conditions. Soda lime glass spheres with a mean diameter of 35-45 microns are used in experiments as a surrogate for airborne particulates encountered during flight. We will present measurements of turbulent flow and particle dynamics using flow visualization techniques to understand the multiphase fluid behavior. This knowledge can contribute to design better performing IPS systems for UAVs.

We present a simulation of 2,000 fluidized particles generated with the Physalis method, and focus on probing the data with tools from statistical physics. In particular, the study of particle triads and tetrads has been used to study the dispersion of passive scalars in turbulence. Knowledge of the average shape and size of these structures over time provides insight into particle diffusion and the persistence of clusters.

1Supported by NSF award No CBET 1335965

9:05AM G3.00006 Particle dispersion in an inhomogeneous turbulent flow, PETER HUCK, ROMAIN VOLK, Ecole Normale Superieure de Lyon — An experimental study of the von Kármán swirling flow in counter rotation at $R_e = 200$ is presented and investigates the interaction of tracer particles with a spatially inhomogeneous environment. Using ombroscopic PTV, eulerian conditioning of lagrangian trajectories permits visualization of the well known stagnation point topology and allows the calculation of lagrangian statistics which reveal the small scale anisotropy characteristic of this flow. Neighboring regions dominated by rotation or shear are also presented as anisotropy varies with respect to the predominant mean flow. The length of particle tracks needed to estimate velocity correlations is considerable and renders the rms velocity non stationary as particles explore their inhomogeneous surroundings. We conclude on the role of inhomogeneity and non stationarity in the concomitant process of particle dispersion.

9:18AM G3.00007 Laboratory measurement of non-spherical particle rotation in turbulence: analysis in lab and local reference frames, ANKUR BORDOLOI, EVAN VARIANO, University of California Berkeley — Small axisymmetric particles are known to show shape dependence in their rotational kinematics in homogeneous isotropic turbulence. For example, Byron et al. (2015) demonstrated that, rod-shaped particles rotate very differently compared to disc-shaped ones. This motivates us to extend this understanding to non-spherical particles. We have overcome the experimental challenges that have heretofore prevented the simultaneous measurement of orientation and rotation of large particles. We present this method and report results for large cylinders from 2D3C stereoscopic PIV data. These preliminary results show that there is equipartition of particle enstrophy into spinning about each of the particle’s local axes. In other words, there is no preference for rotation about a particle’s symmetry axis. Time permitting, effects of size on rotation for Taylor microscale particles will also be discussed.

9:3AM G3.00008 Visualization of Air Particle Dynamics in an Engine Inertial Particle Separator$^1$, JASON WOLF, WEI ZHANG, Cleveland State Univ — Unmanned Aerial Vehicles (UAVs) are regularly deployed around the world in support of military, civilian and humanitarian efforts. Due to their unique mission profiles, these advanced UAVs utilize various internal combustion engines, which consume large quantities of air. Operating these UAVs in areas with high concentrations of sand and dust can be hazardous to the engines, especially during takeoff and landing. In such events, engine intake filters quickly become saturated and clogged with dust particles, causing a substantial decrease in the UAVs’ engine performance and service life. Development of an Engine Air Particle Separator (EAPS) with high particle separation efficiency is necessary for maintaining satisfactory performance of the UAVs. Inertial Particle Separators (IPS) have been one common effective method but they experience complex internal particle-laden flows that are challenging to understand and model. This research employs an IPS test rig to simulate dust particle separation under different flow conditions. Small glass spheres with a mean diameter of 35-45 microns are used in experiments as a surrogate for airborne particulates encountered during flight. We will present measurements of turbulent flow and particle dynamics using flow visualization techniques to understand the multiphase fluid behavior. This knowledge can contribute to design better performing IPS systems for UAVs.

$^1$Cleveland State University, Cleveland, Ohio, 44115

9:44AM G3.00009 Phoresis-induced clustering of particles in turbulence, LUKAS SCHMIDT, ETH Zurich, ITZHAK FOUXON, Yonsei University Seoul, DOMINIK KRUG, University of Melbourne, MAARTEN VAN REEUWIJK, Imperial College London, MARKUS HOLZNER, ETH Zurich — We demonstrate phoresis-induced clustering of non-inertial particles in turbulent flows. Phoretic mechanisms such as thermophoresis, chemotaxis or diffusiochemotaxis are known to create a particle drift with respect to the fluid. Theory, based on the framework of weakly compressible flow, predicts that particles in turbulence streaked by salinity gradients experience a diffusiochemotactic drift and will thus form particle clusters. An inclined gravity current setup is used to analyse clustering due to the diffusiochemotactic effect in turbulent flow experimentally. Simultaneous 3D particle tracking velocimetry and laser induced fluorescent measurements provide the full Lagrangian velocity field and the local salt concentration in the observed 3D domain. Two independent methods show consistent evidence of the theoretically predicted particle clustering in turbulence. This clustering mechanism can provide the key to the understanding of spontaneous clustering phenomena such as the formation of marine snow in the ocean.
9:57AM G3.00010 Clustering and relative velocity of heavy particles under gravitational settling in isotropic turbulent flows. GUODONG JIN, GUO-WEI HE, LNM, Institute of Mechanics, Chinese Academy of Sciences — Clustering and intermittency in radial relative velocity (RRV) of heavy particles settling in turbulent flows can be remarkably changed due to gravity. Clustering is monotonically reduced at Stokes number less than 1 under gravity due to the disability of the centrifugal mechanism, however it is non-monotonically enhanced at Stokes number greater than 1 due to the multiplicative amplification in the case that the proposed effective Kubo number is less than 1. Although gravity causes monotonical reduction in the rms of RRV of particles at a given Stokes number with decreasing Froude number, the variation tendency in the tails of standardized PDF of RRV versus Froude number is obviously different, the tails become narrower at a small Stokes number, while they become broader at a large Stokes number. The mechanism of this variation stems from the compromise between the following two competing factors. The mitigation of correlation of particle positions and the regions of high strain rate which are more intermittent reduces the intermittency in RRV at small Stokes numbers, while the significant reduction in the backward-in-time relative separations will make particle pairs see small-scale structures, leading to a higher intermittency in RRV at large Stokes numbers.

1NSAF of China (grant number U1230126);NSFC (grant numbers 11072247 and 11232011)

Monday, November 23, 2015 8:00AM - 10:10AM –
Session G4 General Fluid Dynamics: Obstacles, Flow Constrictions, Channels 103 -
Srinivas Kosaraju, Northern Arizona University

8:00AM G4.00001 Effect of Local Junction Losses in the Optimization of T-shaped Flow Channels, SRINIVAS KOSARAJU, Northern Arizona University — T-shaped channels are extensively used in flow distribution applications such as irrigation, chemical dispersion, gas pipelines and space heating and cooling. The geometry of T-shaped channels can be optimized to reduce the overall pressure drop in stem and branch sections. Results of such optimizations are in the form of geometric parameters such as the length and diameter ratios of the stem and branch sections. The traditional approach of this optimization accounts for the pressure drop across the stem and branch sections, however, ignores the pressure drop in the T-junction. In this paper, we conduct geometry optimization while including the effect of local junction losses in laminar flows. From the results, we are able to identify a non-dimensional parameter that can be used to predict the optimal geometric configurations. This parameter can also be used to identify the conditions in which the local junction losses can be ignored during the optimization.

8:13AM G4.00002 Wakes from submerged obstacles in an open channel flow, GEOFFREY B. SMITH, GEORGE MARMORINO, Naval Research Laboratory, CHARLES DONG, UCLA, W. D. MILLER, RICHARD MIED, Naval Research Laboratory — Wakes from several submerged obstacles are examined via airborne remote sensing. The primary focus will be bathymetric features in the tidal Potomac river south of Washington, DC, but others may be included as well. In the Potomac the water depth is nominally 10 m with an obstacle height of 8 m, or 80% of the depth. Infrared imagery of the water surface reveals thermal structure suitable both for interpretation of the coherent structures and for estimating surface currents. A novel image processing technique is used to generate two independent scenes with a known time offset from a single overpass from the infrared imagery, suitable for velocity estimation. Color imagery of the suspended sediment also shows suitable texture. Both the ‘mountain wave’ regime and a traditional turbulent wake are observed, depending on flow conditions. Results are validated with in-situ ADCP transsects. A computational model is used to further interpret the results.

8:26AM G4.00003 The effect of a large upstream bluff body on the flow through and around an array of cylinders, CHRISTIAN KLETTNER, University College London — Risers from the sea floor to installations at the sea surface are an integral part of the flow assurance of deep sea oil exploration, which has become necessary in the last decade. These risers are subjected to various hydrodynamic forcing, particularly sea currents, wakes of upstream installation members and surface waves, which can result in vortex or wake induced vibrations and these in turn can result in riser fatigue or collision. The free stream flow past groups of cylinders has been studied numerically in two- and three-dimensions for Re ~ O(10^3) (based on the cylinder diameter) by Nicolle & Eames (2011) (direct numerical simulations) and Chang & Constantinescu (2015) (large eddy simulations) respectively. In this study we will be focusing on the first two aspects listed above, specifically laboratory experiments and high resolution numerical simulations will be performed to investigate the effect of an upstream truss on an array of cylinders. The main diagnostic will be how varying the void fraction of the array of cylinders affects the frequency and magnitude of the drag and lift forces on the cylinders.

8:39AM G4.00004 Optimal design of artificial reefs for sturgeon, CODY YARBROUGH, ALINE COTEL, ABBY KLEINHEKSEL, University of Michigan — The Detroit River, part of a busy corridor between Lakes Huron and Erie, was extensively modified to create deep shipping channels, resulting in a loss of spawning habitat for lake sturgeon and other native fish (Caswell et al. 2004, Bennion and Manny 2011). Under the U.S.– Canada Great Lakes Water Quality Agreement, there are remediation plans to construct fish spawning reefs to help with historic habitat losses and degraded fish populations, specifically sturgeon. To determine optimal reef design, experimental work has been undertaken. Different sizes and shapes of reefs are tested for a given set of physical conditions, such as flow depth and flow velocity, matching the relevant dimensionless parameters dominating the flow physics. The physical conditions are matched with the natural conditions encountered in the Detroit River. Using Particle Image Velocimetry, Acoustic Doppler Velocimetry and dye studies, flow structures, vorticity and velocity gradients at selected locations have been identified and quantified to allow comparison with field observations and numerical model results. Preliminary results are helping identify the design features to be implemented in the next phase of reef construction.

1Sponsored by NOAA

8:52AM G4.00005 Physics in water slides, JEAN-BAPTISTE THOMAZO, ETIENNE REYSSAT, MARC FERMIGIER, PMMH, ESPCI — Water slides are body-size inclined pipes fed with water to improve sliding. Water is allowed to freely flow down the slide. It forms a lubrication film that reduces friction between the slide and the body, allowing sliders to travel down at high speeds. We present the results of an experimental study on a model water slide at the scale of the laboratory. We analyze the sliding velocities of cylindrical objects of various masses and sizes sliding down an inclined gutter fed with a controlled flux of water. In the range of parameters that we have studied, we show that the speed of the model sliders is faster than the flow of the environing water. We propose a minimal model to account for the observed sliding velocities measured in our experiments. The sliding velocity is set by a balance of the apparent weight with inertial drag or viscous friction in the lubrication film under the slider. Other resisting mechanisms will also be discussed.
The classic work by G.I. Taylor describes the enhanced longitudinal diffusivity of a passive tracer in laminar pipe flow. Much work since then has gone into extending this result particularly in calculating the evolution of the scalar variance. However, less work has been done to describe the asymmetry of the distribution. We present the results from a modeling effort for the general picture of how the higher moments of the tracer distribution depend on the geometry of the system. We explore analytically, numerically and experimentally the evolution of the dispersion of a solute, focusing primarily on computing and measuring the first four moments (mean, variance, skewness and kurtosis) of solute concentration in two-dimensional channel models and three-dimensional glass pipes with circular or square cross-sections. Our experimental setup allows us to observe the effects of Poiseuille flow as either advection or diffusion dominates in different regimes and the position of the interface between the two fluids has been varied in the vertical direction. Two cases have been considered, one where interface is above the crests plane, and one where the interface is below the crests plane. When a thin film of low viscosity fluid is above the crests, the stagnation point on the leading edge of the roughness elements moves upward and the form drag decreases thus leading to a drag reduction of up to 30%. Drag reduction can be achieved even for $m > 1$ due to the damping of wall normal velocity fluctuations.

1 Numerical simulations were performed on XSEDE TACC under Grant CTS070066. This work was supported by ONR MURI grants N00014-12-01-0875 and N00014-12-01-0962.

9:18AM G4.00007 Geometric Skewness in the Passive Tracer Problem1, MANUCHEHR AMINIAN, FRANCESCA BERNARDI, ROBERTO CAMASSA, RICHARD MCLAUGHLIN, UNC Chapel Hill — The classic work by G.I. Taylor describes the enhanced longitudinal diffusivity of a passive tracer in laminar pipe flow. Much work since then has gone into extending this result particularly in calculating the evolution of the scalar variance. However, less work has been done to describe the asymmetry of the distribution. We present the results from a modeling effort for the general picture of how the higher moments of the tracer distribution depend on the geometry of the system. We do this via analysis of “channel-limiting” geometries (rectangular ducts and elliptical pipes parameterized by their aspect ratio), using both new analytical tools and Monte-Carlo simulation, which have revealed a wealth of nontrivial behavior of the distributions at short and intermediate time.

1 Funding from NSF grant Nos.: RTG DMS-0943851, CMG ARC-1025523, and DMS-1009750.

9:31AM G4.00008 Experimental Analysis of the Diffusion of a Passive Scalar Subject to Steady Flow in a Circular Pipe1, FRANCESCA BERNARDI, MANUCHEHR AMINIAN, University of North Carolina - Chapel Hill, SARAH BURNETT2, Los Alamos National Laboratory, ROBERTO CAMASSA, RICHARD MCLAUGHLIN, University of North Carolina - Chapel Hill — The Taylor Pipe Flow experiment at UNC’s Joint Fluids Lab was designed to be a continuation of the research on the dispersion of soluble matter through a tube conducted by G.I. Taylor in the ’50s. We explore analytically, numerically and experimentally the evolution of the dispersion of a solute, focusing primarily on computing and measuring the first four moments (mean, variance, skewness and kurtosis) of solute concentration in two-dimensional channel models and three-dimensional glass pipes with circular or square cross-sections. Our experimental setup allows us to observe the effects of Poiseuille flow as either advection or diffusion dominates in different regimes and timescales set by the Taylor time scale $\tau_T \propto a^2/\kappa$, depending on the cross-sectional characteristic length $a$ and the diffusion coefficient $\kappa$. We conduct experiments to illustrate these regimes, characterized by the dimensionless Péclet number, $Pe = u a/\kappa$, where $u$ is the characteristic velocity. Experimentally, we take the intensity of a fluorescein-dyed portion of distilled water and find its corresponding concentration by solving an inverse problem of intensity to concentration. The experimental results validate the theoretical approach.

1 Founded by NSF Grant Nos.: RTG DMS-0943851, CMG ARC-1025523, DMS-1009750.

2 Graduated from University of North Carolina - Chapel Hill.

9:44AM G4.00009 Experimental Investigation of Spatially-Periodic Scalar Patterns in an Inline Mixer, OZGE BASKAN, MICHEL F. M. SPEETJENS, HERMAN J. H. CLERCX, Eindhoven University of Technology — Spatially persisting patterns with exponentially decaying intensities form during the downstream evolution of passive scalars in three-dimensional (3D) spatially periodic flows due to the coupled effect of the chaotic nature of the flow and the diffusivity of the material. This has been investigated in many computational and theoretical studies on 3D spatially-periodic flow fields. However, in the limit of zero-diffusivity, the evolution of the scalar fields results in more detailed structures that can only be captured by experiments due to limitations in the computational tools. Our study employs the state-of-the-art experimental methods to analyze the evolution of 3D advective scalar field in a representative inline mixer, called Quatro static mixer. The experimental setup consists of an optically accessible test section with transparent internal elements, accommodating a pressure-driven pipe flow and equipped with 3D Laser-Induced Fluorescence. The results reveal that the continuous process of stretching and folding of material creates finer structures as the flow progresses, which is an indicator of chaotic advection and the experiments outperform the simulations by revealing far greater level of detail.

9:57AM G4.00010 Asymptotic scalings of developing curved pipe flow, JESSE AULT, Princeton University, KEVIN CHEN, University of Southern California, HOWARD STONE, Princeton University — Asymptotic velocity and pressure scalings are identified for the developing curved pipe flow problem in the limit of small pipe curvature and high Reynolds numbers. The continuity and Navier-Stokes equations in toroidal coordinates are linearized about Dean’s analytical curved pipe flow solution (Dean 1927). Applying appropriate scaling arguments to the perturbation pressure and velocity components and taking the limits of small curvature and large Reynolds number yields a set of governing equations and boundary conditions for the perturbations, independent of any Reynolds number and pipe curvature dependence. Direct numerical simulations are used to confirm these scaling arguments. Fully developed straight pipe flow is simulated entering a curved pipe section for a range of Reynolds numbers and pipe-to-curvature radius ratios. The maximum values of the axial and secondary velocity perturbation components along with the maximum value of the pressure perturbation are plotted along the curved pipe section. The results collapse when the scaling arguments are applied. The numerically solved decay of the velocity perturbation is also used to determine the entrance/development lengths for the curved pipe flows, which are shown to scale linearly with the Reynolds number.
Finally, we consider some practical implications of our results for inducing air movement in rooms. The Reynolds number was varied from 10 to 200. The jet is laminar when the Reynolds number is less than 60. If the Reynolds number exceeds 60, the jet is flapping. If the Reynolds number further exceeds 150, the flow fluctuations in the jet are significant, and the jet is flapping. Next, a numerical simulation was performed for the two-dimensional jet at the Reynolds number of 300. When the spatio-temporally random disturbance is given at the issuing velocity, the velocity fluctuation is generated at peak frequency corresponding to instability in the share layer. However, the velocity fluctuation decays immediately. If the spatially asymmetrical disturbance is given, the velocity fluctuation is generated at peak frequency of the order of 1/100 of the above frequency. The asymmetrical fluctuation rapidly grows and the higher frequencies are generated in the fluctuation.

8:13AM G5.00002 Mapping the Interactions between Shocks and Mixing Layers in a 3-Stream Supersonic Jet1, JACQUES LEWALLE, Syracuse University, CHRISTOPHER RUSCHER, Spectral Energies LLC, PINQING KAN, ANDREW TENNEY, Syracuse University, SIYAMAR GOGINENI, Spectral Energies LLC, BARRY KIEL, Air Force Research Laboratory — Pressure is obtained from an LES calculation of the supersonic jet (Ma = 1.0) issuing from a rectangular nozzle in a low-subsonic co-flow; a tertiary flow, also rectangular with Ma = 1 insulates the primary jet from an aft-deck plate. The developing jet exhibits complex three-dimensional interactions between oblique shocks, multiple mixing layers and corner vortices, which collectively act as a skeleton for the flow. Our study is based on several plane sections through the pressure field, with short signals (0.1 s duration at 80 kHz sampling rate). Using wavelet-based band-pass filtering and cross-correlation, we map the directions of propagation of information among the various “bones” in the skeleton. In particular, we identify upstream propagation in some frequency bands, 3-dimensional interactions between the various shear layers, and several key bones from which the pressure signals, when taken as reference, provide dramatic phase-locking for parts of the skeleton.

1We acknowledge the support of AFRL through an SBIR grant.

8:26AM G5.00003 Jet-mixing of initially stratified flows1, STUART WRIGHT, CHRISTOS MARKIDES, OMAR MATAR, Imperial College London — Low pipeline velocities in the oil-and-gas industry generally lead to liquid-liquid flows stratifying due to density differences. Pipeline stratified flows inherently have no single point for sub-sampling and phase slip leads to in situ phase fractions differing from input volume fractions. Establishing representative or average properties and phase fractions is therefore difficult for industry. This leads to sampling errors through measurement uncertainty. In-line mixing overcomes liquid-liquid stratification, establishing a liquid-liquid dispersion that minimises slip between phases. Here, we use jets-in-crossflow (JICF) as a means of mixing. We present results of CFD simulations using the volume-of-fluid method that demonstrate the breakup of stratification as a result of the application of JICF. A number of simple jet configurations are described, and their effectiveness in generating dispersions is compared. We also present preliminary experimental results based on the use of a matched-refractive-index method, laser-induced fluorescence, particle-tracking and particle-image velocimetry.

1Funding from Cameron for Ph.D. studentship (SW) gratefully acknowledged.

8:39AM G5.00004 CFD simulation of boundary effects on closely spaced jets, ISHITA SHRIVASTAVA, ERIC ADAMS, Massachusetts Inst of Tech-MIT — In coastal areas characterized by shallow water depth, industrial effluents are often diluted using multiple closely spaced jets. Examples of such effluents include brine from desalination plants, treated wastewater from sewage treatment plants and heated water from thermal power plants. These jets are arranged in various orientations, such as unidirectional diffusers and rosette groups, to maximize mixing with ambient water. Due to effects of dynamic pressure, the jets interact with each other leading to mixing characteristics which are quite different from those of individual jets. The effect of mutual interaction is exaggerated under confinements, when a large number of closely spaced jets discharge into shallow depth. Dilution through an outfall, consisting of multiple jets, depends on various outfall and ambient parameters. Here we observe the effects of shoreline proximity, in relation to diffuser length and water depth, on the performance of unidirectional diffusers discharging to quiescent water. For diffusers located closer to shore, less dilution is observed due to the limited availability of ambient water for dilution. We report on the results of Computational Fluid Dynamics (CFD) simulations and compare the results with experimental observations.

8:52AM G5.00005 Entrainment in a Free Surface Plunging Jet, SYED HARRIS HASSAN, PAVLOS P. VLACHOS, Purdue Univ — Ambient fluid entrainment and near-field flow characteristics of a free surface plunging jet are investigated using time resolved particle image velocimetry. The plunging height is about twice the nozzle diameter and we study five different Reynolds numbers ranging from Re$\_z = 5000$ to Re$\_z = 13,070$. We found that the near-field entrainment in the Re$\_z = 5000$ and Re$\_z = 6086$ jets is enhanced significantly than the latter cases due to mixing transition that occurs at about 2-6D$\_n$ below the free surface resulting in the breakdown of laminar vortices into smaller secondary structures. With an increase in the Reynolds number, mixing transition declines in the latter three cases as they have a more homogenous and turbulent flow structure. In addition, plunging jets show a reduction in the penetration depth of about 20-30% and the length of Zone of Flow Establishment (ZFE) of about 40-60% when compared to free jets at the same Reynolds numbers. Finally, we look into the distribution of coherent structures in the near-field over time in order to find the depth of mixing transition. The depth where primary structures breakdown into smaller secondary structures decreases as the Reynolds number is increased and is consistent with the reduction in the length of ZFE.

9:05AM G5.00006 The development of a turbulent jet issuing from an annular source with an open centre1, SHAHID PADHANI, University of Cambridge, TIM JUKES, Dyson Technology Ltd, GARY R. HUNT, University of Cambridge — Using particle image velocimetry (PIV) and flow visualisation we investigate the dynamics of a turbulent jet that issues from a thin slot annular source with an open centre - by "thin" we refer to a slot width that is at least an order of magnitude smaller than the diameter of the annulus. For this geometry, entrainment into the near-field region of the jet results in an induced flow in the environment that passes through the open annulus. We explore the streamwise development of the jet towards an approximately self-similar form and identify the key regions of the flow. For the far-field, we show that the flow approximately to a round jet from a circular source and we use our results to estimate the location of the origin for the virtual point source of a round jet that matches the far-field flow produced by an annular source. Finally, we consider some practical implications of our results for inducing air movement in rooms.

1This work is supported by Dyson Technology Ltd.
9:18AM G5.00007 Large Eddy Simulations and an Analysis of the Flow Field of a Radially Lobed Nozzle. NOUSHIN AMINI, Texas A&M University, AARThI SEKARAN, Jawaharlal Nehru Center for Advanced Scientific Research — Lobed nozzles have been a subject of regained interest over the past couple of decades owing to their established mixing capabilities. Despite experimental (Hu et al, 1999 and Hu et al, 2008) and limited numerical studies (Bouloufa et al 2011 and Cooper et al, 2005), the exact nature of the jet ensuing from this nozzle is yet to be completely understood. The present numerical study is intended to complement prior experimental investigation, involving the analysis of the flow field downstream of a six lobed nozzle (Amini et al, 2012). Preliminary results (presented at DFD 2014, Amin and Sekaran), which involved three dimensional simulations of the full domain via URANS and Large Eddy Simulations (LES) were used to assess the domain extents and simulation technique. Based on these results it was seen that LES were able to capture the region of interest satisfactorily and a qualitative corroboration with previous studies was obtained. The study is thus extended to analyzing the flow originating from within the nozzle, following it downstream in order to confirm the vortical interaction mechanisms inside the lobed nozzle.

9:31AM G5.00008 Modal and non-modal evolution of perturbations for parallel round jets. JOSE IGNACIO JIMENEZ-GONZALEZ, Universidad de Jaen, PIERRE BRANCHER, Institut de Mecanique des Fluides de Toulouse, CARLOS MARTINEZ-BAZAN, Universidad de Jaen — We investigate the local modal and non-modal stability of round jets for varying aspect ratios and Reynolds numbers. The competition between axisymmetric (azimuthal wavenumber m = 0) and helical (m = ±1) perturbations depending on the jet aspect ratio, \( \alpha = R/\theta \), where R is the jet radius and \( \theta \) the shear layer momentum thickness, is quantified at different time horizons. Optimal excitation and optimal perturbation analyses allow us to characterize the transient dynamics of jets, showing that two mechanisms may cause large energy gains, namely the Orr mechanism at small wavelengths and the “shift-up” mechanism, in the long wavelength limit, which is found to shift the jet as a whole in a way that resembles the classical lift-up effect active in wall shear flows. The “shift-up” mechanism is found to be especially efficient for vanishing perturbations axial wavenumbers. Furthermore, it has been found that adjoint modes drive the transient process at relatively short temporal horizons, in such a way that, for large aspect ratios, an optimal excitation analysis might suffice to properly characterize transient dynamics.

1Supported by the Spanish MINECO, Junta de Andaluca and EU Funds under projects DPI2011-28356-C03-03 and P11-TEP7405.

9:44AM G5.00009 Numerical simulation of air-blast atomization of a liquid layer. G. GILOU AGBAGLAH, JEREMY MCCASLIN, OLIVIER DESJARDINS, Cornell University — Numerical simulations of a planar co-flowing air/water airblast atomization is performed using an in-house multiphase Navier-Stokes solver based on a semi-lagrangian geometric Volume of Fluid (VOF) method to track the position of the interface. This solver conserves mass and momentum exactly within each phase. Excellent agreement with recent experiments is obtained when comparing physical quantities, such as the liquid cone length, the maximum wave frequency and the spatial growth rate of the primary instability. A full three dimensional simulation is used to analyze the turbulence in the gas phase. The gas layer is laminar close to the injector and becomes turbulent at downstream positions. The transition to the turbulence is shown to increase first as an exponential function of the downstream positions and then reach a statistically stable regime where the liquid wave crests expand in a thin sheet which breaks into secondary droplets.

Monday, November 23, 2015 8:00AM - 9:57AM – Session G6 CFD: Lattice Boltzmann Methods 105 - Julien Favier, Aix Marseille University

8:00AM G6.00001 Simulation of immersed moving porous bodies using a coupled Immersed Boundary - Lattice Boltzmann method. Application to the control of flow separation around bluff bodies., MARIANNA PEPCA, JULIEN FAVIER, Aix Marseille University, M2P2 UMR7340 — A numerical framework to simulate fluid flows in interaction with moving porous bluff bodies of complex geometry is proposed in this work. It is based on the Generalized Lattice Boltzmann method, which models the flow in the Representative Elementary Volume scale including the porous effects (porosity and the Brinkman-Forchheimer extended Darcy force model), coupled to the Immersed Boundary method to handle complex geometries and moving bodies. The coupling between both methods will be presented and the numerical results will be discussed in both porous and rigid configurations. The effect of the structure permeability on the boundary layer separation around a bluff body will be studied in the case of a static body and an oscillating one in cross-flow. A special focus will be placed on the manipulation of the vortex shedding frequency and lock-in phenomenon by a porous actuator.

8:13AM G6.00002 Evaluation of the unstructured lattice Boltzmann method in porous flow simulations. MAREK MISZTAL, RASTIN MATIN, ANIER HERNANDEZ, JOACHIM MATHIESEN, Niels Bohr Inst — Flows in porous media are among the most challenging to simulate using the computational fluid dynamics methods, primarily due to the complex boundaries, often characterized by a very broad distribution of pore sizes. The standard (regular grid based) lattice Boltzmann method with the multi-relaxation time (MRT) collision operator is often used to simulate such flows. However, due to the lack of coupling between the positions of the computational grid nodes and the solid boundaries, the properties of the simulated flow might unrealistically vary with the fluid’s viscosity, depending on the parameters of the MRT operator. This is, for instance, the case with the otherwise popular, single-relaxation time Bhatnagar–Gross–Krook (BGK) collision operator. Our focus has been on the unstructured grid based, finite element variant of the LBM. By using such approach, we can place the computational grid nodes precisely at the solid boundary. Since there is no prior work on the accuracy of this method in simulating porous flows, we perform a thorough permeability study using both BGK and MRT operators at a wide range of viscosities. We benchmark these models on artificial samples with known solutions, and further, we demonstrate the findings of our studies in the porous networks of real rocks.

1Predicting Petrophysical Parameters: A Project Sponsored by HTF and Maersk Oil and Gas
8:26AM G6.00003 A Three-Dimensional Multi-Mesh Lattice Boltzmann Model for Multiphysics Simulations . AMIRREZA HASHEMI, Department of Mechanical Engineering, The University of Akron, Akron, OH 44325, MOHSEN ESHRAGHI, Department of Mechanical Engineering, California State University, Los Angeles, CA 90032, SERGIO FELICELLI, Department of Mechanical Engineering, The University of Akron, Akron, OH 44325 — The lattice Boltzmann method (LBM) is known as an attractive computational method for modeling fluid flow and, more recently, transport phenomena. As any numerical method, the computational cost of LBM simulations depends on the density of the computational grids. The cost of simulations can become enormous when multiple equations are solved in three dimensions. In this work, the development of a multi-block multi-grid LBM model is discussed for three-dimensional (3D) multiphysics simulations. In a system of multiple coupled equations with different length scales, a multi-block mesh with different grids for each model would enhance the computational efficiency and stability of the model. Embedded-type grids facilitate the transfer of information between lattices while allowing larger time steps. In addition, a non-uniform mesh is considered within each mode that allows mesh refinement within each physical model when required. The multi-mesh method was developed to solve for transport phenomena including fluid flow, mass and heat transfer. The huge memory demands of LBM simulations in 3D was significantly reduced using this scheme. Moreover, by reducing the number of lattice points, cost communication in parallel processing was largely decreased.

8:39AM G6.00004 Effects of viscoelasticity on droplet dynamics and break-up in microchannels: a Lattice Boltzmann study1. ANUPAM GUPTA, Department of Physics and INFN, University of “Tor Vergata,” Via della Ricerca Scientifica 1, 00133 Rome, Italy — The effects of viscoelasticity on the dynamics and break-up of liquid threads in microfluidic devices, i.e., T-junctions & Cross-Junction, are investigated using numerical simulations of dilute polymeric solutions for a wide range of Capillary numbers (Ca), i.e., changing the balance between the viscous forces and the surface tension at the interface. A Navier-Stokes (NS) description of the solvent based on the lattice Boltzmann models (LBM) is here coupled to constitutive equations for finite extensible non-linear elastic dumbbells with the closure proposed by Peterlin (FENE-P model). The various model parameters of the FENE-P constitutive equations, including the polymer relaxation time and the finite extensibility parameter, are changed to provide quantitative details on how the dynamics and break-up properties are affected by viscoelasticity.

1European Research Council under the Europeans Communitys Seventh Framework Programme (FP7/2007-2013)/ERC Grant Agreement N. 297004.

8:52AM G6.00005 ANFIS modeling for prediction of particle motions in fluid flows . ARMAN SAFDARI, KYUNG CHUN KIM, Pusan Natl Univ — Accurate dynamic analysis of parcel of solid particles driven in fluid flow system is of interest for many natural and industrial applications such as sedimentation process, study of cloud particles in atmosphere, etc. In this paper, numerical modeling of solid particles in incompressible flow using Eulerian-Lagrangian approach is carried out to investigate the dynamic behavior of particles in different flow conditions; channel and cavity flow. Although modern computers have been well developed, the high computational time and costs for this kind of problems are still demanding. The Lattice Boltzmann Method (LBM) is used to simulate fluid flows and combined with the Lagrangian approach to predict the motion of particles in the range of masses. Some particles are selected, and subjected to Adaptive-network-based fuzzy inference system (ANFIS) to predict the trajectory of moving solid particles. Using a hybrid learning procedure from computational particle movement, the ANFIS can construct an input-output mapping based on fuzzy if-then rules and stipulated computational fluid dynamics prediction pairs. The obtained results from ANFIS algorithm is validated and compared with the set of benchmark data provided based on point-like approach coupled with the LBM method.

9:05AM G6.00006 Reduction of the temperature jump in the immersed boundary-thermal lattice Boltzmann method . TAKESHI SETA, University of Toyama, KOSUKE HAYASHI, AKIO TOMIYAMA, Kobe University — We analytically and numerically investigate the boundary errors computed by the immersed boundary-thermal lattice Boltzmann method (IB-TLBM) with the two-relaxation-time (TRT) collision operator. In the linear collision operator of the TRT, we decompose the distribution function into symmetric and antisymmetric components and define the relaxation parameters for each part. We derive the theoretical relation between the relaxation parameters for the symmetric and antisymmetric parts of the distribution function so as to eliminate the temperature jump. The simple TRT collision operator succeeds in reducing the temperature jump occurring at the high relaxation time in the IB-TLBM calculation. The porous plate problem numerically and analytically demonstrate that the velocity squared terms should be neglected in the equilibrium distribution function in order to eliminate the effect of the advection velocity on the temperature jump in the IB-TLBM. The passive scalar model without the velocity squared terms more accurately calculates the incompressible temperature equation in the IB-TLBM, compared to the double distribution model, which is based on the relation of the distribution function $g_3 = (\epsilon_3 - u)^2 f_3/2$. We apply the passive scalar model without the velocity squared terms to the simulation of the natural convection between a hot circular cylinder and a cold square enclosure. The proposed method adequately sets the boundary values and provides reasonable average Nusselt numbers and maximum absolute values of the stream function.

9:18AM G6.00007 Immersed boundary method implemented in lattice Boltzmann GPU code . BRIAN DEVINCENTIS, KEVIN SMITH, JOHN THOMAS, Johns Hopkins University Applied Physics Laboratory — Lattice Boltzmann is well suited to efficiently utilize the rapidly increasing compute power of GPUs to simulate viscous incompressible flows. Fluid-structure interaction with solids of arbitrarily complex geometry can be modeled in this framework with the immersed boundary method (IBM). In IBM a solid is modeled by its surface which applies a force at the neighboring lattice points. The majority of published IBMs require solving a linear system in order to satisfy the no-slip condition. However, the method presented by Wang et. al. (2014) is unique in that it produces equally accurate results without solving a linear system. Furthermore, the algorithm can be applied in a parallel manner over the immersed boundary and is, therefore, well suited for GPUs. Here, a 2D and 3D version of their algorithm is implemented in Sailfish CFD, a GPU-based open source lattice Boltzmann code. One issue unaddressed by most published work is how to correct force and torque calculated from IBM for translating and rotating solids. These corrections are necessary because the fluid inside the solid affects its inertia in a non-trivial manner. Therefore, this implementation uses the Lagrangian points approximation correction shown by Suzuki and Inamuro (2011) to be accurate.

9:31AM G6.00008 Different Scalable Implementations of Collision and Streaming for Optimal Computational Performance of Lattice Boltzmann Simulations . NICHOLAS GENEVA, LIAN-PING WANG, University of Delaware — In the past 25 years, the mesoscopic lattice Boltzmann method (LBM) has become an increasingly popular approach to simulate incompressible flows including turbulent flows. While LBM solves more solution variables compared to the conventional CFD approach based on the macroscopic Navier-Stokes equation, it also offers opportunities for more efficient parallelization. In this talk we will describe several different algorithms that have been developed over the past 10 plus years, which can be used to represent the two core steps of LBM, collision and streaming, more effectively than standard approaches. The application of these algorithms spans LBM simulations ranging from basic channel to particle laden flows. We will cover the essential detail on the implementation of each algorithm for simple 2D flows, to the challenges one faces when using a given algorithm for more complex simulations. The key is to explore the best use of data structure and cache memory. Two basic data structures will be discussed and the importance of effective data storage to maximize a CPU’s cache will be addressed. The performance of a 3D turbulent channel flow simulation using these different algorithms and data structures will be compared along with important hardware related issues.
concentrated emulsions (Goyon et al., Nature, vol. 454, 2008, pp. 8487), have parallels in the behaviour of other soft glassy ma
features, not previously reported for a confined foam, lend further support to the idea that cooperativity mechanisms, originally invoked for
the distribution of the orientation of plastic events show that there is a non-trivial correlation with the underlying local shear strain. These
features are represented as volumes bounded by surfaces. Our approach conserves mass and does not require the use of CVs that span domain boundaries. Computational efficiency is increased by use of dynamic mesh optimization. We demonstrate that the approach, amongst other features, accurately preserves sharp saturation changes associated with high aspect ratio geologic domains, allowing efficient simulation of flow in highly heterogeneous models. Moreover, accurate solutions are obtained at lower cost than an equivalent fine, fixed mesh and conventional CVFE methods. The use of implicit time integration allows the method to efficiently converge using highly anisotropic meshes without having to reduce the time-step. The work is significant for two key reasons. First, it resolves a long-standing problem associated with the use of classical CVFE methods. Second, it reduces computational cost/increases solution accuracy through the use of dynamic mesh optimization and time-stepping with large Courant number.

1Funding for Dr P. Salinas from ExxonMobil is gratefully acknowledged

JAMES PERCIVAL, ALEXANDER ADAM, ZHIHUA XIE, CHRISTOPHER PAIN, MATTHEW JACKSON, Imperial College London — We present a methodology to concurrently couple particle-based methods via a domain decomposition (DD) technique for simulating viscous flows. In particular, we select two resolutions of the smoothed particle hydrodynamics (SPH) method as demonstration. Within the DD framework, a simulation domain is decomposed into two (or more) overlapping sub-domains, each of which has an individual particle scale determined by the local flow physics. Consistency of the two sub-domains is achieved in the overlap region by matching the two independent simulations based on Lagrangian interpolation of state variables and fluxes. The domain decomposition based SPH method (DD-SPH) employs different spatial and temporal resolutions, and hence, each sub-domain has its own smoothing length and time step. As a consequence, particle refinement and de-refinement are performed asynchronously according to individual time advancement of each sub-domain. The proposed strategy avoids SPH force interactions between different resolutions on purpose, so that coupling, in principle, can go beyond SPH - SPH, and may allow SPH to be coupled with other mesoscopic or microscopic particle methods. The DD-SPH method is validated first for a transient Couette flow, where simulation results base

1This material is based upon work supported by the National Science Foundation Graduate Research Fellowship under Grant No. DGE-1148903

8:13AM G7.00002 Collocated approximations on unstructured grids: a comparison between General Finite Differences (GFD), Moving Least Squares (MLS), and Smoothed Particle Hydrodynamics (SPH)1. YAROSLAV VASYLIV, ALEXANDER ALEXEEV, Georgia Institute of Technology — In the meshfree family of methods, partial differential equations are solved on unstructured grids where a search radius establishes an implicit nodal connectivity used to determine whether to include or exclude neighboring nodes in the constructed approximation. Smoothed Particle Hydrodynamics (SPH) is widely attributed to be the earliest of the meshfree methods dating back to an astrophysics paper published in 1977 by Gingold and Monaghan. However, beating them by five years was Jensen when he published Finite Differences for Arbitrary Grids (FIDAG) in 1972. Ultimately this work and others were generalized by Liszka and Orkisz in 1979 as a weighted least squares formulation solving for the Taylor coefficients and is now commonly known as General Finite Differences (GFD). Shortly after in 1981, Lancaster and Salkauskas introduced the Moving Least Squares (MLS) approximation for surface reconstruction using a weighted least squares formulation where the unknown coefficients are treated as functions varying from node to node in the support domain. Here we examine important differences, similarities and limitations of each method by solving the 2D Poisson equation on unstructured grids.

1US DOE Collaboratory on Mathematics for Mesoscopic Modeling of Materials (CM4)
8:39AM G7.00004 The effect of density estimation on the conservativeness in Smoothed Particle Hydrodynamics. PRANAV SURESH, S.S.PRASANNA KUMAR, B.S.V. PATNAIK, Indian Inst of Tech-Madras — Smoothed Particle Hydrodynamics (SPH) is a popular mesh-free method for solving a wide range of problems that involve interfaces. In SPH, the Lagrangian nature of the method enables mass conservation to be naturally satisfied. However, satisfying the conservation of momentum and energy is indeed formulation dependent. One major aspect of ensuring conservativeness comes from the density estimation. There are two distinct types of density estimation approaches, namely continuity density approach and summation density approach. Both approaches are indeed popular with single and multi-phase flow communities. In the present study, we assess the role of density estimation on the conservativeness, using several representative numerical examples. In particular, we have simulated the Rayleigh-Taylor instability problem, Non-Boussinesq lock exchange problem, bubble rise in water column etc. Although for shorter time scales of simulation, both methods have similar conservativeness properties, we observe that for longer time scales, summation-density approach is better. For free surface detection and normal vector computations, efficient computational procedures have been devised.

8:52AM G7.00005 Advection Scheme for Phase-changing Porous Media Flow of Fluids with Large Density Ratio 1. DUAN ZHANG, JUAN PADRINO, Los Alamos National Laboratory — Many flows in a porous media involve phase changes between fluids with a large density ratio. For instance, in the water-steam phase change the density ratio is about 1000. These phase changes can be results of physical changes, or chemical reactions, such as fuel combustion in a porous media. Based on the mass conservation, the velocity ratio between the fluids is of the same order of the density ratio. As the result the controlling Courant number for the time step in a numerical simulation is determined by the high velocity and low density phase, leading to small time steps. In this work we introduce a numerical approximation to increase the time step by taking advantage of the large density ratio. We provide analytical error estimation for this approximate numerical scheme. Numerical examples show that using this approximation about 40-fold speedup can be achieved at the cost of a few percent error.

9:05AM G7.00006 Simulations of Coalescence and Breakup of Interfaces Using a 3D Front-tracking Method 1. JACAI LU, GRETAR TRYGGVASON, University of Notre Dame — Direct Numerical Simulations (DNS) of complex multiphase flows with coalescing and breaking-up of interfaces are conducted using a 3D front-tracking method. Front-tracking method has been successfully used in DNS of turbulent channel bubbly flows and many other multiphase flows, but as the void fraction increases changes in the interface topology, though coalescence and breakup, become more common and have to be accounted for. Topology changes have often been identified as a challenge for front tracking, where the interface is represented using a triangular mesh, but here we present an efficient algorithm to change the topology of triangular elements of interfaces. In the current implementation we have not included any small-scale attractive forces so thin films coalesce either at prescribed times or when their thickness reaches a given value. Simulations of the collisions of two drops and comparisons with experimental results have been used to validate the algorithm but the main applications have been to flow regime transitions in gas-liquid flows in pressure driven channel flows. The evolution of flow, including flow rate, wall shear, projected interface areas, pseudo-turbulence, and the average size of the various flow structures, is examined as the topology of the interface changes through coalescence and breakup.

9:18AM G7.00007 Addressing the Numerical Challenges Associated With Laser-Induced Melt Convection. BRIAN WESTON, Univ of California - Davis, ROBERT NOURGALIEV, Lawrence Livermore National Laboratory, JEAN PIERRE DELPLANQUE, Univ of California - Davis, ANDY ANDERSON, Lawrence Livermore National Laboratory — We present a new robust and efficient numerical framework for simulating multi-material flows with phase change. The work is motivated by laser-induced phase change applications, particularly the selective laser melting (SLM) process in additive manufacturing. Physics-based simulations of the laser melt dynamics requires a fully compressible framework, since incompressible flow solvers are inefficient for stiff systems, arising from laser-induced rapid phase change. In this study, the liquid and solid phases are both modeled with the compressible Navier-Stokes equations. The solid phase has an additional combined variable viscosity and drag force model to suppress the velocity in the solid. Our all-speed Navier-Stokes solver is based on a fully-implicit, high-order reconstructed Discontinuous Galerkin method. A Newton-Krylov based framework is used to solve the resulting set of non-linear equations, enabling robust simulations of the highly stiff compressible Navier-Stokes equations. We demonstrate the method’s capabilities for phase change on several different melting and freezing configurations, including a three-dimensional laser-induced melt convection problem. Future model enhancements will incorporate material evaporation and rapid solidification.

9:31AM G7.00008 An inviscid regularization technique for two-phase flows with shocks and turbulence. KAMRAN MOHSENI, TENG LI, University of Florida — An inviscid regularization technique for the simulation of multiphase flows with sharp interfaces is introduced. This methodology is based on a similar approach successfully used by our group in the past for regularizing single-phase problems with shocks and/or turbulence. The observable divergence theorem is employed to ensure mass conservation, the velocity ratio between the fluids is of the same order of the density ratio. As the result the controlling Courant number for the time step in a numerical simulation is determined by the high velocity and low density phase, leading to small time steps. In this work we introduce a numerical approximation to increase the time step by taking advantage of the large density ratio. We provide analytical error estimation for this approximate numerical scheme. Numerical examples show that using this approximation about 40-fold speedup can be achieved at the cost of a few percent error.

9:44AM G7.00009 Modeling of Two-Phase Immiscible Flow with Moving Contact Lines. MOATAZ ABU ALSAUD, CYPRIEN SOULAINNE, Stanford University, AMIR RIAZ, University of Maryland, College Park, HAMDI TCHELEPI, Stanford University, STANFORD UNIVERSITY COLLABORATION, UNIVERSITY OF MARYLAND, COLLEGE PARK COLLABORATION — A new numerical method based on the implicit interface approach on Cartesian grids is proposed for modeling two-phase immiscible flow with moving contact lines. The reinitialization of the reconstructed interface to obtain signed distance function is extended to include the contact angle boundary condition. The physics of contact line dynamics is implemented using the Cox-Voinov hydrodynamic theory that efficiently captures the effect of the microscopic contact line region. The numerical method is validated through various examples. Parasitic currents are studied in the case of static and constantly advected parabolic interface intersecting the domain boundary with an imposed contact angle. Moving contact line in the viscous dominated regime is studied and verified through comparison with experiments.

1 Work partially supported by LDRD project of LANL.

1 Research supported by DOE (CASL).
9:57AM G7.00010 A low Mach number preconditioned scheme for a two-phase liquid-gas compressible flow model. MARICA PELANTI, ENSTA ParisTech — The simulation of liquid-gas flows such as cavitating flows demands numerical methods efficient for a wide range of Mach number regimes, due to the large and rapid variation of the speed of sound in these two-phase flows. When classical upwind finite volume discretizations for compressible flow models are employed, suitable strategies are needed to overcome the well known difficulty of loss of accuracy encountered at low Mach number by these methods. In this work we present a novel finite volume wave propagation scheme with low Mach number preconditioning for the numerical approximation of a six-equation two-phase liquid-gas compressible flow model with stiff mechanical relaxation. A Turkel-type preconditioner is designed to correct the acoustic fields at low Mach number, by altering the numerical dissipation tensor of the scheme. We present numerical results for two-dimensional liquid-gas nozzle flow tests both for low Mach number regimes and for transonic regimes with shock formation, which show the effectiveness and accuracy of the proposed preconditioned method. In particular, in the low Mach number limit the order of pressure perturbations at the discrete level agrees with the theoretical results for the continuous two-phase flow model.

Monday, November 23, 2015 8:00AM - 10:10AM –
Session G8 Microscale Flows: Oscillatory Fluid Dynamics 108 - Mark Paul, Virginia Tech

8:00AM G8.00001 The Fluid Coupled Dynamics of Small Oscillating Elastic Objects, MARK PAUL, Virginia Tech — There is broad interest in the correlated motion of small elastic objects in a viscous fluid. In many cases, the coupled motion of an array of objects provides more information, and improved resolution, when compared with what is learned from the motion of a single object alone. The physical implications and subtleties of the fluid dynamics caused by oscillating objects for a range of applications will be discussed. The theoretical and numerical ideas needed for the quantitative description of these systems will be presented for external harmonic excitation and for dynamics driven by Brownian motion. An analytical approach for the correlated motion of two elastic objects in fluid will be presented. Each oscillating object is replaced with a two-dimensional cylinder of finite mass that is attached to a spring. The fluid motion is governed by the Navier-Stokes equations and the linear fluctuation-dissipation theorem, analytical expressions will be developed for the motion of the two fluid-coupled cylinders. A comparison of the analytical results with full finite element numerical simulations yields excellent agreement. These ideas can be extended to include different geometries, large arrays, and objects that are tethered together.

8:13AM G8.00002 Effects of Length Scale and Frequency in Oscillatory Flows Induced by Micro- and Nano-mechanical Resonators, KAMIL EKINCI, VURAL KARA, VICTOR YAKHOT, Department of Mechanical Engineering, Boston University — It is challenging to formulate a theory of flows induced by oscillating micro- and nano-mechanical resonators even in simple fluids. The characteristic length and time scales of these devices may lead to surprising deviations from classical fluid dynamics. Here, we study this problem in a near-ideal gas. By changing the gas pressure, we control the mean free path and the relaxation time of the gas. We measure, as a function of pressure, the fluidic dissipation of micro- and nano-mechanical resonators with length scales (sizes) and frequencies that span many orders of magnitude. We show conclusively how a subtle interplay between its length scale (size) and frequency determines the nature of flow around a mechanical resonator, resulting in a low-frequency regime dominated by length scale and a high-frequency regime dominated by frequency. We propose an analytical formula that incorporates both the dimensionless size and frequency and show excellent agreement over the entire parameter space between theory and experiment. Our results are significant for understanding high-frequency and nanoscale fluid-structure interactions as well as for designing improved devices.

8:26AM G8.00003 Measuring the Size and Slip Lengths of Individual Nanoparticles using Suspended Microchannel Resonators, JESSE COLLIS, JOHN SADER, The University of Melbourne, SELIM OLCUM, SCOTT MANALIS, Massachusetts Institute of Technology — Characterizing nanometer-scale particles immersed in liquids using cantilever-based sensing methods can be challenging due to large hydrodynamic damping forces. Suspended Microchannel Resonators (SMRs) differ to conventional cantilever sensors by embedding a microfluidic channel within a vacuum-encased cantilever. These devices can be used as sensitive mass balances for individual nanoparticles flowing through the microfluidic channel; resolution at the attogram scale has been demonstrated recently. We explore a new modality for these devices, where the particle size and surface properties can be characterized. The theoretical framework for this modality is developed using both asymptotic and numerical methods, for which excellent agreement is observed. Comparison of experimental data with Monte-Carlo simulations shows we are able to accurately quantify the slip lengths of these particles.

8:39AM G8.00004 Dynamic Wetting at MHz Vibration: Simple and Complex Liquid Films on an Ultrasonic Actuator, OFER MANOR, GENNADY ALTHSHULER, SAMEER MHA'TRE, LUDMILA ABEZGAUZ, Technion — We excite simple and complex liquid films on ultrasonic actuators that produce a MHz substrate vibration in the form of a surface acoustic wave (SAW). Transfer of momentum from the MHz vibration in the solid substrate to the neighboring liquid translates to convective stresses within the liquid and on the film free boundary. These stresses further invoke various flow mechanisms, also known as acoustic streaming and may support dynamic wetting or dewetting of liquid films. In particular, we use theory and experiment to study the interplay between viscous, capillary, and the vibrational dynamics of liquid films and their internal structure. We employ MHz ultrasonic actuators to study the dynamic wetting and dewetting of free and confined films of oil and water/surfactant solutions on flat surfaces and within microfluidic channels. We further excite films of evaporating solutions and suspensions in order to study the active influence of the solid vibration on the geometry of the molecule and particle patterns that are deposited in this process. We show the physics underlying these different systems may be explained using the convective dynamics the MHz substrate vibration excites in liquid films.

8:52AM G8.00005 Resonance and streaming of armored microbubbles, TAMSIN SPELMAN, University of Cambridge, NICOLAS BERTIN, OLIVIER STEPHEN, PHILIPPE MARMOTTANT, Universite Joseph Fourier, Grenoble, ERIC LAUGA, University of Cambridge — A new experimental technique involves building a hollow capsule which partially encompasses a microbubble, creating an ‘armored microbubble’ with long lifespan. Under acoustic actuation, such bubble produces net streaming flows. In order to theoretically model the induced flow, we first extend classical models of free bubbles to describe the streaming flow around a spherical body for any known axisymmetric shape oscillation. A potential flow model is then employed to determine the resonance modes of the armored microbubble. We finally use a more detailed viscous model to calculate the surface shape oscillations at the experimental driving frequency, and from this we predict the generated streaming flows.
The acoustic vibration of a single bubble results in streaming flows, a non-linear effect that is localized near the bubble, and that is useful to mix and shear fluids in the vicinity only. Here we show that this streaming is much more extended around pair of bubbles because they interact. We perform experiments with flattened microfluidic bubbles, undergoing a volumic vibration mode in response to ultrasound. We observe very long-range recirculating flow around pairs of bubbles. Using a large lattice of these microbubbles, we obtain a unique acoustic bubble pinball driving fluid and particle in complex paths, following elaborate microstreaming vortices. We predict the streamlines to be the consequence of volumic and translational vibration of the bubbles. The translational vibration is the sign of the interaction between bubbles, here mediated by Rayleigh waves on the elastic channel walls. This work, part of the project Bubbleboost, gives a new insight into bubbles efficiency to trigger mixing in laminar flows.

1 European Community’s Seventh Framework Programme (FP7/2007–2013) ERC Grant Agreement Bubbleboost no. 614655

9:31AM G8.00008 Oscillatory Motion of a Bi-Phasic Slug in a Teflon Reactor

MILAD ABOLHASANI, KLAVS JENSEN, Massachusetts Institute of Technology — Bi-phasic physical/chemical processes require transfer of solute/reagent molecules across the interface. Continuous multi-phase flow approaches (using gas as the continuous phase), usually fail in providing sufficient interfacial area for transfer of molecules between the aqueous and organic phases. In continuous segmented flow platforms (with a fluorinated polymer-based reactor), the higher surface tension of the aqueous phase compared to the organic phase of a bi-phasic slug, in combination with the low surface energy of the reactor wall result in a more facile motion of the aqueous phase. Thus, upon applying a pressure gradient across the bi-phasic slug, the aqueous phase of the slug moves through the organic phase and leads the bi-phasic slug, thereby limiting the available interfacial area for the bi-phasic mass transfer only to the semi-spherical interface between the two phases. Disrupting the quasi-equilibrium state of the bi-phasic slug through reversing the pressure gradient across the bi-phasic slug causes the aqueous phase to move back through the organic phase. In this phase, we experimentally investigate the dynamics of periodic alteration of the pressure gradient across a bi-phasic slug, and characterize the resulting enhanced interfacial area on the bi-phasic mass transfer rate. We demonstrate the enhanced mass transfer rate of the oscillatory flow strategy compared to the continuous multi-phase approach using bi-phasic Pd catalyzed carbon-carbon and carbon-nitrogen cross coupling reactions.

1 NSERC Postdoctoral Fellowship, Novartis Center for Continuous Manufacturing

9:44AM G8.00009 Formation of inverse Chladni patterns at macroscale by acoustic streaming on a silicon membrane immersed in a liquid

CÉDRIC POULAIN, CEA, Grenoble, France, GAEL VUILLEMET, Ecole Polytechnique, FABRICE CASSET, CEA, Grenoble, France — High frequency acoustics (in the MHz range) is known to be very efficient to handle micro particles or living cells in microfluidics by taking advantage of the acoustic radiation force. Here, we will show that low frequency (~50kHz) together with use ultra thin silicon plate can give rise to a micro streaming that enables to move particles at will. Indeed, by means of silicon membranes excited in the low ultrasound range, we show that it is possible to form inverse two-dimensional Chladni patterns of micro-beads in liquid. Unlike the well-known effect in a gaseous environment at macroscale, where gravity effects are generally dominant, leading particles towards the nodal regions of displacement, we will show that the micro scale streaming in the vicinity of the plate tends to gather particles in antinodal regions. Moreover, a symmetry breaking effect together with the streaming can trigger a whole rotation of the beads in the fluidic cavity. We demonstrate that it is possible to make the patterns rotate at a well defined angular velocity where beads actually jump from one acoustic trap to another.

9:57AM G8.00010 The acoustic radiation force on a small thermoviscous or thermoelastic particle suspended in a viscous and heat-conducting fluid

JONAS KARLSEN, HENRIK BRUUS, Tech Univ of Denmark — We present a theoretical analysis (arxiv.org/abs/1507.01043) of the acoustic radiation force on a single small particle, either a thermoviscous fluid droplet or a thermoelastic solid particle, suspended in a viscous and heat-conducting fluid. Our analysis places no restrictions on the viscous and thermal boundary layer thicknesses relative to the particle radius, but it assumes the particle to be small in comparison to the acoustic wavelength. This is the limit relevant to scattering of ultrasound waves from sub-micrometer particles. For particle sizes smaller than the boundary layer widths, our theory leads to profound consequences for the acoustic radiation force. For example, for liquid droplets and solid particles suspended in gasses we predict forces orders of magnitude larger than expected from ideal-fluid theory. Moreover, for certain relevant choices of materials, we find a sign change in the acoustic radiation force on different-sized but otherwise identical particles. These findings lead to the concept of a particle-size-dependent acoustophoretic contrast factor, highly relevant to applications in acoustic levitation or separation of micro-particles in gases, as well as to handling of m- and nm-sized particles such as bacteria and virus in lab-on-a-chip systems.

Monday, November 23, 2015 8:00AM - 10:10AM – Session G9 Nanoscale Flows: Basic Flow Physics

8:00AM G9.00001 Continuum Navier-Stokes modelling of water flow past fullerene molecules

J. H. WALTHER, Technical University of Denmark, Denmark, A. POPADIC, National Institute of Chemistry, Slovenia, P. KOUΜOUTSAKΟΣ, ETH Zurich, Switzerland, M. PRAPROTNIK, National Institute of Chemistry, Slovenia — We present continuum simulations of water flow past fullerene molecules. The governing Navier-Stokes equations are complemented with the Navier slip boundary condition with a slip length that is extracted from related molecular dynamics simulations. We find that several quantities of interest as computed by the present model are in good agreement with results from atomistic and atomistic-continuum simulations at a fraction of the computational cost. We simulate the flow past a single fullerene and an array of fullerenes and demonstrate that such macroscale flows can be computed efficiently by continuum flow solvers, allowing for investigations into spatiotemporal scales inaccessible to atomistic simulations.
8:13AM G9.00002 Nanofluidic Brownian Ratchet via atomically-stepped surfaces
AMIR RAHMANI, CARLOS COLOSQUI, State Univ of NY- Stony Brook — Theoretical analysis and fully atomistic molecular dynamics simulations reveal a Brownian ratchet mechanism by which thermal motion can drive the directionnal displacement of liquids confined in micro- or nanoscale channels and pores. The particular systems discussed in this talk consist of two immiscible liquids confined in a slit-like nanochannel with atomically-stepped surfaces. Mean displacement rates reported in molecular dynamics simulations are in close agreement with theoretical predictions, via analytical solution of a Smoluchowski equation for the probability density of the position of the liquid-liquid interface. The direction of the thermally-driven displacement of liquid is determined by the nanostructure surface geometry and thus imbibition or drainage can occur against the direction of action of capillary forces. The studied surface nanostructure with directional asymmetry can control the dynamics of wetting processes such as capillary filling, wicking, and imbibition in porous materials. The proposed physical mechanisms and derived analytical expressions can be applied to design nanofluidic and microfluidic devices for passive handling and separation.

8:26AM G9.00003 Modeling anomalous diffusion of dense fluids in carbon nanotubes
GERALD WANG, NICOLAS HADJICONSTANTINOU, MIT — Molecular diffusive mechanisms exhibited under nanoflconfinement can differ considerably from the Fickian self-diffusion expected in a bulk fluid. We propose a theoretical description of this phenomenon in a nanofluidic system of considerable interest - namely, a dense fluid confined within a carbon nanotube (CNT). We show that the anomalous diffusion reported in the literature is closely related to the fluid layering widely observed in this system and recently theoretically described [Wang and Hadjiconstantinou, Physics of Fluids, 052006, 2015]. In particular, we find that the key to describing the anomalous molecular diffusion (within sufficiently large CNTs) lies in recognizing that the diffusion mechanism is spatially dependent: while fluid in the center of the nanotube (at least three molecular diameters away from the wall) exhibits Fickian diffusion, fluid near the CNT wall can demonstrate non-Fickian diffusive behavior. The previously reported anomalous diffusive behavior can be reproduced, to a good approximation level, by appropriately combining the bulk and near-wall behavior to form a model for the overall diffusion rate within the nanotube. Such models produce results in quantitative agreement with molecular dynamics simulations.

8:39AM G9.00004 Water transport in graphene nano-channels1
ENRIQUE WAGEMANN, ELTON OYARZUA, University of Concepcion, J. H. WALther, Technical University of Denmark — The transport of water in nanopores is of both fundamental and practical interest. Graphene Channels (GCs) are potential building blocks for nanofluidic devices due to their molecularly smooth walls and exceptional mechanical properties. Numerous studies have found a significant flow rate enhancement, defined as the ratio of the computed flow rate to that predicted from the classical Poiseuille model. Moreover, these studies point to the fact that the flow enhancement is a function of channel height and the fluid-wall physical-chemistry. In spite of the intensive research, an explicit relation between the chirality of the graphene walls and the slip length has not been established. In this study, we perform non-equilibrium molecular dynamics simulations of water flow in single- and multi-walled GCs. We examine the influence on the flow rates of dissipating the viscous heat produced by connecting the thermostat to the water molecules, the CNT wall atoms or both of them. From the atomic trajectories, we compute the fluid flow rates in GCs with zig-zag and armchair walls, heights from 1 to 4 nm and different number of graphene layers on the walls. A relation between the chirality, slip length, and flow enhancement is found.

1We aknowledge partial support from Fondecyt project 11130559 and Redoc udec.

8:52AM G9.00005 Flow enhancement of water flow through silica slit pores with graphene-coated walls1
HARVEY ZAMBRANO, ENRIQUE WAGEMANN, ELTON OYARZUA, University of Concepcion, J. H. WALTHER, Technical University of Denmark — Nanofluidic devices such as Lab-On-a-Chip often are designed to transport water solutions through hydrophobic nano-conduits. In these systems with narrow confinement, the viscous forces dominate the flow and as a result, the hydrodynamic friction drag is very high. Moreover, the drag and the amount of energy required for pumping a fluid are directly related. Therefore, it is desirable to explore drag reduction strategies in nanoconfined flows. Liquids are known to slip past non-wetting surfaces. Graphene is a single-atom-thick sheet of carbon atoms arranged in a hexagonal honeycomb lattice, which features a unparalleled combination of high specific surface area, chemical stability, mechanical strength and flexibility. Recently, the wettability of water droplets on multilayer graphene sheets deposited on a silica substrate has been investigated. In this study, we investigate the role of graphene coatings to induce flow enhancement in silica channels. We conduct molecular dynamics simulations of pressurized water flow inside silica channels with and without graphene layers covering the walls. In particular, we compute density and velocity profiles, flow enhancement and slip lengths to understand the drag reduction capabilities of multilayer graphene coatings.

1We aknowledge partial support from Fondecyt project 11130559

9:05AM G9.00006 Experimental Study of Water Transport through Hydrophilic Nanochannels1
MOHAMMAD AMIN ALIBAKHSHI, QUAN XIE, YINXIAO LI, CHUANHUA DUAN, Boston University — In this paper, we investigate one of the fundamental aspects of Nanofluidics, which is the experimental study of water transport through nanoscale hydrophilic conduits. A new method based on spontaneous filling and a novel hybrid nanochannel design is developed to measure the pure mass flow resistance of single nanofluidic channels/tubes. This method does not require any pressure and flow sensors and also does not rely on any theoretical estimations, holding the potential to be standard for nanofluidic flow characterization. We have used this method to measure the pure mass flow resistance of single 2-D hydrophilic silica nanochannels with heights down to 7 nm. Our experimental results quantify the increased mass flow resistance as a function of nanochannel height, showing a 45% increase for a 7nm channel compared with classical hydrodynamics, and suggest that the increased resistance is possibly due to formation of a 7-angstrom-thick stagnant hydration layer on the hydrophilic surfaces. It has been further shown that this method can reliably measure a wide range of pure mass flow resistances of nanoscale conduits, and thus is promising for advancing studies of liquid transport in hydrophobic graphene nanochannels, CNTs, as well as nanoporous media.

1The work is supported by the American Chemical Society Petroleum Research Fund (ACS PRF # 54118-DNI7) and the Faculty Startup Fund (Boston University, USA).
9:18AM G9.00007 Kinetic Limited Water Evaporation in Hydrophilic Nanofluidic Channels

YINXIAO LI, MOHAMMAD AMIN ALIBAKHSHI, QUAN XIE, CHUANHUA DUAN, Department of Mechanical Engineering, Boston University — Capillary evaporation governed by the kinetic theory has remained poorly understood. Here we report experimental studies of the kinetic-limited water capillary evaporation in 2-D hydrophilic nanochannels. A novel hybrid nanochannel design is employed to guarantee sufficient water supply to the liquid/vapor evaporation interface and to enable precise evaporation rate measurements. We study the effects of confinement (16 ~ 105nm), temperature (20 ~ 40 °C), and relative humidity (5% ~ 60%) on the evaporation rate and the evaporation coefficient. A maximum evaporation flux of 21287 micron/s is obtained in 16-nm nanochannels at 40°C and RH=0%, which corresponds to a heat flux of 4804 W/cm^2. The evaporation coefficient is found to be independent on geometrical confinement, but shows a clear dependence on temperature, decreasing from 0.55 at 20°C to 0.5 at 40 °C. These findings have implications for understanding heat and mass transport in nanofluidic devices and porous media, and shed light on further development of evaporation-based technologies for thermal management, membrane purification and lab-on-a-chip devices.

The work is supported by the American Chemical Society Petroleum Research Fund (ACS PRF #: 54118-DN17) and the Faculty Startup Fund (Boston University, USA).

9:31AM G9.00008 Insights from plants: tunable nano-flows induced by drying

OLIVIER VINCENT, ANTOINE ROBIN, ALEXANDRE SZENICER, ABRAHAM STROOCK, Cornell University — Moving fluids through nanofluidic confinements is a difficult process due to high friction with the walls. Pushing fluids to achieve significant (or even measurable) flows requires very large pressures, which can be inconvenient and costly. Inspired by plants, we used evaporation to generate controlled steady-state nano-flows in pores ~ 3 nm in diameter embedded in a silicon-based micro-platform. The capillary negative pressure that drives the flow, on the order of tens to hundreds of MPa in magnitude, develops spontaneously upon drying and can be externally tuned by changing the relative humidity (vapor saturation) outside of the sample. We show that the analysis of the dynamic drying response allows to get precise measurements of the behavior of highly confined liquids and could be used both as tool for the study of nanoscale fluid physics and as a method to handle liquids in a controlled way for lab-on-chip applications. We also discuss flow enhancement possibilities based on ideas from the vascular anatomy of plants.

9:44AM G9.00009 Landau-Squire jet as a versatile probe to measure flow rate through individual nanochannel and nanotubes

ELEONORA SECCHI, SOPHIE MARBACH, ALESSANDRO SIRIA, LYDERIC BOCQUET, Department of Physics, Ecole Normale Superieure, Paris — Over the last decade, nanometric sized channels have been intensively investigated since new model of fluid transport are expected due to the flow confinement at the nanometric scale. Nanoconfined generates new phenomena, such as superfast flows in carbon nanotubes and slippage over smooth surfaces. However, a major challenge of nanofluidics lies in fabricating nanoscale fluidic devices and developing new velocimetry techniques able to measure flow rates down to femtol/s. In this work we report the experimental study of the velocity fields generated by pressure driven flow from glass nanochannel with a diameter ranging from 1µm to 100nm. The flow emerging from these channels can be described by the classical Landau-Squire solution of the Navier-Stokes equation for a point jet. We show that due to the peculiarity of this flow, it can be used as an efficient probe to characterize the permeability of nanochannels. Velocity field is measured experimentally seeding the fluid in the reservoir with 500 nm Polystyrene particles and measuring the velocity with a standard PIV algorithm. Predictions are tested for nanochannels of several dimensions and supported by ionic current measurements. This demonstrates that this technique is a powerful tool to characterize the flow through nanochannels. We finally apply this method to the measurement of the flow emerging from a single carbon nanotube inserted in the nanochannels and present first data of permeability measurement through a single nanotube.


MASANARI HATTORI, Department of Mechanical Engineering and Science, Kyoto University, Kyoto 615-8540, Japan — SHIGERU TAKATA, Department of Aeronautics and Astronautics & Advanced Research Institute of Fluid Science and Engineering, Kyoto University, Kyoto 615-8540, Japan — A systematic asymptotic analysis of the Boltzmann equation shows that the overall behavior of a gas can be described by fluid-dynamic-type equations with the appropriate slip/jump boundary condition when the Knudsen number is small [the generalized slip-flow theory; see Y. Sone, Molecular Gas Dynamics (Birkhäuser, Boston, 2007)]. Near the boundary, a non-fluid-like correction (the Knudsen-layer correction) to the overall solution is required. Although the theory itself has been established up to the second order of the Knudsen number expansion, the data of the correction have been lacking for a long time for the original Boltzmann equation. Recently, we have obtained the data for boundary curvature by assuming the hard-sphere molecules and the diffuse reflection boundary condition. In the present work, the effects of boundary curvature have been clarified in details, thereby completing the required numerical data. A local singularity appears at the level of the velocity distribution function. We have developed the numerical method that handles such a singularity safely.

The present work is supported in part by KAKENHI from JSPS (Nos. 23360083 and 13J01011).

Monday, November 23, 2015 8:00AM - 10:10AM

Session G10 Non-Newtonian Flows: Rheometry and Applications 110 - Vivek Sharma, University of Illinois - Chicago

8:00AM G10.00001 Characterizing Printability of Complex Fluids using Dripping-O nto-Substrate Extensional Rheometry

VIVEK SHARMA, JELENA DINIC, LEIDY N. JIMENEZ, MADELEINE BIAGIOLI, ALEXANDRO ESTRADA, Chemical Engineering, University of Illinois at Chicago — Liquid transfer and drop formation/deposition processes involved in printing, jetting, spraying and coating involve the formation of columnar necks that undergo spontaneous surface tension-driven instabilities, thinning and pinch-off. The thinning and pinch-off dynamics are determined by the relative magnitude of capillary forces, and inertial, viscous stresses for simple (Newtonian and inelastic) fluids. Stream-wise velocity gradients that arise within the thinning columnar neck create an extensional flow field, which induces micro-structural changes within complex fluids, contributing extra elastic stresses that change thinning and pinch-off dynamics. Though it is well-established that the quantitative analysis of neck thinning can provide a measure of extensional rheology response and arguably printability, such measurements require bespoke instrumentation not available, or easily replicated, in most laboratories. In this contribution, we describe a method that relies on understanding, visualization and analysis of capillary-driven self-thinning dynamics in an asymmetric liquid bridge formed by dripping a finite volume of fluid from a nozzle onto a substrate.
8:13AM G10.00002 The importance of flow history in mixed shear and extensional flows. CAROLINE WAGNER, GARETH MCKINLEY, Massachusetts Institute of Technology — Many complex fluid flows of experimental and academic interest exhibit mixed kinematics with regions of shear and elongation. Examples include flows through planar hypersonic contractions in microfluidic devices and through porous media or geometric arrays. Through the introduction of a "flow-type parameter" $\alpha$ which varies between 0 in pure shear and 1 in pure elongation, the local velocity fields of all such mixed flows can be concisely characterized. It is tempting to then consider the local stress field and interpret the local state of stress in a complex fluid in terms of shearing or extensional material functions. However, the material behavior of such fluids exhibit a fading memory of the entire deformation history. We consider a dilute solution of Hookean dumbbells and solve the Oldroyd-B model to obtain analytic expressions for the entire stress field in any arbitrary mixed flow of constant strain rate and flow-type parameter $\alpha$. We then consider a more complex flow for which the shear rate is constant but the flow-type parameter $\alpha$ varies periodically in time (reminiscent of flow through a periodic array or through repeated contractions and expansions). We show that the flow history and kinematic sequencing (in terms of whether the flow was initialized as shearing or extensional) is extremely important in determining the ensuing stress field and rate of dissipated energy in the flow, and can only be ignored in the limit of infinitely slow flow variations.

8:26AM G10.00003 Physical gelation of a microfiber suspension. ANTONIO PERAZZO, JANINE K NUNES, Department of Mechanical and Aerospace Engineering, Princeton University, STEFANO GUIDO, Department of Chemical, Materials and Production Engineering, University of Napoli "Federico II", HOWARD A STONE, Department of Mechanical and Aerospace Engineering, Princeton University — Hydrogels are among the most exploited materials in tissue engineering and there is growing interest in injectable hydrogels, especially as applied to surgical adhesives and bioprinting materials. Here we report a method to produce a hydrogel in a desired location by simply extruding a suspension of high aspect ratio and flexible microfibers from a syringe. The mechanism of gel formation is purely physical and based on irreversible entanglements formed by the microfibers under the action of flow. The single microfibres have been produced and finely tailored by microfluidic methods. Shear rheology has been performed in order to get insights on the entanglements, and results show that the formation of entanglements is related to a shear thickening behavior of the suspension, which in turn depends on shear rate and concentration of fibers. When shearing the suspension, highly non-linear viscoelastic behavior is observed and probed by a highly positive first normal stress difference. We also show the hydrogel swelling behavior and its linear viscoelastic properties as obtained by imposing small oscillatory stress to the material.

8:39AM G10.00004 Extensional Relaxation Times and Pinch-off Dynamics of Dilute Polymer Solutions. JELENA DINIC, YIRAN ZHANG, LEIDY JIMENEZ, VIVEK SHARMA, Chemical Engineering, University of Illinois at Chicago — We show that visualization and analysis of capillary-driven thinning and pinch-off dynamics of the columnar neck in an asymmetric liquid bridge created by dripping-onto-substrate can be used for characterizing the extensional rheology of complex fluids. Using a particular example of dilute, aqueous PEO solutions, we show the measurement of both the extensional relaxation time and extensional viscosity of weakly elastic, polymeric complex fluids with low shear viscosity $\eta < 20$ mPa·s and relatively short relaxation time, $\lambda < 1$ ms. Characterization of elastic effects and extensional relaxation times in these dilute solutions is beyond the range measurable in the standard geometries used in commercially available shear and extensional rheometers (including CaBER, capillary breakup extensional rheometer). As the radius of the neck that connects a sessile drop to a nozzle is detected optically, and the extensional response for viscoelastic fluids is characterized by analyzing their elastocapillary self-thinning, we refer to this technique as optically-detected elastocapillary self-thinning dripping-onto-substrate (ODES-DOS) extensional rheometry.

8:52AM G10.00005 Spray Formation of Herschel–Bulkley Fluids using Impinging Jets1, NEIL RODRIGUES2, JIAN GAO3, JUN CHEN, PAUL E. SOJKA, M.J. Zucrow Laboratories, Purdue University — The impinging jet spray formation of two non-Newtonian, shear-thinning, Herschel–Bulkley fluids was investigated in this work. The water-based gelled solutions used were 1.0 wt.% agar and 1.0 wt.% kappa carrageenan. A rotational rheometer and a capillary viscometer were used to measure the strain-rate dependency of viscosity and the Herschel–Bulkley Extended (HBE) rheological model was used to characterize the shear-thinning behavior. A generalized HBE jet Reynolds number $Re_j, gen$ was used as the primary parameter to characterize the spray formation. A like-for-like impinging jet doublet was used to produce atomization. Shadowgraphs were captured in the plane of the sheet formed by the two jets using a CCD camera with an Nd:YAG laser beam providing the back-illumination. Typical behavior for impinging jet atomization using Newtonian liquids was not generally observed due to the non-Newtonian, viscous properties of the agar and kappa carrageenan gels. Instead, various spray patterns were observed depending on $Re_j, gen$. Spray characteristics of maximum instability wavelength and sheet breakup length were extracted from the shadowgraphs.

1Multi-University Research Initiative Grant Number W911NF-08-1-0171
2Currently: Engineer, Xerox Corporation.
3Currently: Postdoctoral Researcher, Johns Hopkins University.

9:05AM G10.00006 Phase diagram of vorticity response to surface waviness in viscoelastic Couette flow. JACOB PAGE, Imperial College London, TAMER ZAKI, Johns Hopkins University — The response of viscoelastic Couette flow to surface waviness on the lower wall is examined for both Oldroyd-B and FENE-P fluids. The elasticity of the fluid supports vorticity wave propagation along the tensioned streamlines, which results in the formation of a critical layer where the elastic wave speed matches the base velocity. The induced vorticity is quantified using an integral measure of its penetration into the bulk flow. The flow response to the roughness is classified using a phase diagram, analogous to the Newtonian problem (Charru & Hinch, J. Fluid Mech. 2000). The main parameters in the viscoelastic configuration are the ratios of the channel depth and the critical layer height to the surface wavelength. In deep channels, a significant vorticity is generated away from the wall at the critical layer through a kinematic amplification mechanism. For shallow channels the flow response is dictated by elastic effects, and vorticity amplification occurs in a thin boundary layer at the upper wall. Fourier superposition is used to extend the results to localized wall bumps. Unlike the Newtonian fluid where a single vortex forms above the bump, in the viscoelastic flow an additional upstream vortex of opposite sign is induced by the large vorticity at the upper wall.

9:18AM G10.00007 Surface textures and Non-Newtonian fluids for decreased friction. JONATHON SCHUH, RANDY EWOLDT, University of Illinois at Urbana-Champaign — Using surface textures has been shown to decrease friction in fluid-solid sliding contact. A growing trend in the lubrication industry is to add polymers to base oils in order to improve the oil's effectiveness as a lubricant. These polymer additives cause the oil to become a viscoelastic lubricant that will behave differently than a simple Newtonian lubricant. We present an experimental investigation varying both the surface texture depth profile and the viscoelastic lubricant in order to determine their effects on friction reduction. Gap-controlled experiments were performed on a custom tribo-rheometer in order to systematically examine the friction reduction by varying the Reynolds number, Weissenberg number, and Deborah number in bi-directional motion. Cavitation effects are not present so that the normal force is produced solely by the surface textures and the lubricants. We show that the symmetry of the surface textures must be broken in order to produce normal forces above the viscoelastic response, and that an optimal angle of asymmetry $\beta$ exists for decreasing friction with asymmetric surface textures and viscoelastic lubricants.
and geometries that the proposed ansatz performs consistently better than the standard approach in describing the heat transport. Furthermore, with respect to the heat transport, measured in terms of the Nusselt number. We then demonstrate at numerical simulations of different RB setups the generalized energy, i.e. kinetic energy plus temperature variance. We extend the technique so that instead it gives the optimal modes with perturbations to capture the coherent structures in the flow. The traveling wave on Nusselt number around the cylinder is also investigated. We use Dynamically Orthogonal (DO) decomposition with stochastic and axial traveling waves. In this study the physical mechanisms leading to the formation of the traveling wave are characterized. The effect of are investigated. The cylinders with a heat source are confined vertically. The natural convection flow around the cylinders leads to horizontal convective heat transport, measured by the Nusselt number $\text{Nu}$, scales with the Rayleigh number $\text{Ra}$ and Prandtl number $\text{Pr}$ and derive multiple scaling regimes, one of which is the Rossby scaling (H.T. Rossby, Deep Sea Res., 12, 1965) for laminar horizontal convection flows. System heat and fluid flow occurs from a differential heating/cooling of the bottom surface of a fluid layer. In the present work we study how the convective heat transport, measured by the Nusselt number $\text{Nu}$, scales with the Rayleigh number $\text{Ra}$ and Prandtl number $\text{Pr}$ and derive the Rossby number, $\text{Ro}$, and the Prandtl number, $\text{Pr}$.

The work is supported by the Deutsche Forschungsgemeinschaft (DFG) under the grant SH405/4 - Heisenberg fellowship.

1Supported by North Dakota NASA EPSCoR

Monday, November 23, 2015 8:00AM - 9:57AM – Session G11 Rayleigh-Bénard Convection I

8:00AM G11.00001 Traveling Waves in Natural-convection Flow Around an Array of Heated Cylinders, HESSAM BABAEE, FANGFANG XIE, CHRYSSOSTOMOS CHRYSSOSTOMIDIS, Massachusetts Institute of Technology, GEORGE KARNIADAKIS, Brown University — In this numerical study traveling waves formed around an array of heated cylinders are investigated. The cylinders with a heat source are confined vertically. The natural convection flow around the cylinders leads to horizontal and axial traveling waves. In this study the physical mechanisms leading to the formation of the traveling wave are characterized. The effect of traveling wave on Nusselt number around the cylinder is also investigated. We use Dynamically Orthodox (DO) decomposition with stochastic perturbations to capture the coherent structures in the flow.

8:13AM G11.00002 Main Modes of Heat Transport in Rayleigh-Bénard Convection Analyzed by a POD approach, JOHANNES LUELFF, WWU Muenster — Rayleigh-Bénard convection, i.e. the buoyancy-induced movement of a fluid enclosed between two horizontal plates, is the definite setup to study thermal convection. We are interested in the heat transport of the main modes that are found in the convection cell. To this end, we apply the technique of proper orthogonal decomposition (POD) to obtain a set of empirical basis modes from simulation data. Usually the POD method results in modes that are optimal in describing the generalized energy, i.e. kinetic energy plus temperature variance. We extend the technique so that instead it gives the optimal modes with respect to the heat transport, measured in terms of the Nusselt number. We then demonstrate at numerical simulations of different RB setups and geometries that the proposed ansatz performs consistently better than the standard approach in describing the heat transport. Furthermore, the coherent structures that are connected to the biggest heat transport are examined.

8:26AM G11.00003 Prandtl number dependence of heat and mass transfer in horizontal convection1, OLGA SHISHKINA, Max Planck Institute for Dynamics and Self-Organization — In a horizontal convection system heat and fluid flow occurs from a differential heating/cooling of the bottom surface of a fluid layer. In the present work we study how the convective heat transport, measured by the Nusselt number $\text{Nu}$, scales with the Rayleigh number $\text{Ra}$ and Prandtl number $\text{Pr}$ and derive multiple scaling regimes, one of which is the Rossby scaling (H.T. Rossby, Deep Sea Res., 12, 1965) for laminar horizontal convection flows. Our theoretical results are supported by direct numerical simulations for a wide range of $\text{Ra}$ and $\text{Pr}$.

1The work is supported by the Deutsche Forschungsgemeinschaft (DFG) under the grant SH405/4 - Heisenberg fellowship.

8:39AM G11.00004 Global and local statistics in turbulent convection at low Prandtl numbers, JANET SCHEEL, Occidental College, JOERG SCHUMACHER, TU Ilmenau — Very high resolution direct numerical simulations (DNS) of turbulent Rayleigh-Benard Convection (RBC) for low Prandtl numbers which are typical for liquid metals such as mercury/gallium (0.021) or sodium (0.005) will be presented. The scaling of global momentum and heat transport is determined and compared to experimental and theoretical results. We also present mean profiles of root-mean-square velocity and vorticity as well as the thermal and kinetic energy dissipation rates. The velocity boundary layer is found to be much thinner than the thermal boundary layer, and the consequences of this for the heat transport as well as the nature of turbulence in RBC will also be discussed. Finally we investigate the skin friction coefficient and shear Reynolds numbers for these systems. Results will also be compared and contrasted with results from DNS for Prandtl numbers of 0.7 and 6.0 and similar Rayleigh numbers.
8:52AM G11.00005 Turbulent structures in convection from a heated sidewall in a stratified fluid, KEATON BURNS, Massachusetts Institute of Technology, ANDREW WELLS, University of Oxford, GLENN FLIERL, Massachusetts Institute of Technology — We present direct numerical simulations of 2D turbulent convection along a heated vertical wall in a fluid with a stable background stratification. Our model considers a Boussinesq fluid with a constant background temperature gradient in a horizontally bounded and vertically periodic domain. The temperature along one sidewall is increased by a constant amount, driving an upward convective boundary layer, the wall and introducing a potential-rise length scale to the system. We examine the resulting turbulent structures and statistics at and above Reynolds numbers of $10^7$, which lies in the range of well-developed turbulent heat transfer for the unstratified case. We also discuss the applicability of this system as a model of melt water flows alongside icebergs and ice shelves, and the potential emergence of convective layers without double-diffusion in geophysical scale problems, in contrast to the double-diffusive layering in laboratory models.

9:05AM G11.00006 Penetrative internally heated convection in two and three dimensions, DAVID GOLUSKIN, University of Michigan, ERWIN VAN DER POEL, University of Twente — We carry out 2D and 3D direct numerical simulations of penetrative convection in a fluid layer. The convection is driven by uniform internal heating between top and bottom plates of equal temperature. The Prandtl number is varied between 0.1 and 10, and a Rayleigh number based on the heating rate is varied up to $5 \times 10^{10}$. The asymmetry between upward and downward heat transport is greatly affected by spatial dimension. The fraction of internally produced heat escaping across the bottom plate, as opposed to the top one, is 1/2 without flow and initially falls as convection strengthens. As convection becomes very strong, however, this fall continues in 3D but reverses in 2D. The mean fluid temperature is much less sensitive to dimension, growing with the heating rate ($H$) like $H^1/5$ in both 2D and 3D. We draw analogy between the inverse of this fluid temperature and the Nusselt number in ordinary Rayleigh-Bénard convection.

9:18AM G11.00007 Mixed insulating and conducting boundary conditions in Rayleigh-Bénard convection, DENNIS BAKHUIS, RODOLFO OSSILLO MONICO, ERWIN VAN DER POEL, Univ of Twente, ROBERTO VERZICCO, Univ of Rome Tor Vergata, DETLEF LOHSE, Univ of Twente — We report the results of 3D direct numerical simulations of a rectangular doubly periodic Rayleigh-Bénard system. These results are an extension of earlier 2D work by Ripesi et al. (Journal of Fluid Mechanics 742, 636, 2014). The Rayleigh number is between $10^3$ and $10^5$ and the Prandtl number is set to unity. The bottom plate is homogeneously heated and the cold top plate of this setup has been split into conducting and insulating regions. While keeping both areas equal, we investigate the effect of introducing and varying the parameters of the insulating layer, and the impact on the mean temperature and on the turbulence statistics. For lower Rayleigh numbers, we observe the typical two-cell pattern, with a system consisting of a hot core and a cold outer layer. For higher Rayleigh numbers, we observe a more complex flow structure with multiple cells.

9:31AM G11.00008 High Rayleigh number simulations in a slender laterally periodic domain, ROBERTO VERZICCO, Dept. Industrial Engineering, Universita’ di Roma Tor Vergata, ERWIN VAN DER POEL, DETLEF LOHSE, Department of Physics, Mesa+ Institute, and J. M. Burgers Centre for Fluid Dynamics, University of Twente, 7500 AE Enschede, The Netherlands — The results of three-dimensional DNS simulations of Rayleigh-Bénard convection with Ra up to $10^{14.5}$ in a laterally periodic geometry with progressively decreasing aspect-ratios are presented. We show global quantities such as the heat transport as well as local time-averages and vertical profiles. It is observed that the heat transport for laterally unconfined geometries can be computed at relatively small aspect-ratios whose value decreases with Rayleigh number. This is beneficial in terms of computational cost, as the total simulated domain gets smaller. The boundary layers profiles are studied and movies of horizontal cross-section of the bulk and the boundary layer are shown.

9:44AM G11.00009 Thermal boundary layer profiles in turbulent Rayleigh-Bénard convection, PENCER TONG, YIN WANG, Department of Physics, Hong Kong University of Science and Technology*, XIAOZHOU HE, Max Planck Institute for Dynamics and Self Organization — We have studied the mean temperature boundary layer profile $T(z)$ and root-mean-square (rms) temperature profile $S(z)$ in turbulent Rayleigh-Bénard convection along the central axis $z$ of a convection cell, which has a thin vertical disk shape with an inner diameter $D = 18$ cm. The temperature measurements were made at fixed Prandtl numbers $Pr = 4.3$ and $Pr = 7.6$ and with the Rayleigh number $Ra$ varied in the range between $1 \times 10^9$ and $1 \times 10^{10}$. The measured $T(z)$ for different values of $Pr$ and $Ra$ can all be well described by the newly proposed boundary layer model [Shishkina et al., Phys. Rev. Lett. 114, 114302 (2015)] with a parameter $c$ varying from 1 to 2.1. The measured rms temperature profile $S(z)$ is found to be a single-peak function with the peak position located at $z \approx 0.88$, where $\delta$ is the boundary layer thickness. The measured $S(z)$ has two separate scaling lengths. Within the boundary layer, it scales with $\delta$ and can be fitted to a power law, $S(z) \sim (z/\delta)^{\alpha}$ with $\alpha \approx 0.6$. Outside the boundary layer, it scales with the cell size $D$ and follows a different power law, $S(z) \sim (z/D)^{\beta}$ with $\beta = 0.42$. *This work was supported by the Research Grants Council of Hong Kong SAR.

Monday, November 23, 2015 8:00AM - 10:10AM — Session G12 Wind Turbines: Wind Farms I

8:00AM G12.00001 Wind Turbine Box - energy fluxes around a characteristic wind turbine, MARC CALAF, GERARD CORTINA, Department of Mechanical Engineering, University of Utah, VARUN SHARMA, School of Architecture, Civil and Environmental Engineering, cole Polytechnique Fedrale de Lausanne — This research project presents a new tool, so called “Wind Turbine Box”, that allows for the direct comparison between the flow around a single wind turbine and the flow around a characteristic wind turbine immersed within a large wind farm. The Wind Turbine Box consists of a limited control volume defined around each wind turbine regardless of whether the turbine is fully isolated or it is plunged within a large wind farm. The Wind Turbine Box tool has been used to compute the energy fluxes around a characteristic wind turbine of a large wind farm to better understand the wake replenishment processes throughout a complete diurnal cycle. The effective loading of the wind farm has been gradually increased, ranging from quasi-isolated wind turbines to a highly packed wind farm. For this purpose, several Large Eddy Simulations have been run, forced with a constant geostrophic wind and a time varying surface temperature extracted from a selected period of the CASES-99 field experiment. Results illustrate the differences in the flow dynamics it evokes around a characteristic wind turbine within a large wind farm and its asymptotic transition to the fully developed wind turbine array boundary layer.
8:13 AM G12.00002 Mean kinetic energy budget of wakes within an array of model wind turbines and porous discs, RAUL BAYOÁN CAL, ELIZABETH CAMP, Portland State University — Wind turbines are often modeled as porous actuator discs within computational studies. In this wind tunnel study, stereo particle image velocimetry (SPIV) is used to characterize the wakes within a 4×3 model wind turbine array and an analogous array of porous disks. The SPIV measurements are performed upstream between −2.9 ≤ x/D ≤ −0.3 and downstream between 0.7 ≤ x/D ≤ 5.6 of the center turbine in the fourth row. To provide context, the similarities and differences in the flow fields as well as the mean and turbulent stresses are found. The primary analysis revolves around the mean kinetic energy budget in the wakes for both cases, model turbines and discs, obtained by the computation of mean kinetic energy, production of turbulence and flux of kinetic energy as these are equivalent to a measure of extracted power.

8:26 AM G12.00003 Proper orthogonal decomposition of wakes within a model wind turbine array and a matched array of porous discs, ELIZABETH CAMP, RAUL BAYOÁN CAL, Portland State University — Porous actuator discs are commonly used in computational simulations to represent wind turbines. Wind tunnel data of a 4×3 model wind turbine array and an array porous discs is obtained via stereo particle image velocimetry. Snapshot Proper Orthogonal Decomposition (POD) is used to compare the characteristics of the wake of the center turbine model in the fourth row with those of a matched porous disk in the same position. In considering the near- and far-wake, an examination of the energy content of the modes, nature of the modes themselves as well as the rate of reconstruction of low dimensional representations of flow quantities is attained.

8:39 AM G12.00004 Effects of Turbine Spacing in Very Large Wind Farms1, SØREN JUHL ANDERSEN, JENS NØRKÆR SØRENSEN, ROBERT FLEMMING MIKKELSEN, Technical University of Denmark — The Dynamic Wake Meandering model (DWM) by Larsen et al. (2007) is considered state of the art for modelling the wake behind a wind turbine. DWM assumes a quasi-steady wake deficit transported as a passive tracer by large atmospheric scales. The approach is also applied to wake interaction within wind farms, although certain aspects of the complex wake interaction are not captured, see Churchill et al. (2014). Recent studies have shown how turbines introduce low frequencies in the wake, which could describe some of the shortcomings. Chamorro et al. (2015) identified three regions of different lengths scales. Iungo et al. (2013) related low frequencies to the hub vortex instability. Okulov et al. (2014) found Strouhal numbers in the far wake stemming from the rotating helical vortex core. Simulations by Andersen et al. (2013) found low frequencies to be inherent in the flow inside an infinite wind farm. LES simulations of large wind farms are performed with full aero-elastic Actuator Lines. The simulations investigate the inherent dynamics inside wind farms in the absence of atmospheric turbulence compared to cases with atmospheric turbulence. Resulting low frequency structures are inherent in wind farms for certain turbine spacings and affect both power production and loads.

1 Funded by Danish Council for Strategic Research (grant 2104-09-067216/DSF), the Nordic Consortium on Optimization and Control of Wind Farms, and EuroTech wind project. The proprietary data for Vestas’ NM80 turbine has been used.

8:52 AM G12.00005 LES studies of wind farms including wide turbine spacings and comparisons with the CWBL engineering model1, RICHARD STEVENS, Johns Hopkins University; Twente University, DENNICE GAYME, Johns Hopkins University, JOHAN MEYERS, University of Leuven, CHARLES MENEVERAEU, Johns Hopkins University — We present results from large eddy simulations (LES) of wind farms consisting of tens to hundreds of turbines with respective streamwise and spanwise spacings approaching 35 and 12 turbine diameters. Even in staggered farms where the distance between consecutive turbines in the flow direction is more than 50 turbine diameters, we observe visible wake effects. In aligned farms, the performance of the turbines in the fully developed regime, where the power output as function of the downstream position becomes constant, is shown to primarily depend on the streamwise distance between consecutive turbine rows. However, for other layouts the power production in the fully developed regime mainly depends on the geometrical mean turbine spacing (inverse turbine density). These findings agree very well with predictions from our recently developed coupled wake boundary layer (CWBL) model, which introduces a two way coupling between the wake (Jensen) and top-down model approaches (Stevens et al. JRSE 7, 023115, 2015). To further validate the CWBL model we apply it to the problem of determining the optimal wind turbine thrust coefficient for power maximization over the entire farm. The CWBL model predictions agree very well with recent LES results (Golt & Meyers, JFM 768, 5-50, 2015).

1 FOM Fellowships for Young Energy Scientists (YES!), NSF (IIA 1243482, the WINDINSPIRE project), ERC (FP7-Ideas, 306471)

9:05 AM G12.00006 Shifted periodic boundary conditions for large-eddy simulation of wind farms1, WIM MUNTERS, KU Leuven, CHARLES MENEVERAEU, Johns Hopkins University, JOHAN MEYERS, KU Leuven — In wall-boundary turbulent flow simulations, periodic boundary conditions combined with insufficiently long domains lead to persistent spanwise locking of large-scale turbulent structures. In the context of wind-farm large-eddy simulations, this effect induces artificial spanwise inhomogeneities in the time-averaged local wind conditions as seen by the wind turbines, leading to spurious differences in power prediction between otherwise equivalent columns of wind turbines in a wind farm (a column is defined here as a set of turbines parallel to the mean flow direction). We propose a shifted periodic boundary condition that eliminates this effect without the need for excessive streamwise domain lengths. Instead of straightforwardly reintroducing the velocity from the outlet plane back at the inlet, as in classic periodic boundary conditions, this plane is first shifted in the spanwise direction by a predefined and constant distance. The method is tested based on a set of direct numerical simulations of a turbulent channel flow, and large-eddy simulations of a high Reynolds number rough-wall half-channel flow. Finally, we apply the method in a precursor simulation, generating inlet conditions for a spatially developing wind-farm boundary layer.

1 WM and JM are supported by the ERC (ActiveWindFarms, grant no: 306471). CM acknowledges support by the NSF (grant IIA-1243482, the WINDINSPIRE project)
The instability is of convective type, such that waves and eventually drops form while the perturbation is moving downwards the inclined plate. For the R-T instability in a falling film between the regime of absolute and convective (A/C) instability using the weighted integral boundary condition, it can result in the formation of droplet and finally droplet detachment if no saturation mechanism arises. This study examines the critical angle of Technology — Liquid films flowing down the underside of inclined plates are subject to film flow instabilities causing a patterned and wavy topology as well as to the classical Rayleigh-Taylor (R-T) instability. The R-T instability results from the denser liquid film being located above a less dense liquid, which is in this case the ambient gaseous phase. Owing to the instability, large amplitude surface deformations form which may result in the formation of droplet and finally droplet detachment if no saturation mechanism arises.

1The work is partially supported by the NSF PIRE Award IIA 1243482. TACC is acknowledged for providing computational time.

Structure Function Scaling Exponent and Intermittency in the Wake of a Wind Turbine Array

Inverse structure functions in the canonical wind turbine array boundary layer

Wind Turbine Experiments at Full Dynamic Similarity

Critical inclination for absolute dripping in falling films subject to Rayleigh-Taylor instability

Monday, November 23, 2015 8:00AM - 10:10AM – Session G13 Free Surface Flow IV: Thin Film Flow

8:00AM G13.00001 Critical inclination for absolute dripping in falling films subject to Rayleigh-Taylor instability

9:18AM G12.00007 Effects of subgrid-scale modeling on wind turbines flows

9:31AM G12.00008 Structure Function Scaling Exponent and Intermittency in the Wake of a Wind Turbine Array

9:44AM G12.00009 Inverse structure functions in the canonical wind turbine array boundary layer

9:57AM G12.00010 Wind Turbine Experiments at Full Dynamic Similarity

1CBET-1034581

1National Science Foundation - CBET-1034581

1supported under NSF grant CBET-1435254 (program manager Gregory Rorrer)
**8:13AM G13.00002 Rayleigh-Plateau instabilities on Drop on Demand Jetting**

Cristina Rodriguez-Rivero, University of Cambridge, Jose Rafael Castrejon-Pita, Queen Mary University of London, Ian Hutchings, University of Cambridge — The fate of liquid filaments is a complex phenomenon; a filament can either collapse into a single drop or break-off into multiple droplets. The final result depends on the liquid viscosity, the shape and the inner dynamics of the filament. In addition, it has been suggested that Rayleigh-Plateau instabilities also play a role in the breakup. In this work we use high-speed imaging and the adequate instrumentation to control the breakup of liquid filaments generated from a drop on demand system. In these experiments, we induce a controlled perturbation, matching the optimal wave number from the Rayleigh-Plateau model, on liquid filaments produced by a droplet generator. Our setup can control the wave number, duration and time of the perturbation. Our results found that both the amplitude and frequency of the Rayleigh-Plateau instability are critical on the break-off behavior. This work was supported by the UK EPSRC (Grant EP/H019813/1) and the Impact Acceleration grant from the University of Cambridge (EP/K03757/1).

**8:26AM G13.00003 Thin liquid film in polymer tubing: dynamics and dewetting in partial wetting condition**

Pascaline Hayoun, Soft Matter Sciences and Engineering, ESPCI ParisTech, France. Albain Letailleur, Composites and Coatings Department, Saint-Gobain Research, France. Jerome Teisseyre, Glass Surface and Interface, Saint-Gobain/CRNS, France. Emilie Verneuil, Francois Lequeux, Etienne Barthele, Soft Matter Sciences and Engineering, ESPCI ParisTech, France — Polymers such as PVC and Silicone are low cost materials widely used in industry to produce tubing for fluid transport. Many applications involving the pumping of liquid films have been reported, which led to unwanted liquid accumulation. This study aims at better understanding contamination mechanisms during intermittent flow in polymer tubing, and at elucidating the relation between flow, wetting and contamination. We experimentally and theoretically investigate, flow regimes as well as dewetting processes at the triple line induced by gravity flow of a vertical liquid slug in a cylindrical geometry. Our results for Newtonian fluids evidence a succession of thick film formation, hydraulic jump creation in the thickness profile, oscillatory regime and destabilization leading to substrate contamination. In order to understand theoretically the flow, one crucial quantity to assess is the film thickness in the inside of the tube. Based on an absorption measurement method, we provide explanations for behaviors and flow regimes observed experimentally.

**8:39AM G13.00004 Stability of Liquid Films on Strings**

Vineet Naik, Ishan Sharma, Viswanathan Shankar, Indian Inst of Tech-Kanpur — The dynamics and stability of liquid films on rigid substrates is a well studied problem with recent studies extending the analysis to flexible substrates. Here, we study the stability of a liquid film on a string. The string is a one-dimensional continuum and we consider it to be linear elastic, isotropic, homogeneous, and flexible. It is assumed that the slope made by the string is small and that the motion is planar and the displacements are in transverse direction. The liquid film is a two-dimensional continuum and we consider it to be a Newtonian fluid with uniform density, viscosity, and surface tension. We consider the cases where the string has an initial horizontal configuration and inclined configuration, including both the finite and infinite cases. We use the lubrication approximation to simplify the governing equations and boundary conditions. The fluid-solid coupling results in a set of two coupled nonlinear partial differential equations in film thickness and string displacement. Subsequently, a linear stability analysis will be carried out and the equations will be solved numerically. The ultimate objective of this study is to understand the behavior of liquid film flow over translating structures such as strings and beams that may or may not be of finite extent.

**8:52AM G13.00005 Inviscid instability of two-fluid free surface flow down an incline**

Sukhendu Ghosh, R Usaha, IIT Madras, India, Ramu Govindarajan, TIFR Centre Interdisciplinary Science, India, Outi Tammisola, The University of Nottingham, UK — Film flow down an incline is known to display an interesting array of instabilities. Such flows are often strongly stratified and this stratigraphic effect (rotor) is subject to gravity. We examine how much of this occurs through an inviscid mechanism, by modeling the velocity profile as piecewise linear. Besides obtaining qualitative agreement between viscous and inviscid results we present several limiting cases. It is interesting to show that a variation in viscosity acts via an inviscid mechanism to stabilize or destabilize the flow.

**9:05AM G13.00006 Gravity-driven liquid flow over a flexible beam**

Hyoungsoo Kim, Princeton University, Peter Howell, University of Oxford, Marinela Popova, University of Toronto, Howard Stone, Princeton University — We study theoretically and experimentally the time dependence of a liquid spreading along a flexible beam. The flow is modeled using a shallow water model and the substrate is modeled as an Euler-Bernoulli elastic beam. We classify the model problem into two cases depending on the maximum beam deflection angle $\phi_{max}$ from the horizontal, i.e. a small deflection ($\phi_{max} < 30^\circ$) and large deflection ($30^\circ < \phi_{max} < 90^\circ$). For a small deflection case, we obtain asymptotic solutions for the liquid propagation speed for the early time and terminal time periods, which for the front position $\sigma(t)$ show power-law behaviors $\sigma(t) \sim t^{4/5}$ and $\sigma(t) \sim t^4$, respectively. The theoretical model also predicts the deflection angle of the beam at the propagating liquid front. We validate the results with experiments, which show good agreement with theory. Furthermore, for large beam deflections, we obtain experimental results demonstrating power-law behaviors, $\sigma(t) \sim t$ and $\phi(t) \sim t^2$ for the early time period.

**9:18AM G13.00007 A model for liquid film in steam turbine**

Amelie Simon, EDF R&D, 78400, Chatou, France / LMFA 69134, Ecully, France, Sergey Meryem Marcelet, EDF R&D, 78400, Chatou, France, Jean-Marc Herard, EDF R&D, 78400, Chatou, France / I2M, UMR CNRS 7353, F13453 Marseille, France, Michel Lancel, LMFA 69134, Ecully, France — Wetness in steam turbines induces losses and erosion. Drops are created due to the fast expansion of the liquid film is subject to high rotational effect (rotor) and/or to negative gravity. Moreover, due to interfacial instabilities, some drops are torn off from the film. The retained approach is an integral formulation of the Navier-Stokes equation (or shallow water equation) with specific terms. The derivation of these equations requires some closure laws for the convection contributions, the Coriolis terms and for terms related to the additional mass coming from the drops. Once chosen, mathematical and mechanical analyses are performed (hyperbolicity, entropy, galilean and rotational invariance). A two-dimensional code has been developed based on finite volume method to simulate numerically this liquid film model for steam turbines.

**9:31AM G13.00008 Healing Capillary Films**

Zhong Zheng, Department of Mechanical and Aerospace Engineering, Princeton University, Marco Fonteles, Departamento de Matematicas, Universidad Autonoma de Madrid, Sangwo Shinh, Howard Stone, Department of Mechanical and Aerospace Engineering, Princeton University — We study the dynamics of a healing viscous thin film driven by surface tension, i.e., the inward spreading of a film to fill a hole in a thin film. A fourth-order nonlinear partial differential equation is obtained to characterize the time evolution of the film thickness and the novel part of study is then to seek self-similar solutions of the second kind. In this way, we are able to obtain a self-similar solution that describes the interface shape, with the scaling exponent determined by solving a nonlinear eigenvalue problem. The self-similar solution is then compared with the full numerical solution of the partial differential equation, and we observe good agreement. Laboratory experiments have also been conducted using various silicone oils, and the time evolution of the front location and the interface shape can be obtained. A comparison between the theoretical predictions and the experimental observations produces good agreement in both the front location and the interface shape.
devices. Also, these effects are also shown to be strongly modulated by the non-ideal permselectivity of the nanochannel.

The role of various electro-convection mechanisms becomes dominant. In particular, electro-osmotic of the second kind and electro-osmotic potential at high voltage to be obtained. Further, we extend the study to microchannels of moderate to large depths where classical, diffusion-limited current and surface-conduction dominant over-limiting currents in a shallow micro-nanochannel device. The extended work is supported by the Faculty Startup Fund (Boston University, USA).

Monday, November 23, 2015 8:00AM - 10:10AM – Session G14 Electrokinetics: Nanochannels, Surface Conduction, Concentration Polarization 202 - Jarrod Schifferbauer, Technion - Israel Institute of Technology

8:00AM G14.00001 Resolving Overlimiting Current Mechanisms in Microchannel-Nanochannel Interface Devices, GILAD YOSSIFON, NETA LEIBOWITZ, URI LIEL, JARROD SCHIFFBAUER, SINWOOK PARK, Technion - Israel Institute of Technology — We present results demonstrating the space charge-mediated transition between classical, diffusion-limited current and surface-conduction dominant over-limiting currents in a shallow micro-nanochannel device. The extended space charge layer develops at the depleted micro-nanochannel entrance at high current and is correlated with a distinctive maximum in the dc resistance. Experimental results for a shallow surface-conduction dominated system are compared with theoretical models, allowing estimates of the effective surface charge at high voltage to be obtained. Further, we extend the study to microchannels of moderate to large depths where the role of various electro-convection mechanisms becomes dominant. In particular, electro-osmotic of the second kind and electro-osmotic instability (EOI) which competes each other at geometrically heterogeneous (e.g. undulated nanoslot) nanoslot devices. Also, these effects are also shown to be strongly modulated by the non-ideal permselectivity of the nanochannel.

8:13AM G14.00002 Chronopotentiometric response of non-ideal ion selective microchannel-nanochannel interface device, NETA LEIBOWITZ, JARROD SCHIFFBAUER, SINWOOK PARK, GILAD YOSSIFON, Technion - Israel Institute of Technology — The passage of an electric current through an ion permselective medium (e.g. membranes and nanochannels) under an applied electric field is characterized by the formation of ionic concentration gradients which result in regions of depleted and enriched ionic concentration at its opposite ends, i.e. concentration polarization. As a result, it can be shown that the chronopotentiometric response of an ideal permselective interface (e.g. permselective membranes) is a monotonic function of the voltage with time regardless of the current intensity. In contrast, a microchannel-nanochannel interface device exhibits a non-monotonic chronopotentiometric response for over-limiting currents. This is shown both numerically and experimentally to result from the non-ideal ion permselectivity of the fabricated nanochannels. This is further supported using experimental visualization techniques that indicate the existence of concentration-polarization within the nanochannel itself and not only within the microchannels.

8:26AM G14.00003 Electrokinetic Transport in Nanochannels Grafted with Polyelectrolyte Brushes with End-Charging, SIDDHARTHA DAS, GUANG CHEN, Univ of Maryland-College Park — Electrokinetic transport in nanochannels grafted with polyelectrolyte (PE) brushes is important for applications such as ion transport, ion manipulation, flow valving, etc. We discuss here a semi-analytical mean field theory approach to study electrokinetic transport in nanochannels grafted with polyelectrolyte brushes with end-charging. The model first probes the thermodynamics and the electrostatics of the PE brushes by appropriately accounting for the entropic (elastic), excluded volume, and electrostatic effects. The resulting knowledge on the electrostatic potential and the PE configuration is next used to obtain the electroosmotic transport. Results demonstrate the role of surface charges (at the end of the PE brushes) in modifying (shrinking or swelling) the brush height. This, in turn, alters the electroosmotic body force and the PE brush layer induced drag force on the fluid flow; therefore, the flow field eventually evolves from a non-trivial interplay of the PE electrostatic, entropic, and excluded volume effects.

8:39AM G14.00004 Ion Transport in 2-D Graphene Nanochannels1, QUAN XIE, ELBERT FOO, CHUANHUA DUAN, Boston Univ — Graphene membranes have recently attracted wide attention due to its great potential in water desalination and selective molecular sieving. Further developments of these membranes, including enhancing their mass transport rate and/or molecular selectivity, rely on the understanding of fundamental transport mechanisms through graphene membranes, which has not been studied experimentally before due to fabrication and measurement difficulties. Herein we report the fabrication of the basic constituent of graphene membranes, i.e. 2-D single graphene nanochannels (GNCs) and the study of ion transport in these channels. A modified bonding technique was developed to form GNCs with well-defined geometry and uniform channel height. Ion transport in such GNCs was studied using DC conductance measurement. Our preliminary results showed that the ion transport in GNCs is still governed by surface charge at low concentrations (10^-6M to 10^-4M). However, GNCs exhibit much higher ionic conductances than silica nanochannels with the same geometries in the surface-charge-governed regime. This conductance enhancement can be attributed to the pre-accumulation of charges on graphene surfaces.

1The work is supported by the Faculty Startup Fund (Boston University, USA).
8:52AM G14.00005 Ion transport and rectification in a charged nanoscale cone 1. FAN YANG, Princeton University, LI ZHANG, QIAN MÃO. Tsinghua University. HOWARD STONE, Princeton University — The possibility of rectification for ion transport in nanofluidic systems offers a potential route for developing a nanofluidic diode that mimics a semiconductor diode or captures some features of a biological ion channel. The rectification phenomenon, in which a solution would be enriched in one ion, results from asymmetric effects in ionic transport that can be realized by discontinuities in surface charge, concentration differences across a pore, or an asymmetric pore shape such as a cone. In this paper, we focus on the latter two effects and seek to capture the rectification effect in simple terms with a non-dimensional model representative of the many systems studied to date. Specifically, we analyze the rectification phenomenon in a charged nanoscale cone with a concentration difference and/or an electrical potential difference across the pore. Based on the Poisson-Nernst-Planck model and the assumption of one-dimensional transport, we derive a model based on two coupled ordinary differential equations to determine significant parameters such as ionic current. We identify several dimensionless parameters that have not been recognized previously and study the influence of the dimensionless parameters on the rectification.

1The authors would like to thank The Center for Combustion Energy (CCE) of Tsinghua University for supporting this project.

9:05AM G14.00006 Geometric effects on electrocapillarity in nanochannels 1. JUNG A. LEE, IN SEOK KANG, Pohang Univ of Sci & Tech — Electrocapillarity phenomenon at the electrified surfaces due to an external voltage or surface charge has been regarded as an efficient tool in micro/nanofluidics. Especially in nanochannels, high surface area with small fluid volume makes the problem more attractive. However, the overlapped electric double layer (EDL) should be carefully considered. In this study, the effects of nanochannel geometry on the electrocapillarity have been studied. Poisson-Boltzmann (PB) equation is solved to get the electric potential distribution of electrolyte solution. Total stress exerted on the gas-liquid interface is expressed by the sum of electric stress from Maxwell stress tensor and the osmotic pressure due to the ionic concentration. The average value of this total stress can be regarded as the measure of electrocapillarity. In the present work, nanochannels with various cross sectional shapes are considered. Using the linearized PB equation, analytic solution for the circular cross-sectional case is obtained and this solution is compared with other cross-sectional cases with the same hydraulic diameter. Several equilateral polygon cases are also analyzed numerically and the results can be unified if they are represented in terms of hydraulic diameter.

1This research was supported by Basic Science Research Program through the national Research Foundation of Korea (NRF) funded by the Ministry of Science, ICT and Future Planning (Grant No. NRF-2013R1A1A2011956).

9:18AM G14.00007 Experimental study on polydisperse nanochannel system with dispersity and voltage variation , LONGNAN LI, DAEJOONG KIM, Department of Mechanical Engineering, Sogang University — Ion exchange membrane (IEM) has great potential for the biological, chemical, energy and desalination applications. Generally, IEM is fabricated by the polymer material and it has non-uniform size of nanopore (nanochannel) matrix structure. We can explain this kind of nanopore non-uniformity by the dispersity of different size of nanopores. The property of IEM strongly depends on nanopore dispersity as the degree of electric double layer (EDL) overlap in the nanopore is depend on nanopore dimension. In this study, polydisperse nanochannel array was fabricated on the silicon wafer to model realistic IEM. To investigate ion transporting behavior through polydisperse nanochannel array, concentration polarization (CP) phenomena is examined. To quantitatively show the CP phenomena in the polydisperse nanochannel system, dispersity of nanochannels and applied voltage are examined as a variable. The experiment result shows that the high dispersity nanochannel system (even with 50% dispersity) still show typical CP behavior that depletion zone at the anodic side of nanochannel. For the voltage-current characteristics of polydisperse nanochannel system, the mononanochannel (50 nm) system and lower dispersity (12.5%) system show typical behavior of CP process.

9:31AM G14.00008 Modelling nanofluidic field amplified sample stacking with inhomogeneous surface charge , CHRISTOPHER MCCALLUM, SUMITA PENNATHUR, Univ of California - Santa Barbara — Nanofluidic technology has exceptional applications as a platform for biological sample preconcentration, which will allow for an effective electronic detection method of low concentration analytes. One such preconcentration method is field amplified sample stacking, a capillary electrophoresis technique that utilizes large concentration differences to generate high electric field gradients, causing the sample of interest to form a narrow, concentrated band. Field amplified sample stacking has been shown to work well at the microscale, with models and experiments confirming expected behavior [1]. However, nanofluidics allows for further concentration enhancement due to focusing of the sample ions toward the channel center by the electric double layer [2]. We have developed a two-dimensional model that can be used for both micro- and nanofluidics, fully accounting for the electric double layer. This model has been used to investigate even more complex physics such as the role of inhomogeneous surface charge.


9:44AM G14.00009 The dominant role of surface conduction in electro-osmotic flows through periodically varying narrow channels , LOTAN LUDAR, EHUD YARIV, Technion — As surface conduction has a non-negligible effect on ion transport in nanochannels, where the tangential Debye-layer current is very high, our analysis suggests that the contribution of surface conduction to the total flow is proportional to the electric double-layer thickness. The model incorporates the complete physics of electro-osmotic flows, including the frictional forces on the channel walls, the electrostatic forces due to the electric field, and the osmotic pressure due to the ionic concentration. This model is solved numerically for a periodic channel geometry, and the results are compared with experimental data. The model predicts that the electro-osmotic flow is dominated by surface conduction, which is consistent with experimental observations. This finding has important implications for the design of nanofluidic devices, as it suggests that surface conduction should be taken into account when designing these devices.

9:57AM G14.00010 Primary, Secondary and Tertiary Vortex Formation in the Ion Concentration Polarization , FRANCISCO J. DIEZ, Rutgers, The state University of New Jersey. SRINIVAS HANASOGE, Georgia Institute of Technology — The experimentally observed formation of multiple micro-vortices in the ion concentration polarization region (ICP) is presented. This is attributed to non-uniform electrokinetic phenomenon effects in the ICP such as the local increase in the electric field due to the change in the electrolyte concentration. Experimentally, the ICP is induced by a patterned nanoporous self-assembling membrane integrated inside a single microchannel. Bottom view images of the channel in the depletion region reveals the to-and-fro motion of micro particles which are a projection of a primary vortex. Side view images of the channel reveal the existence of not one, but a series of three vortices all rotating in the same direction and decreasing in size. We propose a model that predicts the formation of these vortices. It shows how the field amplification together with a 2-Dimensionally varying concentration profile is responsible for these multiple vortices.
8:00AM G15.00001 Taming a flow with a string1, STEPHANE DORBULO, FNRS, GRASP, Departement de Physique B5, NICOLAS VANDEWALLE, BAPTISTE DARBOIS-TEXIER, GRASP, Departement de Physique B5, GRASP TEAM — The speed of a liquid jet out of a pipe is a function of the flow and of the pipe section. Consequently, the trajectory of the liquid jet is governed by the flow and the geometry of the pipe (section and angle with respect to the gravity). We propose to regulate the trajectory of the jet by introducing a flexible wire in the outflow. According to the flow and according to the length of the wire, three regimes can be obtained: (i) no change, (ii) the control of the trajectory, (iii) the guide of the jet direct downwards the vertical. We also show that the wire acts as a free pipe.

1FNRS is thank for financial support.

8:13AM G15.00002 Active flow control for a NACA-0012 profile, H. OUALLI, M. MEKADEM, M. BOUKRIF, S. SAAD, Ecole Militaire Polytechnique, Algiers, Algeria, A. BOUABDALLAH, Université des Sciences et de la Technologie Houari Boumediene, Algiers, Algeria, M. GAD-EL-HAK, Virginia Commonwealth University, Richmond, Virginia, USA — Active flow control is applied on a NACA-0012 profile. The experiments are carried out in a wind tunnel, and flow visualizations are conducted using high-resolution visible-light and infrared cameras. Numerical LES finite-volume code is used to complement the physical experiments. The symmetric wing is clipped into two parts, and those parts extend and retract along the chord according to the same sinusoidal law we optimized last year for a circular/elliptical cylinder (B. Am. Phys. Soc., vol. 59, no. 20, p. 319, 2014). The Reynolds number varies in the range of 500–100,000, which is typical of UAVs and micro-UAVs. The nascent cavity resulting from the oscillatory motion of the profile segments is kept open allowing the passage of fluid between the intrados and extrados. The pulsatile motion is characterized by an amplitude and frequency, and the airfoil’s angle of attack is changed in the range of 0–30 deg. For certain amplitude and frequency, the drag coefficient is increased over the uncontrolled case by a factor of 300. But when the cavity is covered to prevent the flow from passing through the cavity, the drag coefficient becomes negative, and significant thrust is produced. The results are promising to achieve rapid deceleration and acceleration of UAVs.

8:26AM G15.00003 The Flow Field Downstream of a Dynamic Low Aspect Ratio Circular Cylinder: A Parametric Study1, SAMANTHA GILDERSLEEVE, Rensselaer Polytechnic Institute, CLING-MAN DAN, The Boeing Company, MICHAEL AMITAY, Rensselaer Polytechnic Institute — Flow past a static, low aspect ratio cylinder (pin) has been known to generate formation of vortical structures, namely the horseshoe and arch-type vortex. These vortical structures may have substantial effects in controlling flow separation over airfoils. In the present experiments, the flow field associated with a low aspect ratio cylinder as it interacts with a laminar boundary layer under static and dynamic conditions was investigated through a parametric study over a flat plate. As a result of the pin being actuated in the wall-normal direction, the structures formed in the wake of the pin were seen to be a strong function of actuation amplitude, driving frequency, and aspect ratio of the cylinder. The study was conducted at a Reynolds number of 1075, based on the local boundary layer thickness, with a free stream velocity of 10 m/s. SPIV data were collected for two aspect ratios of 0.75 and 1.125, actuation amplitudes of 6.7% and 16.7%, and driving frequencies of 175 Hz and 350 Hz. Results indicate that the presence and interactions between vortical structures are altered in comparison to the static case and suggest increased large-scale mixing when the pin is driven at the shedding frequency (350 Hz).

1Supported by the Boeing Company.

8:39AM G15.00004 Solution to Shape Identification of Steady-state Viscous Flow Fields to Prescribe Flow Velocity Distribution, EIJI KATAMINE, RYOMA KANAI, Department of Mechanical Engineering, Gifu National College of Technology — This paper presents a numerical solution to shape identification problem of steady-state viscous flow fields. In this study, a shape identification problem is formulated for flow velocity distribution prescribed problem, while the total dissipated energy is constrained to less than a desired value, in the viscous flow field. The square error integral between the actual flow velocity distributions and the prescribed flow velocity distributions in the prescribed sub-domains is used as the objective functional. Shape gradient of the shape identification problem is derived theoretically using the Lagrange multiplier method, adjoint variable method, and the formulae of the material derivative. Reshaping is carried out by the traction method proposed as an approach to solving shape optimization problems. The validity of proposed method is confirmed by results of 2D numerical analysis.

8:52AM G15.00005 Limitations of Adjoint-Based Optimization for Separated Flows, J. JAVIER OTERO, ATI SHARMA, University of Southampton, RICHARD SANDBERG, University of Melbourne — Cabin noise is generated by the transmission of turbulent pressure fluctuations through a vibrating panel and can lead to fatigue. In the present study, we model this problem by using DNS to simulate the flow separating off a backward facing step and interacting with a plate downstream of the step. An adjoint formulation of the full compressible Navier-Stokes equations with varying viscosity is used to calculate the optimal control required to minimize the fluid-structure-acoustic interaction with the plate. To achieve noise reduction, a cost function in wavenumber space is chosen to minimize the excitation of the lower structural modes of the structure. To ensure the validity of the cost function, it is essential that the time horizon is long enough to be a representative sample of the statistical behaviour of the flow field. The results from the current study show how this scenario is not always feasible for separated flows, because the chaotic behaviour of turbulence surpasses the ability of adjoint-based methods to compute time-dependent sensitivities of the flow.

9:05AM G15.00006 Input-output dynamic mode decomposition1, JENNIFER ANNONI, MIHAILO JOVANOVIC, JOSEPH NICHOLS, PETER SEILER, Univ of Minn - Minneapolis — The objective of this work is to obtain reduced-order models for fluid flows that can be used for control design. High-fidelity computational fluid dynamic models provide accurate characterizations of complex flow dynamics but are not suitable for control design due to their prohibitive cost and non-salient complexity. A variety of methods, including proper orthogonal decomposition (POD) and dynamic mode decomposition (DMD), can be used to extract the dominant flow structures and obtain reduced-order models. In this presentation, we introduce an extension to DMD that can handle problems with inputs and outputs. The proposed method, termed input-output dynamic mode decomposition (IODMD), utilizes a subspace identification technique to obtain models of low-complexity. We show that, relative to standard DMD, the introduction of the external forcing in IODMD provides robustness with respect to small disturbances and noise. We use the linearized Navier-Stokes equations in a channel flow to demonstrate the utility of the proposed approach and to provide a comparison with standard techniques for obtaining reduced-order dynamical representations.

1NSF Career Grant No. NSFCMMI-1254129
9:18AM G15.00007 A novel control strategy for a Taylor–Couette flow , A. BOUABDALLAH, Université des Sciences et de la Technologie Houari Boumedienne, Algiers, Algeria, H. OUALLI, M. MEKADEM, M. BOUKRIF, S. SAAD, Ecole Militaire Polytechnique, Algiers, Algeria, M. GAD-EL-HAK, Virginia Commonwealth University, Richmond, Virginia, USA — Advancing transition is desired in applications where heat, mass, or momentum transfer needs to be augmented. On the other hand, delaying transition is imperative in crystal growth devices, where all instabilities are to be avoided in order to prevent the appearance of geometrical irregularities in the resulting crystal. The hydrodynamic stability of a viscous flow in a closed, fully filled Taylor–Couette system is considered in the present numerical study. The fluid evolves in an annular cavity between the rotating inner cylinder and the outer fixed one. The base flow is axisymmetric with two counter-rotating vortices each wavelength. The Taylor number varies in the range of 0–50. Numerical simulations are implemented on a finite-volume CFD code. The control strategy involves a pulsatile motion superimposed separately on the inner and outer cylinder’s cross-section, with maximum amplitude of, respectively, 5% and 15% of the radius. The frequency varies in the range of 0–100 Hz. The objective is to localize the transition and to assess the flow’s response to the imposed boundary motions. Substantial advancement of transition is found when the inner cylinder’s cross-section is varied, while this transition is delayed when the outer cylinder’s cross-section is pulsating.

9:31AM G15.00008 Controlling turbulence , JAKOB KHNEN, BJRN HOF, IST Austria — We show that a simple modification of the velocity profile in a pipe can lead to a complete collapse of turbulence and the flow fully relaminarises. The annihilation of turbulence is achieved by a steady manipulation of the streamwise velocity component alone, greatly reducing control efforts. Several different control techniques are presented: one with a local modification of the flow profile by means of a stationary obstacle, one employing a nozzle injecting fluid through a small gap at the pipe wall and one with a moving wall, where a part of the pipe is shifted in the streamwise direction. All control techniques act on the flow such that the streamwise velocity profile becomes more flat and turbulence gradually grows faint and disappears. In a smooth straight pipe the flow remains laminar downstream of the control. Hence a reduction in skin friction by a factor of 8 and more can be accomplished. Stereoscopic PIV-measurements and movies of the development of the flow during relaminarisation are presented.

9:44AM G15.00009 Modification of shear layer characteristics using local periodic heating1 , CHI-AN YEH, PHILLIP MUNDAY, KUNIHIKO TAIRA, Florida State University — Motivated by the recent development of carbon-based thermophobe membranes, we examine their use as a flow control actuator by performing 2D DNS of a compressible subsonic shear layer downstream of a splitter plate for a plate thickness based Reynolds number of 4000. Time varying heat flux boundary condition is utilized as the membrane actuator model on the elliptic nose of the splitter plate. A range of boundary layer thicknesses θ and actuation frequencies are chosen to study the effectiveness of the actuator in modifying the shear layer physics through changing vortex rollup and vortex merging dynamics. For incoming boundary layer with large θ, the heat injection does not shift the rollup frequency when using actuation frequencies between the baseline rollup frequency and its first subharmonic. However, vortex merging is observed to occur earlier downstream. When a positive mean heat is introduced at the same frequency, the early occurrence of the vortex merging is still observed even if the fundamental rollup is delayed due to increased viscosity from the local heating near the nose. For shear layers with small θ, the rollup occurs earlier than the baseline and is locked onto the actuation frequency, but no significant change in the merging is observed.

1This work was supported by the US Army Research Office (Grant W911NF-14-1-0224)

9:57AM G15.00010 Coupled Modification of Body-Wake Flow on an Axisymmetric Moving Platform1 , THOMAS LAMBERT, BOJAN VUKASINOVIC, ARI GLEZER, Georgia Institute of Technology — The unsteady interactions between fluidic actuators and the cross flow over the aft end of a moving bluff body are exploited for modification of the global unsteady aerodynamic loads in wind tunnel experiments using a moving axisymmetric model. The present study focuses on the effects of actuation by an azimuthally-segmented array of four aft-facing synthetic jet modules around the tail end of the model on the coupling between the moving body and its near wake. The model is supported by eight servo-controlled wires, each including a miniature inline force transducer for measurements of the time-resolved tension during the time-dependent six degrees of freedom motion along a prescribed trajectory. In the present investigations the model’s motion is described by parameterized Lissajous rotation (combined pitch and yaw), which is designed to mimic the natural unstable motion of a similar aircraft platform in the absence of roll. Enhancement and suppression of stabilizing aerodynamic loads on the model are each investigated using coupled force and moment measurements and particle image velocimetry in the near wake at reduced frequencies of up to 0.259.

1Supported by ARO
The first author was supported by Texas A&M University Institute for Advanced Study HEEP fellowship.
9:31AM G16.00008 Kelvin-Helmholtz Instability in Compressible Flows and Mixing Inhibition

MONA KARIMI, Department of Mathematics, Texas A&M University, SHARARATH GIRIMAJI, Aerospace Engineering Department, Texas A&M University — It is well-established that the Kelvin-Helmholtz (KH) instability is central to shear flow mixing. Toward understanding the suppression of turbulent mixing under the influence of compressibility, we first examine the modification to KH instability in a planar mixing layer at high speeds. In this presentation, combining the outcomes of the linear stability analysis with the results of the numerical simulation, we establish that the flow domain can be classified into two main regions: the outer regions on the fast and slow sides and dilatational interface layer (DIL) in the middle. Compressibility engenders the formation of a dilatational or acoustic layer at the high-shear interface between two streams of different speeds. Within the DIL, the velocity perturbations become oscillatory. In the incompressible shear layers, the interface experiences steady vortical motion that entrains fluid from both streams leading to familiar KH behavior. In contrast, in the compressible case, the interface motion is oscillatory inhibiting vortex-merging and roll-up, thereby suppressing entrainment that leads to inhibition of the KH instability. Analysis and illustrations of the constituent mechanisms are presented.

9:44AM G16.00009 New approach of a traditional analysis for predicting near-exit jet liquid instabilities

GUILLERMO JARAMILLO1, STEVEN COLLICOTT2, Purdue University — Traditional linear instability theory for round liquid jets requires an exit-plane velocity profile be assumed so as to derive the characteristic growth rates and wavelengths of instabilities. This requires solving an eigenvalue problem for the Rayleigh Equation. In this new approach, a hyperbolic tangent velocity profile is assumed at the exit-plane of a round jet and a comparison is made with a hyperbolic secant profile. Temporal and Spatial Stability Analysis (TSA and SSA respectively) are the employed analytical tools to compare results of predicted most-unstable wavelengths from the given analytical velocity profiles and from previous experimental work. The local relevance of the velocity profile in the near-exit region of a liquid jet and the validity of an inviscid formulation through the Rayleigh equation are discussed as well. A comparison of numerical accuracy is made between two different mathematical approaches for the hyperbolic tangent profile with and without the Ricatti transformation. Reynolds number based on the momentum thickness of the boundary layer at the exit plane non-dimensionalizes the problem and, the Re range, based on measurements by Portillo in 2011, is 185 to 600. Wavelength measurements are taken from Portillo’s experiment.

1School of Mechanical Engineering at Universidad del Valle, supported by a grant from Fulbright and Colciencias. Ph.D. student at the School of Aeronautics and Astronautics Purdue University.
2Professor at the School of Aeronautics and Astronautics. Purdue University

9:57AM G16.00010 Bifurcations in Flow through a Wavy Walled Channel

ZACHARY MILLS, WON SUP SONG, ALEXANDER ALEXEEV, Georgia Institute of Technology — Using computational modeling, we examine the bifurcations that occur in laminar flow of a Newtonian fluid in a channel with sinusoidal walls, driven by a constant pressure gradient. The lattice Boltzmann method was used as our computational model. Our simulations revealed that for a set of geometric parameters the flow in the channel undergoes multiple bifurcations across the range of flow rates investigated. These bifurcations take the form of an initial Hopf bifurcation where the flow transitions from steady to unsteady. The subsequent bifurcations in the flow take the form of additional Hopf, and period-doubling bifurcations. The type and pressure drop at which these bifurcations occur is highly dependent on the geometry of the channel. By performing simulations to determine the critical pressure drops where bifurcations occur and the type for various geometries we developed a flow regime map. The results are important for designing laminar heat/mass exchangers utilizing unsteady flows for enhancing transport processes.

1This work is supported by General Motors Corporation.

Monday, November 23, 2015 8:00AM - 10:10AM — Session G17 Flow Instability: Nonlinear Dynamics and Global Modes

8:00AM G17.00001 A weakly nonlinear model with exact coefficients for the fluttering and spiraling motions of buoyancy-driven bodies

JACQUES MAGNAUDET, CNRS/IMFT, JOEL TCHOUFFAG, DAVID FABRE, IMFT — Gravity/buoyancy-driven bodies moving in a slightly viscous fluid frequently follow fluttering or helical paths. Current models of such systems are largely empirical and fail to predict several of the key features of their evolution, especially close to the onset of path instability. Using a weakly nonlinear expansion of the full set of governing equations, we derive a new generic reduced-order model of this class of phenomena based on a pair of amplitude equations with exact coefficients that drive the evolution of the first pair of unstable modes. We show that the predictions of this model for the style (eg. fluttering or spiraling) and characteristics (eg. frequency and maximum inclination angle) of path oscillations compare well with various recent data for both solid disks and air bubbles.

8:13AM G17.00002 Nonlinear dynamics in eccentric Taylor–Couette–Poiseuille flow

BENOIT PIER, LMFA, CNRS-Université de Lyon, C. P. CAULFIELD, BP Institute & DAMTP, University of Cambridge — The flow in the gap between two parallel but eccentric cylinders and driven by an axial pressure gradient and inner cylinder rotation is characterized by two geometrical parameters (radius ratio and eccentricity) and two dynamic parameters (axial and azimuthal Reynolds numbers). Such a theoretical configuration is a model for the flow between drill string and wellbore in the hydrocarbon drilling industry. The linear convective and absolute instability properties have been systematically derived in a recent study [Leclercq, Pier & Scott, J. Fluid Mech. 2013 and 2014]. Here we address the nonlinear dynamics resulting after saturation of exponentially growing small-amplitude perturbations. By using direct numerical simulations, a range of finite-amplitude states are found and characterized: nonlinear traveling waves (an eccentric counterpart of Taylor vortices, associated with constant hydrodynamic loading on the inner cylinder), modulated nonlinear waves (with time-periodic torque and flow rate) and more irregular states. In the nonlinear regime, the hydrodynamic forces are found to depart significantly from those prevailing for the base flow, even in situations of weak linear instability.
8:26AM G17.00003 Finite-amplitude solutions in rotating Hagen-Poiseuille flow\textsuperscript{1} .

BENOIT PIER, LMFA (CNRS-Université de Lyon), France, ABHISHEK KUMAR, Birla Institute of Technology and Science, Pilani, India, RAMA GOVINDARAJAN, Tata Institute of Fundamental Research, Centre for Interdisciplinary Sciences, India — While the pipe Poiseuille base flow is linearly stable at all Reynolds numbers, a small amount of rotation of the pipe around its axis induces linear instability beyond a low critical Reynolds number $R_e \approx 3$ [Pedley, J. Fluid Mech. 1969]. More recently [Fernandez-Feria and del Pino, Phys. Fluids 2002], this configuration has been shown to become absolutely unstable at Reynolds numbers of the same order of magnitude. Using direct numerical simulations, we investigate finite-amplitude solutions resulting from saturation of exponentially growing small-amplitude initial perturbations. The base flow depends on two dynamical parameters (axial Reynolds number and rotation rate) and the initial perturbation is characterized by its axial wavenumber and its azimuthal mode number. The range of nonlinear waves prevailing in this configuration, the associated nonlinear dispersion relation and the spatial structure of these solutions are systematically obtained by exploring the parameter space.

\textsuperscript{1}Funding from CEFIPRA is gratefully acknowledged

8:39AM G17.00004 Nonlinear interaction of stationary and travelling crossflow modes with a common critical layer .

ALEX AMOS, XUESONG WU, Imperial College London — Laminar-turbulent transition of the three-dimensional boundary layer over a swept wing is caused by amplification of crossflow vortices. A puzzling and interesting experimental observation is that that the free-stream turbulence levels affect the development of stationary crossflow vortices. One possible explanation of this effect is that the travelling modes, which are excited by free-stream turbulence, interact nonlinearly with the stationary modes to affect their development. This interaction between modes is likely to be most effective when they share a critical level, where Rayleigh’s equation becomes singular. We have shown that stationary and travelling modes having a common critical layer do exist. Their mutual nonlinear interactions are studied. The matched asymptotic expansion in conjunction with the multiple-scale method is used to derive the evolution equations for the amplitudes of the modes. The effects of the interactions on the growth of the amplitudes will be discussed, and possible self-interactions and their consequence will be addressed.

8:52AM G17.00005 Non-linear state selection of axially confined viscous liquid jets\textsuperscript{1} .

ALEJANDRO SEVILLA, ALEJANDRO MARTINEZ-CALVO, MARIANO RUBIO-RUBIO, Fluid Mechanics Research Group, Universidad Carlos III de Madrid — Viscous liquid jets injected at a constant flow rate vertically downwards into a gaseous atmosphere become globally unstable when the flow rate becomes smaller than a certain critical value. Previous experiments are in good agreement with a global linear stability analysis based on the leading-order one-dimensional (1D) mass and momentum conservation equations, provided that the full curvature is retained in the computations. However, linear theory cannot predict the large-time dynamics of the jet under globally unstable conditions. To that end, here we report new experiments and numerical simulations of the 1D model, showing that the unstable jet may exhibit two markedly different non-linear states in the long term: either a limit cycle featuring self-sustained oscillations without break-up, or a fully-developed dripping regime emerging after the break-up of the liquid column. A bifurcation analysis demonstrates that the length of the jet is the key parameter that controls the selection of the final state. The dependence of the critical length on the liquid viscosity, the injector radius and the liquid flow rate are also characterized in detail.

\textsuperscript{1}Supported by the Spanish MINECO (proyect DPI2014-50292-C3-1-p)

9:05AM G17.00006 Empirical resolvent mode decomposition .

AARON TOWNE, TIM COLONIUS, OLIVER SCHMIDT, California Institute of Technology — The computation of resolvent modes is a popular method for studying the input/output behavior of fluid dynamical systems. These modes maximize the linear gain between the inputs and outputs of the system as a function of frequency and are computed via a singular decomposition of the linearized operator relating these quantities. Typically, the inputs are meant to represent the nonlinear interactions that are otherwise omitted in linear models. Here, we develop a data-based input/output methodology. The method constructs orthogonal input and output modes from ensembles of flow data that maximize the gains. The essential difference compared to traditional resolvent modes is that the empirical modes are constrained to lie within the subspace spanned by the data. The empirical modes can be shown to be equivalent to either traditional resolvent modes or proper-orthogonal-decomposition modes in appropriate limits. We demonstrate the properties and utility of the method using the complex Ginzburg-Landau equation and LES data from a Mach 0.9 turbulent jet, and compare the empirical modes to traditional resolvent modes in both cases.

9:18AM G17.00007 Linear global modes in a high Reynolds number Mach 0.9 turbulent jet .

OLIVER SCHMIDT, AARON TOWNE, TIM COLONIUS, Caltech — A global linear stability and resolvent analysis of the mean flow from a carefully validated Mach 0.9 turbulent jet large eddy simulation (LES) is conducted. Spatiotemporal Fourier decomposition of the simulation data reveals the presence of large scale coherent structures at small azimuthal wavenumbers. The latter wave packets appear as discrete sets of lightly damped modes in the linear global stability analysis. Their common feature is a spatial separation into an upstream traveling acoustic perturbation in the potential core region, and a Kelvin-Helmholtz-like vortical perturbation which is advected downstream. The least stable branch of discrete modes observed at Strouhal numbers $0.38 < St < 0.42$ exhibits the same acoustic super-directivity as found in the LES and various experimental studies, and hence establishes a direct link between global linear instabilities and low-angle acoustic radiation. Branches at higher frequencies and azimuthal wavenumbers show multi-directive acoustic emission patterns. This observation is of particular interest since high angle, broadband radiation is commonly attributed to stochastic fluctuations of the turbulent jet shear layer.

9:31AM G17.00008 Secondary instability of laminar separation bubbles in the absence of external disturbances\textsuperscript{1} .

DANIEL RODRIGUEZ, Pontificia Universidade Catolica de Rio de Janeiro (PUC-Rio), ELMER GENNARO, UNESP - Universidade Estadual Paulista, LEANDRO SOUZA, Universidade de Sao Paulo — Previous studies demonstrate that the primary instability of laminar separation bubbles (LSB) on a flat-plate in the absence of external forcing is a three-dimensional centrifugal one. This work develops a weakly non-linear expansion of the associated symmetry-breaking bifurcation, showing that it corresponds to a supercritical pitchfork bifurcation. The secondary instability of the fully 3D bifurcated LSB is then investigated by means of the temporal instability of 3D global modes, computed either as solutions of a 3D (Tri-global) eigenvalue problem, or based on a WKBJ approximation and the existence of local regions of absolute instability of the cross-stream planes. Both methodologies recover an amplified global oscillator, originated by the spanwise velocity gradients, that can explain the origin of the unsteadiness observed in numerical simulations of unforced LSBs with peak reversed flows below 15\%, as the results of a secondary instability of the 3D separation bubble.

\textsuperscript{1}Supported by CAPES-Science without borders and FAPESP
9:44AM G17.00009 Stability sensitivity to gravity and base flow density modifications

and the sensitivity theory shows that regions both immediately upstream and immediately downstream of the plate contribute most significantly to the stability sensitivity. On the other hand, stable stratification increases the coherence and persistence of turbulent wakes; on the other hand, it can destabilize vortex structures, such as vortex pairs and rings. We present an application of the sensitivity theory to a stably density-stratified flow around a flat plate at a 90 degree angle of attack. The global mode analysis reveals lightly damped lee wave undulations, other hand, it can destabilize vortex structures, such as vortex pairs and rings. We present an application of the sensitivity theory to a stably density-stratified flow around a flat plate at a 90 degree angle of attack. The global mode analysis reveals lightly damped lee wave undulations.
8:39AM G18.00004 Dynamic measurement of the evolving mechanical properties of thin drying films via induced wrinkling¹. MANUELA NANNIA, GIULIA FERRETTI, OMAR MATAR, JOAO CABRAL, Imperial College London — Surface patterning is important for controlled liquid spreading, adhesion and assembly of smart coatings. Patterns with feature sizes in the 100nm-100μm range can be achieved via wrinkling of bilayers, an inherently inexpensive, scalable and robust method. Conversely, measuring wrinkling of well-defined bilayers and multilayers represents a valuable way to measure mechanical properties of laminate thin films supported by well-defined substrates. We focus on the dynamic measurement of the elastic modulus of micrometer scale layers of ternary solutions during drying and thinning. An atomistic model including conformational change, excluded volume, solvent effects, and van der Waals interactions is used to describe spontaneous wrinkling pattern formation on gel surfaces. The model is based on nonlinear poroelasticity and the flow of liquid is described by a generalisation of Darcy’s law that accounts for the thermodynamics of mixing. A combination of linear stability theory and finite-element simulations is used to explore the surface morphologies in the linear and nonlinear regimes. We show that the model is able to accurately reproduce experimental observations.

²EPSRC Grant number EP/L022176/1

8:52AM G18.00005 Mathematical modelling of swelling-induced surface instabilities in deformable porous media. MATTHEW HENNESSY, ALESSANDRA VITALE, JOAO CABRAL, OMAR MATAR, Imperial College London — The swelling of a deformable porous medium as it absorbs liquid can generate large compressive stresses which, in turn, can induce a rich variety of surface instabilities. When controlled, these instabilities can be used to drive the self-assembly of microscale structures that find practical applications in fields such as surface patterning, imprint lithography, optically-active surfaces, and flexible electronics. Recent experiments by our group have suggested that a swelling-induced instability can occur at a surface of crosslinked polymer gels exposed to a good solvent. In this talk, we present a mathematical model for a swelling-induced instability and use it to describe spontaneous pattern formation on gel surfaces. The model is based on nonlinear poroelasticity and the flow of liquid is described by a generalisation of Darcy’s law that accounts for the thermodynamics of mixing. A combination of linear stability theory and finite-element simulations is used to explore the surface morphologies in the linear and nonlinear regimes. We show that the model is able to accurately reproduce experimental observations.

9:05AM G18.00006 Rupture of thin films of power law fluids on a substrate. VISHRUT GARG, SUMEET THETE, OSMAN BASARAN, Purdue University — Applications in coating, drying, foam stability and drop coalescence require an in-depth understanding of the dynamics of the rupture of thin films. A number of emerging applications in the field involve fluids that exhibit power law (deformation-rate-thinning) rheology. In a power law fluid, viscosity is not constant but is proportional to the deformation rate raised to the n−1 power, where 0<n<1 is the power law exponent (n=1 for a Newtonian fluid). Previous studies by Vanblat and co-workers (2001) and Zhang and Lister (1999) have focused on the rupture of free films and ones supported on a substrate, respectively, for Newtonian fluids. Here, we study the rupture of a thin film of a power law fluid on a substrate under the balance between destabilizing Van der Waals pressure and stabilizing capillary pressure. The power law scaling in time of the film thickness, the lateral length scale, and fluid velocity is determined analytically and confirmed by numerical simulations.

9:18AM G18.00007 Modeling of liquid electrolyte films on non-uniformly charged solid substrates. MAHNPRIT JUTLEY, Southern Methodist Univ, VLADIMIR AJAEV, Southern Methodist University — We consider a thin electrolyte film on a solid substrate characterized by a space-dependent electrical charge density. Using the Debye-Hückel equation to model the electrostatic potential and the Navier-Stokes equations for fluid flow, we consider both steady-state interface shapes and their stability resulting from small perturbations of arbitrary wavelength. Calculations are carried out by two different approaches: Fourier expansion of all terms is used and the corresponding coefficients of the first order correction to the interface shape are found, and, secondly, an evolution equation is obtained within the framework of a lubrication-type model. Stability analysis of the linearized problem is conducted.

9:31AM G18.00008 Experimental free-surface instability growth in gravity-driven film flows of Newtonian and non-Newtonian liquids. JEFFREY OLANDER, ROBERTO CAMASSA, M. GREGORY FOREST, University of North Carolina at Chapel Hill, H. REED OGROSKY, University of Wisconsin-Madison — We present experiments on the growth of free-surface instabilities for Newtonian and viscoelastic film flow in a tube. The liquids used were a Newtonian silicone oil and various concentrations of elastic Boger fluids. The test liquids were injected axisymmetrically into a vertical glass tube at constant volume flow rate and the evolution of the free surface was observed as the film flowed down the tube due to gravity. The range of film thicknesses which exhibited instability growth was smaller for Boger fluids than for silicone oil. Long-wave modeling studies of related problems will be compared with the experimental observations, and our test fluids’ rheology and potential mechanisms for the observed instability growth will be discussed.

9:44AM G18.00009 Long wave evolution of a two-fluid channel flow with surfactant and gravity. DAVID HALPERN, ALEXANDER FRENKEL, University of Alabama — For a horizontal two-fluid channel flow (with top-to-bottom aspect ratio n and viscosity ratio m ) in the presence of surfactants and gravity, with no inertia, the lubrication approximation yields two coupled evolution equations for interface and the insoluble surfactant. Even for arbitrarily strong stabilizing gravity, there is a band of unstable wave numbers for certain (n,m)-ranges. We show that gravity violates the significance of vorticity (Wei 2005) for the surfactant instability. The two types of normal modes are characterised in physical terms, in the spirit of Charru and Hinch (2000). We observe that the role of vorticity hinges on inertia. With no gravity, a small-amplitude saturation of the surfactant instability is possible in contrast to the semi-infinite case studied by A.F. and D.H. (2006). For certain (n,m)-ranges, the interface is governed by a decoupled Kuramoto-Sivashinsky equation, and it provides a source term for the linear convection-diffusion equation of the surfactant. When diffusion is negligible, the surfactant equation has an analytic solution consistent with numerics. The surfactant wave is as chaotic as the interface; however, the ratio of the two waves is just constant. Strongly nonlinear regimes are found at finite n for m < 1.

9:57AM G18.00010 Dynamics of a flowing liquid column with an immiscible reactive micellar interface. ZAHRA NIROOBAKSH, Dept of Materials Science and Engineering, Pennsylvania State University, ANDREW BELMONTE, Dept of Mathematics/Materials Science and Engineering, Pennsylvania State University — We experimentally investigate the instabilities resulting from the reactive formation of a thin layer of micellar material around a flowing liquid column. The material is produced when an aqueous surfactant solution (cetylpyridinium chloride) descends through a reservoir of oleic acid, a room temperature oil which can act as a weak surfactant. A variety of instabilities are observed, including connected and disconnected droplets, a straight cylindrical pipe which undergoes buckling, and various surface wave morphologies on the column. These states appear to be determined by a competition between surface tension and the growth of the interfacial material layer, as a function of imposed flow rate and surfactant concentration. Rheology provides evidence for the structural nature of the oleic/surfactant interaction, in the context of similar observations from other experiments.
We establish a correlation between the outlet state of these solutions and solutions of the columnar (along the pipe center line, vortex breakdown states with a stagnation zone around the pipe center line, and wall-separation states. Then, we first solve the Squire-Long PDE for steady-state flows in a pipe and determine the bifurcation diagram of the various possible flow states as a function of pipe geometry. These include states with a decelerated axial velocity along the pipe center line, an accelerated axial velocity and axial velocity profiles together with a fixed azimuthal vorticity while the outlet flow is characterized by a zero radial velocity state. We of inviscid and incompressible swirling flows in diverging or contracting long circular pipes. The inlet flow is described by the circumferential

Remarkably, we observe twist dynamics capable of conserving total Helicity even in the presence of rapidly changing writhe. Measuring the total Helicity, however, requires additional information about even as those vortices undergo topology changing reconnections. We calculate the curvature invariant and analyze its divergences, which contain the information about phase transitions of the system. The transition points are universal and expressed in terms of ratios of anomaly coefficients.

Analysis of Vortex Line Cutting and Reconnection by a Blade\(^1\). CURTIS SAUNDERS, JEFFREY MARSHALL, The University of Vermont — The essence of vortex reconnection involves the cutting of vortex lines originating from one region and reconnecting to vortex lines originating from another region via the diffusion-regulated annihilation of vorticity. Vortex cutting by a blade is a special case of the more general class of vortex reconnection problems, with an important difference being that vorticity is generated at the reconnection site. In this study, a series of Navier-Stokes simulations of orthogonal vortex cutting by a blade with different values of vortex strength are reported. The three phases of vortex reconnection identified in the literature are found to have counterparts for the vortex cutting problem. However numerous differences between the mechanics of vortex cutting and reconnection within each phase are discussed. In addition, comparisons are made between the temporal changes of the maximum and minimum components of vorticity for vortices of differing strength but still within the vortex cutting regime. The vortex cutting results are also compared with predictions of a simple analytical model that incorporates the key elements of a stretched vorticity field interacting with a solid surface, which is representative of the vortex cutting mechanism near the blade leading edge.

\(^1\)Supported by the National Science Foundation under Grant CBET- 1040236

Hairpin Vortex Regeneration Threshold\(^1\). DANIEL SABATINO, Lafayette College, RIJAN MAHARJAN, Yale University — A free surface water channel is used to study hairpin vortex formation created by fluid injection through a narrow slot into a laminar boundary layer. Particle image velocimetry is used to calculate the circulation of the primary hairpin vortex head which is found to monotonically decrease in strength with downstream distance. When a secondary hairpin vortex is formed upstream of the primary vortex, the circulation strength of the head is comparable to the strength of the primary head at the time of regeneration. However, the legs of the primary vortex strengthen up to the moment the secondary hairpin is generated. Although the peak circulation in the legs is not directly correlated to the strength of the original elongated ring vortex, when the circulation is scaled with the injection momentum ratio it is linearly related to scaled injection time. It is proposed that the injection momentum ratio and nondimensionalized injection time based on the wall normal penetration time can be used to identify threshold conditions which produce a secondary vortex.

A Mathematical Proof of the Vortex Shedding Mechanism\(^1\). MICHAEL BOGHOSIAN, KEVIN CASSEL, Illinois Inst of Tech — A novel mechanism leading to vortex splitting and subsequent shedding that is valid for both inviscid or viscous flows and external, internal, or wall-bounded flows is described. The mechanism, termed the Vortex-Shedding Mechanism (VSM), is simple and intuitive, requiring only two coincident conditions in the flow: (1) the existence of a location with zero momentum and (2) the presence of a net force having a positive divergence. Previous simulations of various flows have demonstrated the VSM numerically. Here, we present a mathematical proof of the VSM which is shown to be both a necessary and sufficient condition for a vortex splitting event in any two-dimensional incompressible flow. The proof includes relating the positive divergence of the net force, condition (2) above, with the second invariant of the velocity gradient tensor, i.e. the Q-criterion. It is shown that the Q-criterion is identical to the determinant of the Hessian matrix for the streamfunction. As a result, the second-partial-derivative test on this Hessian matrix can provide a qualitative description on the behavior of the streamfunction, and thus vortices or recirculation regions, near critical points.

\(^1\)Supported by the National Institute of Diabetes and Digestive and Kidney Diseases of the National Institutes of Health (R01 DK90769)

Swirling flow states in diverging or contracting pipes. ZVI RUSAK, YUXIN ZHANG, Rensselaer Polytechnic Institute, HARRY LI, U Michigan, SHIXIAO WANG, U Auckland, NZ — We study the dynamics of inviscid and incompressible swirling flows in diverging or contracting long circular pipes. The inlet flow is described by the circumferential and axial velocity profiles together with a fixed azimuthal vorticity while the outlet flow is characterized by a zero radial velocity state. We first solve the Squire-Long PDE for steady-state flows in a pipe and determine the bifurcation diagram of the various possible flow states as a function of pipe geometry. These include states with a decelerated axial velocity along the pipe center line, an accelerated axial velocity along the pipe center line, vortex breakdown states with a stagnation zone around the pipe center line, and wall-separation states. Then, we establish a correlation between the outlet state of these solutions and solutions of the columnar (\(\chi\)-independent) Squire-Long ODE. Numerical simulations based on the unsteady stream function-circulation equations shed light on the stability of the various steady states and their domain of attraction in terms of initial conditions. The results show that pipe divergence promotes the appearance of vortex breakdown states while pipe contraction induces the formation of wall-separation states.

Twist Helicity in Classical Vortices\(^1\). MARTIN W. SCHEELE, HRIDESH KEDIA, University of Chicago, DUSTIN KLECKNER, University of California, Merced, WILLIAM T. M. IRVINE, University of Chicago — Recent experimental work has demonstrated that a partial measure of fluid Helicity (the sum of linking and writhe of vortex tubes) is conserved even as those vortices undergo topology changing reconnections. Measuring the total Helicity, however, requires additional information about how the vortex lines are locally twisted inside the vortex core. To bridge this gap, we have developed a novel technique for experimentally measuring twist Helicity. Using this method, we are able to measure the production and eventual decay of twist for a variety of vortex evolutions. Remarkably, we observe twist dynamics capable of conserving total Helicity even in the presence of rapidly changing writhe.

\(^1\)This work was supported by the NSF MRSEC shared facilities at the University of Chicago (DMR-0820054) and an NSF CAREER award (DMR-1351506). W.T.M.I. further acknowledges support from the A.P. Sloan Foundation and the Packard Foundation.
9:18AM G19.00007 Hollow vortices in weakly compressible flows, VIKA$\text{S KRISHNAMURTHY,}$ DARREN CROWDY, Imperial College London — In a two-dimensional, inviscid and steady fluid flow, hollow vortices are bounded regions of constant pressure with non-zero circulation. It is known that for an infinite row of incompressible hollow vortices, analytical solutions for the flow field and the shape of the hollow vortex boundary can be obtained using conformal mapping methods. In this talk, we show how to derive analytical expressions for a weakly compressible hollow vortex row. This is done by introducing a new method based on the Imai-Lamla formula. We will also touch upon how to extend these results to a von-Karman street of hollow vortices.

9:31AM G19.00008 Sadovskii vortex in strain, DANIEL FREILICH, STEFAN LLEWELLYN SMITH, University of California, San Diego — Sadovskii vortices are patches of fluid with uniform vorticity surrounded by a vortex sheet. They were first constructed as models for wakes behind bluff objects. We investigate the Sadovskii vortex in a straining field and examine limiting cases to validate our models for wakes behind bluff objects. We use this as a free-boundary problem, and show that a simple method using the Blot-Savart law quickly gives solutions for stable shapes. When used for the more elongated (stronger straining field) situations, the method also leads to new vortex shapes. In the hollow vortex case, where there is no vortex patch and the circulation is entirely due to the vortex sheet (Llewellyn Smith and Crowdy, J. Fluid Mech. 691, 2012), we use the Birkhoff-Rott equation to calculate the velocity of the fluid on the vortex boundary. The combination of these two methods can then be used to calculate the shape and velocity field of the Sadovskii vortex in strain.

9:44AM G19.00009 Correlating Velocity Information in the Vicinity of Lagrangian Saddle Points to the Spatially and Temporally Resolved Static Pressure Distribution on a Circular Cylinder, MATTHEW ROCKWOOD, MELISSA GREEN, Syracuse University — The locations of Lagrangian saddle points found as the intersections of positive and negative Lagrangian coherent structures (LCS) can be used to determine the location and behavior of von Karman vortices shed in the wake of bluff bodies. Correlating the Lagrangian saddle point locations to physical quantities measurable in real-time is critical to the development of a novel input for closed-loop flow control. As a first step towards finding this correlation, the velocity fluctuations in the vicinity of the Lagrangian saddle point are correlated to the fluctuating static pressure at multiple locations on the cylinder surface to determine the lag time between the two quantities at these locations. This offers insight into the specific location and time of past events on the cylinder that influenced the flow field in the vicinity of the Lagrangian saddle point.

\textsuperscript{1}This work was supported by the Air Force Office of Scientific Research under AFOSR Award No. FA9550-14-1-0210.

9:57AM G19.00010 Helical vortex systems: linear instability analysis and nonlinear dynamics, CAN SELUK, LIMSI — We investigate the stability properties of helical vortices. Instabilities in such vortex systems have mainly been studied theoretically (Widnall 1972, Okulov and Srensen 2007) in an inviscid framework for small core size vortices. The aim of the present study is to generalize these works to the viscous framework for arbitrary core sizes and vorticity profiles. The base flows considered here are helically symmetric: fields are invariant through combined axial translation of distance $\Delta z$ and rotation of angle $\Delta \phi = \Delta z/L$ around the $z$-axis, where $2\pi L$ denotes the helix pitch. We first perform a linear temporal stability analysis of these base flows, using an Arnoldi procedure coupled to two different codes: (i) a linearised version of the helical DNS code HELIX, (ii) another linear code called HELIKZ, which computes the dynamics of arbitrary perturbations in the vicinity of a helically symmetric base flow. These two codes permit the investigation of different types of instability modes: (i) modes having the same helical symmetry as the base flow which generalize the Okulov modes; (ii) modes depending on $z$ as $\exp(ikz)$ which generalize the Widnall modes.

Monday, November 23, 2015 8:00AM - 10:10AM — Session G20 Turbulence: Theory

8:00AM G20.00001 Deriving statistical closure from dynamical optimization, BRUCE TURKINGTON, University of Massachusetts Amherst — Turbulence theorists have traditionally deduced statistical models by generating a hierarchy of moment equations and invoking some closure rules to truncate the hierarchy. In this talk a conceptually different approach to model reduction and statistical closure will be presented, and its implications for coarse-graining fluid turbulence will be indicated. The author has developed this method in the context of nonequilibrium statistical descriptions of Hamiltonian systems with many degrees of freedom. With respect to a chosen parametric statistical model, the lack-of-fit of model paths to the full dynamics is minimized in a time-integrated, mean-squared sense. This optimal closure method is applied to coarse-grain spectrally-truncated inviscid dynamics, including the Burgers-Hopf equation and incompressible two-dimensional flow, using the means and/or variances of low modes as resolved variables. The derived reduced dynamics for these test cases contain (1) scale-dependent dissipation which is not a local eddy viscosity, (2) modified nonlinear interactions between resolved modes, and (3) coupling between the mean and variance of each resolved mode. These predictions are validated against direct numerical simulations of ensembles for the fully resolved dynamics.

8:13AM G20.00002 Backward two-particle dispersion in turbulence: asymptotic behaviors at high Reynolds number, PUI-KUEN YEUNG, D. BUARIA, Georgia Tech, B.L. SAWFORD, Monash Univ, Australia — Backward relative dispersion of fluid elements and diffusing substances or property markers is central to a Lagrangian view of turbulent mixing, but data are not readily available. Recently we have devised a numerical approach based on massively parallel processing of the trajectories of many billions of particle pairs, and have used it to obtain results in simulations of stationary isotropic turbulence up to $4096$ in size and Taylor-scale Reynolds number up to $1000$, with a wide range of initial separations. Backward dispersion is faster than forward, especially at intermediate times after the ballistic range and before long-time diffusive behavior is reached. Richardson scaling is demonstrated for the mean-squared separation, with forward and backward Richardson constants estimated to be 0.55 and 1.5, respectively, which are close to or comparable to other estimates. However, because of persistent dissipation sub-range effects no corresponding scaling was observed for higher order moments. An effort is made to analyze theoretically several key characteristics such as asymmetry in time and exponential growth of third and fourth moments at early times. Related results for marked entities that diffuse relative to the fluid will also be briefly addressed.

\textsuperscript{1}Supported by NSF Grant CBET-1039037 (Fluid Dynamics Program)
8:26AM G20.00003 A Multiscale Morphing Continuum Description for Turbulence, JAMES CHEN, LOUIS WONNELL, Kansas State Univ — Turbulence is a flow physics phenomena involving multiple length scales. The popular Navier–Stokes equations only possess one length/time scale. Therefore, extremely fine mesh is needed for DNS attempting to resolve the small scale motion, which comes with a burden of excessive computational cost. For practical application with complex geometries, the research society rely on RANS and LES, which require turbulence model or subgrid scale (SGS) model for closure problems. Different models not only lead to different results but usually are invalidated on solid physical grounds, such as objectivity and entropy principle. The Morphing Continuum Theory (MCT) is a high-order continuum theory formulated under the framework of thermomechanics for physics phenomena involving microstructure. In this study, a theoretical perspective for the multiscale nature of the Morphing Continuum Theory is connected with the multiscale nature of turbulence physics. The kinematics, balance laws, constitutive equations and a Morphing Continuum description of turbulence are introduced. The equations were numerically implemented for a zero pressure gradient flat plate. The simulations are complete with the laminar, transitional and turbulence cases.

8:39AM G20.00004 Dynamics of the tetrad-based velocity gradient in turbulent flows, HAITAO XU, Center forCombustion Energy, Tsinghua University, Beijing 100084, China and MPI Dynamics & Self-Organization (MPIDS), Goettingen 37077, Germany, ALAIN PUMIR, ENS Lyon, Lyon 69007, France, EBERHARD BODENSCHATZ, MPI Dynamics & Self-Organization (MPIDS), Goettingen 37077, Germany — We investigate the structure and evolution of turbulent flows with the help of the perceived velocity-gradient, determined from four fluid particles initially forming a regular tetrad of size $r_0$. The main feature of the turbulent dynamics can be conveniently captured by a reduced description, in terms of two invariants of the velocity gradient. When $r_0$ is in the inertial range of scales, the evolution of averaged quantities can be parametrized by two dimensionless parameters, which vary slowly with $r_0$. We also characterize the fluctuations around the conditional mean, which represent the dynamics at scales below $r_0$. Using data from both Lagrangian particle tracking experiments and DNS, we show that the behavior qualitatively follows some earlier theoretical prediction, but with interesting new features.

8:52AM G20.00005 Realizable Closure Model for the Reynolds Stress in Rotating Frames, CHARLES PETTY, ANDRE BENARD, Michigan State University — The Reynolds-averaged Navier-Stokes equation for constant property Newtonian fluids is unclosed due to the explicit appearance of the normalized Reynolds stress and the turbulent kinetic energy. A non-negative algebraic mapping of the normalized Reynolds stress into itself provides a practical closure for a wide class of flows. Unlike eddy viscosity closure models, the theory predict the redistribution of the turbulent kinetic energy among the three components of the fluctuating velocity field for statistically stationary spanwise rotating channel flows as well as the Corolis re-direction of turbulent kinetic energy among the three components of the fluctuating velocity field in rotating homogeneous decay. The results partially support the conjecture that the index-of-refraction of the troposphere is anisotropic at all scales.

9:05AM G20.00006 Filtering on the Sphere, HUSSEIN ALJIE, University of Rochester, MATTHEW HECHT, Los Alamos National Laboratory, GEOFFREY VALLIS, University of Exeter — The filtering approach is a natural and valuable framework for analyzing and modeling turbulence, especially within the subject of Large-Eddy Simulation. However, the mathematical development of the approach has been mostly limited to flows in Euclidean (flat) spaces and generalizations to non-Euclidean (curved) manifolds suffer from several shortcomings, such as dependence on the choice of coordinate system, commutation errors, or not preserving volume. Motivated by geophysical flows, we define a new generalized filtering operation on the Sphere which is free from the aforementioned problems. We prove that our filter commutes with spatial derivatives, yielding simple and exact coarse-grained equations for flow on the Sphere. We demonstrate these tools with a-priori tests on flows from high-resolution Ocean simulations.

9:18AM G20.00007 Turbulent Particle Pair Diffusion Using Kinematic Simulations1, NADEEM MALIK, King Fahd University of Petroleum and Minerals — Sweeping errors in Kinematic Simulations (KS) [1] have been shown to be negligible in turbulent flows with extended inertial subranges up to at least $1<k<10^6$ ($k$ is the wavenumber) [2]. The departure from locality scaling observed in the pair diffusivity $K = <\Delta v^2>$ in KS may therefore be a genuine effect, challenging previous assumptions [3] that in turbulence with generalised power-law energy spectra, $E(k) \sim k^{-\gamma}$ for $1<p<3$, locality would lead to, $K \sim \sigma^2_A$, where $\sigma_A = [\gamma <\Delta^2>^{1/2}$, $\Delta$ is the pair separation, $v$ is the pair relative velocity, $<\cdot>$ is the ensemble average, and $\gamma$ = (1+p)/2. For Kolmogorov turbulence this gives, $K \sim \sigma^4_A$. A new analysis, supported by KS [4] confirms that both local and non-local effects govern the pair diffusion process, leading to, $K \sim \sigma^2_A$, where now $\gamma > 3$; for Kolmogorov turbulence, $K \sim \sigma^4_A$. Thus non-local diffusional processes cannot be neglected, and this may have important consequences for the general theory of turbulence. REFERENCES: [1] Fung, J. C. H., Hunt, J. C. R., Malik, N. A., & Perkins, R. J. J. Fluid Mech. 236, 281 (1992). [2] Malik, N. A. Under Review, Physics of Fluids (2015). [3] Richardson, L. F. Proc. Roy. Soc. Lond. A 100, 709 (1926). [4] Malik, N. A. On Turbulent Particle Pair Diffusion. Under review, Physics of Fluids (2015).

1The author acknowledge financial support from SABIC, #SB101011

9:31AM G20.00008 Electrokinetic turbulence in a microchannel at low Reynolds number1, WEI ZHAO, FANG YANG, GUIREN WANG, Department of Mechanical Engineering, University of South Carolina, Columbia — Turbulence is commonly viewed as a type of macroflow phenomena under a sufficiently high Reynolds number (Re). On the other hand, it has been widely perceived in science, engineering and medicine that there is never any turbulence in low Re flow for Newtonian fluids. There is even difficulty to characterize turbulence in microchannels with current available velocimeters, due to the requirement of simultaneously high spatial and temporal resolution. Recently, we generated micro-electrokinetic (EK) turbulence in a microchannel when a pressure driven flow at low Re on the order of unity is electrokinetically forced. We also developed a novel velocimeter, i.e. laser induced fluorescence photobleaching anemometer (LIFPA) that enables us to measure the velocity fluctuations with simultaneously high spatial and temporal resolution. Here we surprisingly observed with LIFPA that the corresponding micro EK turbulence can also have some features of high Re flows, such as Kolmogorov -5/3 spectrum and the exponential tail of probability density function of velocity fluctuation, and the scaling behavior of velocity structure function. This work could provide a new perspective on turbulence.

1The work was supported by NSF under grant no. CAREER CBET-0954977, MRI CBET-1040227.
9:44AM G20.00009 Universality at low Reynolds numbers and the emergence of intermittent behavior in isotropic turbulence. DIEGO DONZIS, Texas A&M University, VICTOR YAKHOT, Boston University, K.R. SREENIVASAN, New York University — Most approaches to understand turbulence have sought universal behavior believed to manifest at high Reynolds numbers ($R_{λ}$). However, recent theory and simulations suggest that universal characteristics, such as the non-trivial anomalous scaling exponents of moments of velocity gradients, emerge even at very low $R_{λ}$, at which no inertial range exists. Furthermore, with decreasing Reynolds numbers, a transition occurs from fully intermittent turbulence to (approximately) Gaussian behavior at an apparently universal critical $R_{λ}$. A potential implication of these observations is that significant information concerning the inertial range (e.g., scaling exponents) is already manifest in the dissipation range at very low $R_{λ}$. Thus, high $R_{λ}$ properties can be studied with well-resolved low-$R_{λ}$ simulations instead of marginally resolved high-Reynolds flows. The focus of this talk is to explore signatures of universality at high-Reynolds numbers in the dissipation range of highly resolved DNS ($k_{max} \eta \sim O(10)$) for $R_{λ}$ up to 90, and decaying simulations close to the critical $R_{λ}$. In addition to statistics of velocity gradients and dissipation we explore evidence of Beltramiization as suggested in past theoretical work.

9:57AM G20.00010 Optimal Energy Dissipation Bounds for 2D and 3D Stress-Driven Shear Flows. GIOVANNI FANTUZZI, ANDREW WYNN, Imperial College London — The background method (Doering & Constantin, 1995) allows the derivation of rigorous bounds on bulk turbulent quantities in a variety of wall-bounded flows as a function of the governing parameters. A classical example is to bound the energy dissipation $\epsilon$ in surface-driven shear flows as a function of the driving force, expressed by the Grashoff number $Gr$. Of particular interest is to compute the best bounds achievable within this framework. However, the variational problem determining the optimal bounds is difficult to solve when the flow is driven by a boundary flux. Tang et al. (2004) first resolved this difficulty by modelling a surface stress with a localised body force. Instead, we propose a novel numerical approach based on Semidefinite Programming that is able to handle fixed-flux boundary conditions directly, and thereby revisit the bounds on $\epsilon$ for surface-stress-driven shear flows. In the 2D case, we find that $\epsilon > Gr^{5/2}$, improving the scaling $\epsilon > Gr^{1/2}$ proven by Hagstrom & Doering (2014). In 3D, we confirm the results of Tang et al., suggesting that a surface stress can be modelled accurately by a body force. Finally, a careful analysis ensures that, in principle, our bounds hold analytically for a fixed $Gr$.

Monday, November 23, 2015 8:00AM - 10:10AM – Session G21 Boundary Layers: Rough or Compliant Walls

8:00AM G21.00001 A numerical investigation of the impact of surface topology on laminar boundary layers. NIKOLAOS BERATLIS, KYLE SQUIRES, Arizona State University, ELIAS BALARAS, The George Washington University — Surface topology, such as dimples or trip wires, has been utilized in the past for passive separation control over bluff bodies. The majority of the work, however, has focused on the indirect effects on the drag and lift forces, while the details of the impact on the boundary layer evolution are not well understood. Here we report a series of DNS of flow over a single row of spherical and hexagonal dimples, as well as, circular grooves. The Reynolds number and the thickness of the incoming laminar boundary layer is carefully controlled. In all cases transition to turbulence downstream of the elements comes with reorientation of the spanwise vorticity into hairpin like vortices. Although qualitatively the transition mechanism amongst different dimples and grooves is similar, important quantitative differences exist: two-dimensional geometries such as the groove, are more stable than three-dimensional geometries. In addition, it was found that the cavity geometry controls the initial thickness of the boundary layer and practically results in a shift of the virtual origin of the turbulent boundary layer.

8:13AM G21.00002 Near surface flow structure over a dimpled surface with blowing1. COLBY BORCHETTA, ALEXANDRE MARTIN, SEAN BAILEY, University of Kentucky — The combined effects of surface roughness with flow injection are of particular interest in understanding the flow over ablative heat-shields, a common form of thermal protection system (TPS) used for atmospheric entry. Stereoscopic, time-resolved particle image velocimetry was used to investigate the near-surface flow over a surface geometry consisting of hexagonal dimples, typical of a TPS. Of particular interest is the modification of the flow structures generated by the dimpled elements caused by flow injection through the surface. Without flow injection, inclined flow structures are generated periodically at the upstream edge of the dimples and convected downstream. This behavior is coupled with fluid becoming entrained inside the dimples, recirculating and ejecting away from the surface. When flow injection occurs through the surface, this process occupies a larger region of the flow, extending further from the surface, with a corresponding increase in the size of the convecting structures and increase in turbulent kinetic energy. These features persist over the range of Reynolds numbers investigated, with increasing Reynolds number resulting in increased turbulence and a corresponding broadening of the region of the flow influenced by the surface.

8:26AM G21.00003 Most-Critical Transient Disturbances in an Incompressible Flat-Plate Boundary Layer1. JASON MONSCHKE, EDWARD WHITE, Texas A&M University — Transient growth is a linear disturbance growth mechanism that plays a key role in roughness-induced boundary-layer transition. It occurs when superposed stable, non-orthogonal continuous spectrum modes experience algebraic disturbance growth followed by exponential decay. Algebraic disturbance growth can modify the basic state making it susceptible to secondary instabilities rapidly leading to transition. Optimal disturbance theory was developed to model the most-dangerous disturbances. However, evidence suggests roughness-induced transient growth is sub-optimal yet leads to transition earlier than optimal theory suggests. This research computes initial disturbances most unstable to secondary instabilities to further develop the applicability of transient growth theory to surface roughness. The main approach is using nonlinear adjoint optimization with solutions of the parabolized Navier–Stokes and BGlobal stability equations. Two objective functions were considered: disturbance kinetic energy growth and sinuous instability growth rate. The first objective function was used as validation of the optimization method. Counter-rotating streamwise vortices located low in the boundary layer maximize the sinuous instability growth rate.

1This research is supported by NASA Award NNX13AN04A

1The authors would like to acknowledge NASA and the AFOSR for funding this work through AFOSR Grant FA9550-09-1-0341.
Turbulence in soft-walled micro-channels. S.S. Srinivas, V. Kumaran, Indian Institute of Science, Bangalore 560 012, India. — Transition to turbulence in soft-walled microchannels occurs at a much lower Reynolds number (Re) than that when the walls are rigid. To gain insights into the transition, we have studied the fluid flow in these channels using Particle Imaging Velocimetry, along the streamwise and wall-normal directions. The dimensions of the microchannels studied are approximately 4 mm × 160 μm × 1.5 mm. We see qualitative agreement between experiments and simulations, considering channel deformation, for laminar flows. Significant departure from the laminar flow profile is seen after transition. The root mean square of velocity fluctuations along streamwise and wall-normal directions is unsymmetric and is non-zero at the wall. This gives rise to non-zero Reynolds stress at the surface, indicating the coupling between soft surface and the fluid. The turbulence production term too is nonzero at the surface, as opposed to that for a rigid channel flow. This implies there is transfer of energy from the surface to the fluid. The scaled maximum of the velocity fluctuations and the Reynolds stress (divided by the fluid density) in the soft-walled microchannel for Re in the range 250-400 are comparable to those in a rigid channel at Re in the range 5000-20000.

The authors would like to thank the Department of Science and Technology, Government of India for financial support.

Correlations of Surface Deformation and 3D Flow Field in a Compliant Wall Turbulent Channel Flow. JIN WANG, CAO ZHANG, JOSEPH KATZ, Johns Hopkins Univ — This study focuses on the correlations between surface deformation and flow features, including velocity, vorticity and pressure, in a turbulent channel flow over a flat, compliant Polydimethylsiloxane (PDMS) wall. The channel centerline velocity is 2.5 m/s, and the friction Reynolds number is 2.3×10^5. Analysis is based on simultaneous measurements of the time resolved 3D velocity and surface deformation using tomographic PIV and Mach-Zehnder Interferometry. The volumetric pressure distribution is calculated plane by plane by spatially integrating the material acceleration using virtual boundary, omni-directional method. Conditional sampling based on local high/low pressure and deformation events reveals the primary flow structures causing the deformation. High pressure peaks appear at the interface between sweep and ejection, whereas the negative deformations peaks (dent) appear upstream, under the sweeps. The persistent phase lag between flow and deformations are presumably caused by internal damping within the PDMS. Some of the low pressure peaks and strong ejections are located under the head of hairpin vortices, and accordingly, are associated with positive deformation (bump). Others bumps and dents are correlated with some spanwise offset large inclined quasi-streamwise vortices that are not necessarily associated with hairpins.

The time-resolved 3D flow field and 2D distribution of wall-normal deformation are measured simultaneously using tomographic PIV combined with Mach-Zehnder interferometry. A new interferogram filtration technique based on spatial correlations of small windows, followed by phase calculation from intensity arccosines, is introduced to capture submicron deformations. It has lower errors and sensitivity to fringe shape compared to spectral band-pass filtering. The measured wavenumber-frequency spectra show the deformation consists of patterns that are larger than the field-of-view, surface waves, and small-scale patterns. Some of the latter are advected at the channel centerline velocity, \( U_c \), but most are advected at 0.7\( U_c \), the mean speed at 10% of the channel half height, \( h \). Correlations between deformation and velocity conditioned on the sign of the deformation indicate the positive and negative deformations are related to the ejection and sweeping events, respectively. The correlation peaks also reside at about 0.1\( h \), suggesting this is the elevation where relevant coherent structures are concentrated.

Sponsored by ONR

Experimental Investigation on Near-wall Turbulent Flow Structures over Deformable Roughness. MOSTAFA TOLOUI, NOLAN JOHN, JIARONG HONG, University of Minnesota — Wall-bounded turbulent flows over rough surfaces have been studied for almost a century. However, in most of the prior studies, little attention has been paid to the role of roughness mechanical properties, e.g. deformability, in altering the flow characteristics including both general turbulent statistics and near-wall flow structures. In this study, high resolution time-resolved digital in-line holographic PIV is employed to investigate the near-wall turbulent structures as well as turbulent statistics around and above deforming roughness structures. The rough wall samples consisting of tapered cylinders of size 0.5 mm in diameter and 3 mm in height are manufactured from transparent PDMS with similar geometrical features but various deformability levels. The experiments are conducted within an optically index-matched facility (using NaI solution) operating with different Reynolds numbers where roughness samples of different deformability are placed downstream of a 1.2 m long acrylic channel of 50 mm square cross section. The follow-up research envisions a large dataset including various Reynolds numbers and deformability to elucidate the role of roughness deformability on near-wall coherent structures and turbulent energy transport within and above the roughness sublayer.

This work is supported by the startup package of Jiarong Hong and the MnDrive Fellowship of Mostafa Tolouei from University of Minnesota.
9:44AM G21.00009 Internal Structure and Interaction Within Turbulent Boundary Layers Following a Change in Surface Roughness, RONALD HANSON, BHRATHRAM GANAPATHISUBRAMANI, University of Southampton — In this experimental study we consider the turbulent boundary layer developing past a surface that abruptly changes from rough to smooth. The change in surface condition leads to the formation of an internal layer. Above this layer the flow is characteristic of the upstream condition. Within the internal layer the near-wall turbulence establishes under the influence of the outer region remaining from the incoming rough-wall boundary layer. Wide-field Particle Image Velocimetry measurements were used to capture the development of the boundary layer over the smooth wall downstream of the rough surface. These measurement enable investigation of key features such as the structure inclination angle, which are considered to be invariant in equilibrium boundary layers. However, the structure within the internal layer resembles a smooth wall boundary layer and above the internal layer the structure resembles the upstream rough wall flow. Using the simultaneously monitored temporal streamwise velocity from two hotwires, one located within the peak energetic region of the near-wall and the other within the outer region of the boundary layer, interaction occurring across the internal layer will be examined with respect to the development of the evolving boundary layer.

9:57AM G21.00010 Analysis of velocity and thermal structures in a transitionally rough turbulent boundary layer, ALI DOOSTTALAB, SURANGA DHARMARATNE, Texas Tech University, GUILLERMO ARAYA, Mechanical Engineering Department, University of Puerto Rico – Mayaguez, MURAT TUTKUN, Institute for Energy Technology (IFE), Department of Process and Fluid Flow Technology, Norway, RONALD ADRIAN, Arizona State University, LUCIANO CASTILLO, Texas Tech University — A zero pressure gradient turbulent boundary layer flowing over a transitionally rough surface (24-grit sandpaper) with \( k^+ = 11 \) and Reynolds numbers based on momentum thickness of around 2400 is studied using direct numerical simulation (DNS). Heat transfer between the isothermal rough-surface and the turbulent flow with molecular Prandtl number \( P_T = 0.71 \) is simulated. The dynamic multi-scale approach developed by Araya et al. (2011) is employed to prescribe realistic time-dependent thermal inflow boundary conditions. Above the roughness sub-layer (3 - 5\( k \)) it is found that statistics of the temperature field, including higher order moments and conditional averages, are the same for the smooth and rough surface flow, showing that the Townsend’s Reynolds number similarity hypothesis applies for the thermal field as well as the velocity field for the Reynolds number and \( k^+ \) considered in this study. Also the velocity and thermal structures of the developing boundary layer were studied by means of multi-point correlations and POD analysis.

Monday, November 23, 2015 8:00AM - 10:10AM – Session G22 Turbulent Boundary Layers II

8:00AM G22.00001 On the scaling of velocity and vorticity variances in turbulent channel flow, A. LEONARD, California Institute of Technology — The availability of new DNS-based statistics for turbulent channel flow (Lee & Moser, JFM 2015) along with previous results (e.g., Hoyas & Jiménez, Phys. Flu. 2006) has provided the opportunity for another look at the scaling laws for this flow. For example, data from the former (fig. 4(e)) for the streamwise velocity variance in the outer region clearly indicate a modified log law for that quantity at \( Re_y = 5200 \), i.e., \( u'^2 > C_0 - C_1 \ln(y/\delta) - C_2 \ln(y/\delta)^2 \) where \( \delta \) is the channel half width. We find that this result fits the data very well for \( 0.1 < y/\delta < 0.8 \). The Reynolds number (5200) is still apparently too low to observe the much-discussed log law (above \( C_2 = 0 \)), which, presumably, would appear for roughly \( y/\delta < 0.1 \), as it does in high \( Re_y \) pipe flow (Hultmark et al., PRL 2012) with \( \delta \) replaced by \( R \). On the other hand, the above modified log law with the same values for \( C_1 \) and \( C_2 \) is a good fit for the pipe data at \( Re_y = 98 \times 10^5 \) for \( y/R > 0.12 \) (fig. 4 of Hultmark et al.).

8:13AM G22.00002 Turbulent inertia and the onset of log region in pipe flows\(^1\), JIMMY PHILIP, CHENG CHIN, Univ of Melbourne, JOSEPH KLEWICKI, Univ of New Hampshire and Univ of Melbourne, ANDREW OOI, IVAN MARUSIC, Univ of Melbourne — The wallnormal \( y \)-location where the log-region begins in wall-turbulence is the same location where the turbulent inertia or TI (\( d(-\Phi)/dy \)) and the pressure gradient terms from the mean-momentum equation start balancing each other. This location is closely followed by the location, \( y_{in} \), where TI vanishes (before becoming negative in the log-region). Dynamics of TI is elucidated using DNS data of pipe flow at \( \delta^+ = 500, 1000 \) and 2000. We decompose TI as (i) velocity-vorticity correlations \( (\Phi_{ux} + \Phi_{w' \omega}) \) and their co-spectra, and (ii) wall-normal gradient of the Reynolds shear stress co-spectra \( \Phi_{w' \omega} \). One interesting result is that the onset of the log-region moves outward with increasing Reynolds number as \( \sim \sqrt{Re} \) because the eddies located close to \( y_{in} \) are influenced by large scale accelerating motions of the type \( -w_{\omega y} \) related to vorticity stretching.

\(^1\)The authors acknowledge the financial support of the Australian Research Council.

8:26AM G22.00003 Role of large scale motion in high Re channel flow\(^1\), MYOUNGKYU LEE, ROBERT D. MOSER, University of Texas at Austin — Direct numerical simulations (DNS) of turbulent channel flow at Reynolds numbers up to \( Re_y \approx 5200 \) have been performed to study high Reynolds number wall-bounded turbulence. DNS result have shown that \( Re_y \approx 5200 \) is high enough to exhibit scale separation between the near-wall and outer regions. [Lee & Moser, J. Fluid Mech., vol 774, 2015]. In this presentation we focus on the role of large scale motion on the transport of turbulent kinetic energy, \( u'w'/2 \), and Reynolds stress, \( u'v' \). Spectral analysis of the evolution equation for the two-point correlation is performed to investigate the contribution of motions at different length scales to transport. It is shown that only the turbulent transport terms show significant \( Re \) dependencies. Furthermore, the turbulent transport terms can be decomposed into two parts, one that contributes to transport in the wall-normal direction and one that is responsible for transfer between length scales. The results show that the large scale motion in the outer region has direct effects on the flow in the near-wall region through transport of turbulent kinetic energy and Reynolds stress.

\(^1\)This work was supported by NSF (OCI-0749223 and PRAC Grant 0832634), and computation resources were provided by the Argonne Leadership Computing Facility through the Early Science, INCITE 2013 and Directors Discretionary Programs.
8:39AM G22.00004 Experiments on low Reynolds number turbulent flow through a square duct. 

Bayode Owolabi, Robert Poole, David Dennis, University of Liverpool — Previous experimental studies on square duct turbulent flow have focused mainly on high Reynolds numbers for which a turbulence induced eight-vortex secondary flow pattern exists in the cross sectional plane. More recently, Direct Numerical Simulations (DNS) have revealed that the flow field at Reynolds numbers close to transition can be very different; the flow in this marginally turbulent regime alternating between two states characterised by four vortices. In this study, we experimentally investigate the onset criteria for transition to turbulence in square ducts. We also present experimental data close to transition can be very different; the flow in this marginally turbulent regime alternating between two states characterised by four vortices.

8:52AM G22.00005 Structural organization of uniform momentum core in turbulent channel flow.

Jongmin Yang, Jinyul Hwang, Hyung Jin Sung, Korea Adv Inst of Sci & Tech — The coherent structures across the boundary of the quiescent region are explored using the direct numerical simulation data of a turbulent channel flow at Re = 930. The quiescent core is the region where the streamwise momentum is relatively uniform with low-level turbulence in channel flow. Across the boundary of this region, the turbulence intensity and the Reynolds shear stress decrease suddenly. The mean velocity profile shows a significant jump which indicates a strong mean shear layer at the boundary of the uniform core region. Due to the strong mean shear, the prograde vortices are dominantly distributed along the boundary with the retrograde vortices below them. The prograde and retrograde vortices are distributed in a pair with a uniform wall-normal distance. Large-scale low- and high-speed structures are characterized by the positions of the core boundary, revealing that the core boundary is modulated by the large-scale structures.

1This work was supported by the Creative Research Initiatives (No. 2015-001828) program of the National Research Foundation of Korea (MSIP) and supported by the Supercomputing Center (KISTI).

9:05AM G22.00006 Multiscale dynamics of the strain and enstrophy in turbulent channels.

Adrian Lozano-Duran, Technical University of Madrid, Markus Holzner, ETH Zurich, Javier Jimenez, Technical University of Madrid — The invariants of the velocity gradient tensor, Q and R, and their enstrophy and strain components are studied in the log-layer of a turbulent channel. The velocities are filtered in the three spatial directions and the results analyzed at different scales. We show that the Q–R plane does not capture the changes undergone by the flow as the filter width increases, and that the enstrophy/enstrophy production and strain/enstrophy planes are better choices. We also show that the conditional mean trajectories may differ significantly from the instantaneous behavior of the flow since they are the result of an averaging process where the mean is 3-5 times smaller than the corresponding standard deviation. Our final goal is to test whether the dynamics of the flow are self-similar in the inertial range and the answer turns out to be no. The mean shear is found responsible for the absence of self-similarity and progressively controls the dynamics of the eddies observed as the filter width increases. However, a self-similar behavior emerges when the calculations are repeated for the fluctuating velocity. Finally, the turbulent cascade in terms of vortex stretching is considered by computing the alignment of the vorticity at a given scale with the strain at a larger one.

1funded by ERC

9:18AM G22.00007 Role of large-scale motions to turbulent inertia in turbulent pipe and channel flows.

Jinyul Hwang, Kaist, Jin Lee, Johns Hopkins University, Hyung Jin Sung, Kaist — The role of large-scale motions (LSMs) to the turbulent inertia (TI) term (the wall-normal gradient of the Reynolds shear stress) is examined in turbulent pipe and channel flows at Re = 930. The TI term in the mean momentum equation represents the net force of inertia exerted by the Reynolds shear stress. Although the turbulence statistics characterizing the internal turbulent flows are similar close to the wall, the TI term differs in the logarithmic region due to the different characteristics of LSMs (λ > 3δ). The contribution of the LSMs to the TI term and the Reynolds shear stress in the channel flow is larger than that in the pipe flow. The LSMs in the logarithmic region act like a mean momentum source (where TI > 0) even the TI profile is negative above the peak of the Reynolds shear stress. The momentum sources carried by the LSMs are related to the low-speed regions elongated in the downstream, revealing that momentum source-like motions occur in the upstream position of the low-speed structure. The streamwise extent of this structure is relatively long in the channel flow, whereas the high-speed regions on the both sides of the low-speed region in the channel flow are shorter and weaker than those in the pipe flow.

1This work was supported by the Creative Research Initiatives (No. 2015-001828) program of the National Research Foundation of Korea (MSIP) and partially supported by KISTI under the Strategic Supercomputing Support Program.

9:31AM G22.00008 The inertial subrange in turbulent pipe flow: centre line.

Jonathan Morrison, Department of Aeronautics, Imperial College London, Margit Vallikivi, GE Global Research, Munich, Alexander Smits, MAE, Princeton University — The inertial subrange scaling of the axial velocity component is examined for the centre line of turbulent pipe flow for Reynolds numbers in the range 249 ≤ Re ≤ 986. Measurements were performed in the Princeton/ONR Superpipe using NSTAP probes of length, l = 30 μm or 60 μm, with temporal resolution up to 300 kHz. Estimates of the dissipation rate, ε, are made by both integration of the one-dimensional dissipation spectra and the third-order moment of the structure function. It is noticeable that neither dissipation estimate provides values of ≥ 2μm/R that asymptote to a constant: rather ε increases almost linearly with Re. We show that complete similarity of the inertial range spectra is not evident: there is little support for K41, and effects of Reynolds number are not well represented by Kolmogorov’s “extended similarity hypothesis,” K52. The second-order moment of the structure function does not show a constant value, even when compensated by K52. Direct effects of viscosity appear at the centre line where correction of the “4/5ths” constant for finite Reynolds number (Lundgren 2002) yields values of 0.80 ± 0.01.

1ONR Grant N00014-13-1-0174
9:44AM G22.00009 Amplitude modulation of vorticity of viscosity and dissipation by large-scale motions in turbulent channel flow\(^1\). YI-CHEN YAO, BING-QING DENG, WEI-XI HUANG, CHUN-XIAO XU, Tsinghua University, TURBULENCE RESEARCH TEAM — Amplitude modulation of both vorticity and dissipation by large-scale out-layer structures is studied using the DNS data of turbulent channel flow at Reynolds numbers up to \(Re_y = 1000\). Carrier and modulated signals are scale decomposed in both streamwise and spanwise directions, and small-scale envelope is extracted by Hilbert transformation. Two-point amplitude modulation correlation is calculated at a range of wall-normal locations. The modulation strength on the vorticity and the dissipation rate of turbulent kinetic energy is found to be much stronger than on all the three components of velocity fluctuations. Distinct peak value of correlation is observed when large-scale signals are extracted from center log region and the corresponding modulated information from below \(y^+ = 10\). Also the strength of this peak value increases with Reynolds number, thus supporting the top-down mechanism that the near-wall layer is becoming more influenced by the large-scale structures which gradually emerge as Reynolds number increases.

\(^1\)The work was supported by National Natural Science Foundation of China (11490551, 11132005, 11322221).

9:57AM G22.00010 Fully developed turbulence in slugs of pipe flows. RORY CERBUS, CHEN-CHIA LIU, Okinawa Institute of Science and Technology, JUN SAKAKIBARA, Meiji University, GUSTAVO GIOIA, PINAKI CHAKRABORTY, Okinawa Institute of Science and Technology — Despite over a century of research, transition to turbulence in pipe flows remains a mystery. In theory the flow remains laminar for arbitrarily large Reynolds number, \(Re\). In practice, however, the flow transitions to turbulence at a finite \(Re\) whose value depends on the disturbance, natural or artificial, in the experimental setup. The flow remains in the transition state for a range of \(Re \sim 0(1000)\); for larger \(Re\) the flow becomes fully developed. The transition state for \(Re > 3000\) consists of axially segregated regions of laminar and turbulent patches. These turbulent patches, known as slugs, grow as they move downstream. Their lengths span anywhere between a few pipe diameters to the whole length of the pipe. Here we report Stereo Particle Image Velocimetry measurements in the cross-section of the slugs. Notwithstanding the continuous growth of the slugs, we find that the mean velocity and stress profiles in the slugs are indistinguishable from that of statistically-stationary fully-developed turbulent flows. Our results are independent of the length of the slugs. We contrast our results with the well-known work of Wygnanski & Champagne (1973), whose measurements, we argue, are insufficient to draw a clear conclusion regarding fully developed turbulence in slugs.

Monday, November 23, 2015 8:00AM - 10:10AM — Session G23 Biofluidics: Active Fluids III

8:00AM G23.00001 Emergent dynamics and phase behavior of interacting ellipsoid micro-swimmers\(^1\). ENKELEIDA LUSHI, Brown University — Suspensions of self-propelling ellipsoid particles are known to display distinct phases depending on the density and particle aspect ratio: dilute gas, jammed state, swarms, bionematic state, turbulence or lanes. Each of these non-equilibrium phases exhibits distinct characteristics. Recent studies on the other hand have shown that the dynamics observed in bacterial suspensions can be better explained when accounting also for the disturbance fluid flow generated by the swimmers and their interactions through it. Using numerical simulations, we show that including the fluid interactions in the dynamics of ellipsoid motile particles modifies the system dynamics, e.g. the clustering of the swimmers and other physical characteristics. We show that the phase state diagram is significantly altered depending on the type of swimmer ("pusher" or "puller" vs. "mover") as well the swimmer density and aspect ratio.

\(^1\)We acknowledge partial support from NSF CBET 1544196.

8:13AM G23.00002 Stability of localized bioconvection patterns of Euglena gracilis suspensions\(^1\). MAKOTO IIIMA, Hiroshima University, TAKAYUKI YAMAGUCHI, Hokkaido University — Suspension of Euglena gracilis forms localized convection cells when it is illuminated form below with strong light intensity. Two elementary localized structures are known. One consists of a single region of high number density of the microorganism sandwiched with a pair of convection cells (bioconvection unit) and the other is a localized traveling wave. Measurements of the flux of the number density suggests that the photomovement due to light gradient plays an important role in generating localized convection cells. We proposed a hydrodynamic model incorporating the effect, and succeed in reproducing bioconvection unit, which can be characterized as steady solutions of the proposed model. Bifurcation structure of the solutions are analyzed. The bistable region due to the subcritical bifurcation from trivial state and folding of branch due the saddle-node bifurcation is observed. The stability analysis in the bistable region revealed that the most unstable mode represents a sweep of number density to the central part and reducing the size of the convection cells, which leads the unstable solution to the stable steady solution representing bioconvection unit.

\(^1\)KAKENHI (26400396)

8:26AM G23.00003 Symmetry-breaking phase-transitions in highly concentrated semen. FRANCK PLOURABOU, ADAMA CREPPIE, OLIVIER PRAUD, XAVIER DRUART, Sbastien CAZIN, Université de Toulouse, INPT, UPS, IMFT Alès Camille Soul, F-31400 Toulouse, France, and CNRS, HUI YU, PIERRE DEGOND, Department of Mathematics, Imperial College London, London SW7 2AZ, United Kingdom — New experimental evidence of self-motion of a confined active suspension is presented. Depositing fresh semen sample in an annular shaped micro-fluidic chip leads to a spontaneous rotation motion of the fluid at sufficiently large \(Re\) whose value depends on the disturbance, natural or artificial, in the experimental setup. The flow remains in the transition state for a range of \(Re \sim 0(1000)\); for larger \(Re\) the flow becomes fully developed. The transition state for \(Re > 3000\) consists of axially segregated regions of laminar and turbulent patches. These turbulent patches, known as slugs, grow as they move downstream. Their lengths span anywhere between a few pipe diameters to the whole length of the pipe. Here we report Stereo Particle Image Velocimetry measurements in the cross-section of the slugs. Notwithstanding the continuous growth of the slugs, we find that the mean velocity and stress profiles in the slugs are indistinguishable from that of statistically-stationary fully-developed turbulent flows. Our results are independent of the length of the slugs. We contrast our results with the well-known work of Wygnanski & Champagne (1973), whose measurements, we argue, are insufficient to draw a clear conclusion regarding fully developed turbulence in slugs.

8:39AM G23.00004 Flock on a chip. DENIS BARTOLO, NICOLAS DESREUMAUX, ENS - Lyon — We will show how to motorize colloidal particles capable of sensing the orientation of their neighbors and how to handle them in microfluidic chips. These populations of colloidal rollers display non-equilibrium transitions toward swarming or swirling motion depending on the system geometry. After characterizing these emergent patterns we will quantitatively describe them by means of an hydrodynamic theory of polar active liquids.
8:52AM G23.00005 Using a stochastic field theory to understand group behavior in microswimmer suspensions, PATRICK UNDERHILL, YUZHOU QIAN, PETER KRAMER, Rensselaer Polytechnic Institute — Active suspensions of microswimmers appear both in biological and artificial systems (e.g. bacteria and algae) and in synthetic systems. Even without external forcing they are out of equilibrium, which gives rise to interesting properties in both small and large concentrations of the particles. These properties have been observed in experiments as well as simulation/modeling approaches. It is important to understand how hydrodynamic interactions between active swimmers cause and/or alter the suspension properties including enhanced transport and mixing. One of the most successful approaches has been a mean field theory. However, in some situations the mean field theory makes predictions that differ significantly from experiments and direct (agent or particle based) simulations. There are also some quantities that cannot be calculated by the mean field theory. In this talk, we will describe our new approach which uses a stochastic field to overcome the limitations of the mean field theory. It allows us to calculate how interactions between organisms alter the correlations and mixing in conditions where the mean field theory cannot.

9:05AM G23.00006 Hydrodynamic analysis and mechanisms of ciliary beating1, ASHOK SANGANI, KENNETH FOSTER, Syracuse University — The scaffold of a cilium or eukaryotic flagellum, known as the axoneme, consists of nine microtubule doublets surrounding a pair of singlet microtubules. Attached to each doublet are periodically-spaced dynein motors that use energy from ATP hydrolysis to exert force on the microtubules, moving them away from the cell. In contrast to the studies that have been put forward over the last several decades to explain how these motors work collectively to produce steady beating, this question remains unresolved: we shall show that the forces generated during the beating as determined from the detailed hydrodynamic analysis are inconsistent with the predictions of the existing theories. We shall also use the experimental data available in the literature to present empirical results for the beat properties, i.e., the frequency, amplitude, wavelength, and energy dissipation, as functions of ATP concentration and fluid viscosity. The power dissipated and the energy per wavelength are approximately the same independent of the viscosity of the fluid and the ATP concentration. We use these observations to suggest a new hypothesis regarding how the cilia beat.

1This work was supported by the National Science Foundation.

9:18AM G23.00007 Emergence of collective motion in suspensions of swimming cells, MARIA CHIARA ROFFIN, PETR DENISSÉNKO, VASILY KANTSLER, University of Warwick, UK — Collective motion is one of the most fascinating manifestations of self-organization in non-equilibrium systems. The phenomena emerges with the increase in concentration of motile individuals ranging from molecular motors to large animals like fish and humans. We have studied the suspension of swimming sperm cells in a microfluidic device which gradually concentrates motile cells in the region of interest. The onset of collective motion is identified by investigating correlations of fluid velocity and image brightness associated with the cell orientation. Cell concentration and the noise parameter are varied to switch on/off the collective interaction. The level of noise is controlled by adjusting the cell motility which depends on the temperature in the microfluidic chip. Fluid velocity is measured by tracing passive fluorescent beads in the suspension.

9:31AM G23.00008 Aerotactic Cell Density Variations in Bacterial Turbulence, VICENTE FERNANDEZ, STEVEN SMRIGA, MIT and ETH Zurich, FILIPPO MENOLASCINA, MIT and University of Edinburgh, ROBERTO RUSCONI, ROMAN STOCKER, MIT and ETH Zurich — Concentrated suspensions of motile bacteria such as Bacillus subtilis exhibit group dynamics much larger than the scale of an individual bacterium, visual similar to high Reynolds number turbulence. These suspensions represent a microscale realization of active matter. Individually, B. subtilis are also aerotactic, and will accumulate near oxygen sources. Using a microfluidic device for generating oxygen gradients, we investigate the relationship between individuals’ attraction to oxygen and the collective motion resultant from hydrodynamic interactions. We focus on changes in density revealed by a fluorescently labeled sub-population of B. subtilis in the dense suspension. This approach allows us to examine changes in density during the onset of collective motion as well as fully developed bacterial turbulence.

9:44AM G23.00009 A Rules-Based Simulation of Bacterial Turbulence, MAXWELL MIKELSTITES, ANNE STAPLES, Virginia Tech — In sufficiently dense bacterial populations (>40% bacteria by volume), unusual collective swimming behaviors have been consistently observed, resembling von Karman vortex streets. The source of these collective swimming behavior has yet to be fully determined, and as of yet, no research has been conducted that would define whether or not this behavior is derived predominantly from the properties of the surrounding media, or if it is an emergent behavior as a result of the “rules” governing the behavior of individual bacteria. The goal of this research is to ascertain whether or not it is possible to design a simulation that can replicate the qualitative behavior of the densely packed bacterial populations using only behavioral rules to govern the actions of each bacteria, with the physical properties of the media being neglected. The results of the simulation will address whether or not it is possible for the system’s overall behavior to be driven exclusively by these inter-agent dynamics. In order to examine this in MATLAB, the fixed-grid, and updated sequentially with the bacterial behavior, including randomized tumbling, gathering and perceptual sub-functions. If the simulation is successful, it will serve as confirmation that it is possible to generate these qualitatively vortex-like behaviors without specific physical media. These properties have been observed in experiments as well as simulation/modeling approaches. It is important to understand how hydrodynamic interactions between active swimmers cause and/or alter the suspension properties including enhanced transport and mixing. One of the most successful approaches has been a mean field theory. However, in some situations the mean field theory makes predictions that differ significantly from experiments and direct (agent or particle based) simulations. There are also some quantities that cannot be calculated by the mean field theory. In this talk, we will describe our new approach which uses a stochastic field to overcome the limitations of the mean field theory. It allows us to calculate how interactions between organisms alter the correlations and mixing in conditions where the mean field theory cannot.

9:57AM G23.00010 Numerical study of the hydrodynamic interactions in an E-coli suspension, XINLIANG XU, Beijing Computational Science Research Center — The active suspension of E-coli displays many interesting non-equilibrium phenomena, e.g. “swarming” at high bacteria concentrations, and viscosity change under simple shear. To understand the micromixing underlying these phenomena reveals in detail about the hydrodynamics of the suspension. Here we numerically study in detail the hydrodynamic interactions between a bacterium and an ellipsoid tracer at small separations, where the tracer can no longer be treated as a point-like particle that creates no disturbance to local flow field. Based on Purcell’s É-ticull model, we observed a significant drop in bacterium swimming velocity, in agreement with previous experimental study.

Monday, November 23, 2015 8:00AM - 10:10AM Session G24 Biofluids: Biofilms II 302 - Justas Dauparas, University of Cambridge

8:00AM G24.00001 Flows around bacterial swarms, JUSTAS DAUPARAS, ERIC LAUGA, Department of Applied Mathematics and Theoretical Physics, University of Cambridge — Flagellated bacteria on nutrient-rich substrates can differentiate into a swarming state and move in dense swarms across surfaces. A recent experiment (HC Berg, Harvard University) measured the flow in the fluid around the swarm. A systematic chiral flow was observed in the clockwise direction (when viewed from above) ahead of a E.coli swarm with flow speeds of about 10 μm/s, about 3 times greater than the radial velocity at the edge of the swarm. The working hypothesis is that this flow is due to the flagella of cells stalled at the edge of a colony which extend their flagellar filaments outwards, moving fluid over the virgin agar. In this talk we quantitatively test his hypothesis. We first build an analytical model of the flow induced by a single flagellum in a thin film and then use the model, and its extension to multiple flagella, to compare with experimental measurements.
8:13AM G24.00002 Bacterial floc mediated rapid streamer formation in creeping flows, MAHTAB RASSANPOURFARD, Department of Chemical and Materials Engineering, University of Alberta, Edmonton, Canada, ZAHRA NIHAKHTARI, Department of Mechanical Engineering, University of Alberta, Edmonton, Canada, RANAJAY GHOSH, Department of Mechanical and Industrial Engineering, Northeastern University, Boston MA 02115, USA, SIDDHARTHDA S, Department of Mechanical Engineering, University of Maryland, College Park, MD 20742, USA, THOMAS THUNDAT, Department of Chemical and Materials Engineering, University of Alberta, Edmonton, Canada — One of the contentious problems regarding the interaction of low Reynolds number (Re<1) fluid flow with bacterial biomass is the formation of filamentous structures called streamers. Recently, we discovered that streamers can be formed from flow-induced deformation of the pre-formed bacterial flocs over extremely small timescales (less than a second). However, these streamers are different than the ones that mediated by biofilms. To optically probe the inception process of these streamers formation, bacterial flocs were embedded with 200 nm red fluorescent polystyrene beads that served as tracers. We also showed that at their inception the deformation of the flocs is dominated by large recoverable strains indicating significant elasticity. These strains subsequently increase tremendously to produce filamentous streamers. At time scales larger than streamers formation time scale, viscous response was observed from streamers. Finally, rapid clogging of microfluidic devices occurred after these streamers formed.

8:26AM G24.00003 Exploration of fluid dynamic indicators/causative factors in the formation of tower structures in staphylococci bacteria biofilms, ERICA SHERMAN, University of Nebraska - Lincoln, MOORMIEIER DEREK, KENNETH BAYLES, University of Nebraska - Medical Center, TIMOTHY WEI, University of Nebraska - Lincoln — Staphylococcus aureus bacteria form biofilms with distinct structures that facilitate their ability to tolerate treatment and spread within the body. As such, staph infections represent one of the greatest threats to post-surgery patients. It has been found that flow conditions play a significant role in the development and dispersal activity of a biofilm. The coupling between the growing biofilm and surrounding flow, however, is not well understood. Indeed, little is known why bacteria form tower structures under certain conditions but not in a predictable way. \( \mu \)-PTV measurements were made in a microchannel to try to identify fluid dynamic indicators for the formation of towers in biofilm growth. Preliminary experiments indicated changes in the near wall flow up to five hours before a tower formed. The reason for that is the target of this investigation. Staphylococcus aureus bacteria were cultured in the Bioflux fluxion channel and subjected to a steady shear rate of 0.5 dynes. In addition to \( \mu \)-PTV measurement, nucleacse production and cell number density counts were observed prior to and during tower development. These were compared against measurements made under the same nominal flow conditions where a tower did not form.

8:39AM G24.00004 A Model Problem for Blob-Driven Tear Film Breakup (TBU), LAN ZHONG, C.F. KETELAAR, R.J. BRAUN, T.A. DRISCOLL, University of Delaware, P.E. KING-SMITH, The Ohio State University, CAROLYN G. BEGLEY, Indiana University — A model problem is developed to simulate tear film break up by assuming the existence of a flexible non-spreadinig blob with constant surfactant surface concentration. These assumptions model in vivo observations of an excess of tear film lipid that does not spread, with the surfactant concentration approximating the lipid layer. The model includes the effects of evaporation, osmolarity, surface tension, viscosity, the Marangoni effect and insoluble surfactant transport. The evaporative fluxes are input as representative functions based on data from experiments. A strong flow driven by surface tension gradient is observed from the numerical results, which may drive TBU at times compatible with in vivo observations. The TBU dynamics are studied as functions of blob size, surfactant properties and other parameters to establish regimes were TBU may be driven primarily by Marangoni effects.

1 NSF grant 1412085 and NIH grant IROI1EY021794

8:52AM G24.00005 Bulk flow coupled to a viscous interfacial film sheared by a rotating knife edge, ADITYA RAGHUNANDAN, FAYAZ RASHEED, AMIR HIRSA, Rensselaer Polytechnic Institute, JUAN LOPEZ, Arizona State University — The measurement of the interfacial properties of highly viscous biofilms, such as DPPC (the primary component of the lung surfactant), present on the surface of liquids (bulk phase) continues to attract significant attention. Most measurement techniques rely on shearing the interfacial film and quantifying its viscous response in terms of a surface (excess) viscosity at the air-liquid interface. The knife edge viscometer offers a significant advantage over other approaches used to study highly viscous films as the film is directly sheared by a rotating knife edge in direct contact with the film. However, accurately quantifying the viscous response is non-trivial and involves accounting for the coupled interfacial and bulk phase flows. Here, we examine the nature of the viscous response of water insoluble DPPC films sheared in a knife edge viscometer over a range of surface packing, and its influence on the strength of the coupled bulk flow. Experimental results, obtained via Particle Image Velocimetry in the bulk and at the surface (via Brewster Angle Microscopy), are compared with numerical flow predictions to quantify the coupling across hydrodynamic flow regimes, from the Stokes flow limit to regimes where flow inertia is significant.

1 Supported by NNX13AQ22G, National Aeronautics and Space Administration

9:05AM G24.00006 Fibrillization kinetics of insulin solution in an interfacial shearing flow, VIGNEISH BALARAJ, SAMANTHA MCBRIDE, AMIR HIRSA, Rensselaer Polytechnic Institute, JUAN LOPEZ, Arizona State University — Although the association of fibril plaques with neurodegenerative diseases like Alzheimer’s and Parkinson’s is well established, in-depth understanding of the roles played by various physical factors in seeding and growth of fibrils is far from well known. Of the numerous factors affecting this complex phenomenon, the effect of fluid flow and shear at interfaces is paramount as it is ubiquitous and the most varying factor in vivo. Many amyloidogenic proteins have been found to denature upon contact at hydrophobic interfaces due to the self-assembling nature of protein in its monomeric state. Here, fibrillization kinetics of insulin solution is studied in an interfacial shearing flow. The transient surface rheological response of the insulin solution to the flow and its effect on the bulk fibrillation process has been quantified. Minute differences in hydrodynamic characteristics between two variants of insulin- Human recombinant and Bovine insulin are found to result in very different responses. Results presented will be in the form of fibrillization assays, images of fibril plaques formed, and changes in surface rheological properties of the insulin solution. The interfacial velocity field, measured from images (via Brewster Angle Microscopy), is compared with numerical simulations.

1 Supported by NNX13AQ22G, National Aeronautics and Space Administration

9:18AM G24.00007 Interaction of bacterial wall with electrically charged solid substrate, VLADIMIR AJAEV, Southern Methodist University — Recent experimental studies indicate that the electrically charged substrates can exhibit antibacterial properties above a certain threshold value of the charge density. To explain these observations, we develop a mathematical model of interaction between a bacterial wall, described as a charge-regulating surface, and a charged solid substrate. Viscous flow in the aqueous film separating the two surfaces is described by a lubrication-type equation. Electrical charge transport is incorporated into the model and coupled to the flow. The complex interplay between charge transport, electrostatic interaction of the surfaces, and viscous flow leads to criteria for the critical charge density needed to achieve antibacterial properties for a range of different types of harmful bacteria.
9:31AM G24.00008 Solute Dynamics and Imaging in the Tear Film on an Eye-shaped Domain1, R.J. BRAUN, University of Delaware, LONGFEI LI, WILLIAM HENSHAW, Rensselaer Polytechnic Institute, TOBIN DRISCOLL, University of Delaware, P.E. KING-SMITH, The Ohio State University — The concentration of ions in the tear film (osmolarity) is a key variable in understanding dry eye symptoms and disease, yet its global distribution is not available; direct measurements are restricted to a region near the temporal canthus. It has been suggested that imaging methods that use solutes such as fluorescein can be used as a proxy for estimating the osmolarity. The concentration of fluorescein is not measured directly either, but the intensity as a function of concentration and thickness of the film is well established. We derived a mathematical model that couples multiple solutes and fluid dynamics within the tear film on a 2D eye-shaped domain. The model includes the physical effects of evaporation, surface tension, viscosity, ocular surface wettability, osmolarity, osmosis, fluorescence and tear fluid supply and drainage. We solved the governing system of coupled nonlinear PDEs using the Overture computational framework developed at LLNL, together with a hybrid time stepping scheme (using variable step BDF and RK4C). Results of our numerical simulations provide new insight about the osmolarity distribution and its connection with images obtained in vivo over the whole ocular surface and in local regions of tear thinning due to evaporation and other effects.

1This work was supported in part by NSF grants 1022706 and 1412085, and NIH grant 1R01EY021794.

9:44AM G24.00009 Characteristics of turbulent boundary layer flow over algal biofilm1, ELIZABETH MURPHY, University of Virginia, JULIO BARROS, MICHAEL SCHULTZ, CECILY STEPPE, KAREN FLACK, United States Naval Academy, MATTHEW REIDENBACH, University of Virginia — Algal biofilms are an important fouling community on ship hulls, with severe economic consequences due to drag-induced increases in fuel use and cleaning costs. Here, we characterize the boundary layer flow structure in turbulent flow over diatomaceous slime, a type of biofilm. Diatomaceous slime composed of three species of diatoms commonly found on ship hulls was grown on acrylic test plates under shear stress. The slime averages 1.6 mm in thickness and has a high density of streamers, which are flexible, elongated growths with a length on the order of 1-2 mm located at the top of the biofilm that interact with the flow. Fouled acrylic plates were placed in a water tunnel facility specialized for detailed turbulent boundary layer measurements. High resolution Particle Image Velocimetry (PIV) data are analyzed for mean velocity profile as well as local turbulent stresses and turbulent kinetic energy (TKE) production, dissipation and transport. Quadrant analysis is used to characterize the impact of the instantaneous events of Reynolds shear stress (RSS) in the flow. To investigate the coherence of the large-scale motion in the flow two-point correlation analysis is employed.

1Funding provided by the Office of Naval Research and the National Science Foundation

9:57AM G24.00010 Impact of Diatomaceous Biofilms on the Frictional Drag of Ship Hull Coatings1, MICHAEL SCHULTZ, U.S. Naval Academy, JESSICA WALKER, University of Tasmania, CECILY STEPPE, KAREN FLACK, U.S. Naval Academy — Skin-friction results are presented for three commercial ship hull coatings in the unexposed, clean condition and after dynamic exposure to diatomaceous biofilms for 3 and 6 months. The experiments were conducted in a fully-developed turbulent channel flow facility spanning a wide Reynolds number range. Results show that the clean coatings tested are hydraulically-smooth over much of the Reynolds number range. Biofilms, however, result in an increase in skin-friction of up to 70%. The roughness functions for the biofilm-covered surfaces do not display universal behavior. The effect of the biofilm is observed to scale with its mean thickness and the square root of percent coverage. Boundary layer similarity-law scaling is used to predict the impact of these biofilms on the required shaft power for a mid-sized naval surface combatant at cruising speed. The increase in power is estimated to be between 1.5% and 10.1% depending on the biofilm thickness and percent coverage.

1This work was supported by ONR

Monday, November 23, 2015 8:00AM - 10:10AM –
Session G25 Biofluids: Respiratory Flows 304 - Alexandra Techet, MIT

8:00AM G25.00001 3D Spray Droplet Distributions in Sneeze ALEXANDRA TECHET, BARRY SCARFMAN, LYDIA BOUROUIBA, Massachusetts Institute of Technology — 3D spray droplet clouds generated during human sneezing are investigated using the Synthetic Aperture Feature Extraction (SAFE) method, which relies on light field imaging (LFI) and synthetic aperture (SA) refocusing computational photographic techniques. An array of nine high-speed cameras are used to image sneeze droplets and tracked the droplets in 3D space and time (3D + T). An additional high-speed camera is utilized to track the motion of the head during sneezing. In the SAFE method, the raw images recorded by each camera in the array are preprocessed and binarized, simplifying post processing after image refocusing and enabling the extraction of feature sizes and positions in 3D + T. These binary images are refocused using either additive or multiplicative methods, combined with thresholding. Sneeze droplet centroids, radii, distributions and trajectories are determined and compared with existing data. The reconstructed 3D droplet centroids and radii enable a more complete understanding of the physical extent and fluid dynamics of sneeze ejecta. These measurements are important for understanding the infectious disease transmission potential of sneezes in various indoor environments.

8:13AM G25.00002 A computational model that simulates mucociliary clearance in the bronchial tree, and a concomitant study on energetics and optimality MICHAEL MANOLIDIS, Visiting Scholar, Biomedical Engineering Department, University of Michigan - Ann Arbor, DANIEL ISABEY, Professor, Cell and Respiratory Biomechanics, Université Paris Est, Créteil, France, BRUNO LOUIS, Professor, Cell and Respiratory Biomechanics, Université Paris Est, Cité des Sciences et de l’Industrie, CRÉTEIL, France — Systemic deterministic models of mucociliary clearance in the bronchial tree are currently scarce. While analytical/computational efforts have focused on microscopic modeling of mucociliary propulsion, macroscopic approaches have been restricted mainly to stochastic methods. We present an analytical/computational model that simulates mucociliary clearance in macroscopic physical domains. The analytical foundations of the model are based on a Stokes flow assumption, whereby, in addition to viscous forces originating in ciliary forcing, the role of surface tension is also considered. The governing equations are solved computationally on a three-dimensional surface mesh. Flow is simulated in an anatomically/geométrically representative bifurcation of the bronchial tree. The directionality of ciliary forcing in our model is optimized in order to maintain near-uniform mucus film thickness throughout the flow field. Based on the optimized version of the model, energetic considerations, as well as aspects of optimality in nature are analyzed and presented.

12nd affiliation: Professor, Physique de la Matière Condensée, Ecole Polytechnique, Palaiseau, France
8:26AM G25.00003 Effect of kinematics and flexibility on the pumping dynamics of an array of oscillating plates¹, FARHAD SAFFARAVAL, KEN KIGER, Dept. of Mech. Engr., Univ. of Maryland — A robotic array of two-dimensional oscillating plates was constructed to examine the net pumping produced over a transition from viscous to inertia dominated flows. The actuators consist of single rigid plates or multiple rigid segments connected with a thin polymer film to provide for a specified degree of flexibility. The parameters for the study include: 1) inter-gill phase difference, 2) asymmetry of the protraction/retraction stroke speeds, and 3) the presence of a one-way elastic hinge. PIV measurements were conducted to examine the unsteady two-dimensional flow field at a sufficient resolution to provide measurements of the net pumped flow rate, energy dissipation, and pumping efficiency. Preliminary results at a Reynolds number of 15 show that the introduction of asymmetric flexibility under synchronous actuation of a sinusoidal waveform provides an increasing flow rate with increased flexibility. Introduction of an asymmetric stroke kinematics, however, appears to nullify the improvement effect of flexibility, with rigid and flexible gills providing comparable levels of pumping performance when using the same stroke pattern. Using a combination of stroke phasing and asymmetric kinematics shows further enhancement beyond the use of either individually.

¹work supported under NSF grant 1067066

8:39AM G25.00004 Investigation of mucus transport in an idealized lung airway model using multiphase CFD analysis. RAHUL RAJENDRAN, ARINDAM BANERJEE, Lehigh University — Mucus, a Bingham fluid is transported in the pulmonary airways by consistent beating of the cilia and exhibits a wide range of physical properties in response to the core air flow and various pathological conditions. A better understanding of the interfacial instability is required as it plays a crucial role in gas transport, mixing, mucus clearance and drug delivery. In the current study, mucus is modelled as a Newtonian fluid and the two phase gas-liquid flow in the airways is investigated using an inhomogeneous Eulerian-Eulerian approach. The complex interface between the phases is tracked using the conventional VOF (Volume of Fluid) method. Results from our CFD simulations which are performed in idealized single and double bifurcation geometries will be presented and the influence of airflow rate, mucus layer thickness, mucus viscosity, airway geometry (branching & diameter) and surface tension on mucus flow behavior will be discussed. Mean mucus layer thickness, pressure drop due single and double bifurcation geometries will be presented and the influence of airflow rate, mucus layer thickness, mucus viscosity, airway geometry (branching & diameter) and surface tension on mucus flow behavior will be discussed. Mean mucus layer thickness, pressure drop due to momentum transfer & increased airway resistance, mucus transport speed and the flow morphology will be compared to existing experimental and theoretical data.

8:52AM G25.00005 A Computational Approach for Capturing Topological Changes during the Splitting of Liquid Plug by a Pulmonary Bifurcation. BENJAMIN VAUGHAN, University of Cincinnati, JAMES GROTBERG, University of Michigan — There are certain medical treatments that involve the introduction of exogenous liquids in the lungs. These liquids can form plugs within the airways that may then propagate throughout the branching network of the lungs. The transport of liquids through the airways can cause the liquid plugs to split. The understanding of this splitting process is important for effective administration of various treatments such as surfactant replacement therapy. A significant complication in modeling the splitting process is the possibility of a topological change where the two air fingers defining the leading and trailing meniscus split into a topologically distinct configuration that consists of two liquid plugs bounded by three air fingers (one trailing and two leading). To study this process, we introduce a two-dimensional computational model that captures the propagation and splitting of a liquid plug along with the topological changes. This model consists of a finite element solver coupled with a narrow band level set approach for tracking the air/liquid interface and is shown to be efficient in capturing the full splitting process and allows for the estimation of the ratio of the resulting plugs to each other after the original plug has been split.

9:05AM G25.00006 Role of Topological Heterogeneity on the Fate of Inhaled Aerosols in the Pulmonary Acinus. PHILIP HOFEMEIER, Department of Biomedical Engineering, Technion - Israel Institute of Technology, KENICHIRO KOSHIYAMA, SHIGEO WADA, Graduate School of Engineering Science, Osaka University, JOSUE SZNITMAN, Department of Biomedical Engineering, Technion - Israel Institute of Technology — Particle transport, and ultimately deposition outcomes, in the acinar region of the lungs are intrinsically coupled with local the shape and morphology of the airways and alveolar cavities (Hofemeier and Sznitman, 2015). Thus, it is paramount to capture the complexity and heterogeneity of the acinar environment in order to predict realistic aerosol dynamics. Recently, Koshiyama and Wada (2015) introduced an algorithm to generate acinar models with space-filling heterogeneous alveolar structures to mimic realistic in vivo environments. Their model is able to reproduce the characteristic polyhedral shape and size of alveolar cavities as well as the length and branching angles of the connecting airways. Here, we utilize for the first time such acinar models as the basis for numerical simulations of respiratory acinar flows and particle transport. By generating and modeling various heterogeneous multi-generation acinar models, we aim to shed light on the role of spatial acinar heterogeneity on particle deposition fate, as a function of inhalation particle size and breathing maneuvers. The present studies are a first step towards predicting realistic acinar deposition patterns indicative for whole lung statistics as well as inter-acinar differences.

9:18AM G25.00007 Experimental evolution of sprays in a lung model. JAVIER BURGUETE, University of Navarra, ALBERTO ALISEDA, University of Washington — We present the first results of an experiment conceived to observe the evolution of sprays inside the lungs. We have built a model that covers the first 6 generations (from the trachea to segmental bronchi of 5th generation). This setup is placed on a wind tunnel, and the flow inside the model is induced by a vacuum pump that emulates the breathing process using a valve. We inject a previously determined distribution of particles (water droplets), whose average diameter can be modified. Then the droplet size and the droplet distribution in different branches and compare how the droplet distribution is modified at each generation. The parameters that control the behavior are the average diameter of the original distribution, the airflow rate inside the model and the frequency of the breathing cycle.

9:31AM G25.00008 3D Model of Surfactant Replacement Therapy. JAMES GROTBERG, CHENG-FENG TAI, University of Michigan, MARCEL FILOCHE, Ecole Polytechnique — Surfactant Replacement Therapy (SRT) involves instillation of a liquid-surfactant mixture directly into the lung airway tree. Though successful in neonatal applications, its use in adults had early success followed by failure. We present the first mathematical model of 3D SRT where a liquid plug propagates through the tree from forced inspiration. In two separate modeling steps, the plug first deposits a coating film on the airway wall which subtracts from its volume, a “coating cost”. Then the plug splits unevenly at the airway bifurcation due to gravity. The steps are repeated until a plug ruptures or reaches the tree endpoint alveolus/acinus. The model generates 3D images of the resulting acinar distribution and calculates two global indexes, efficiency and homogeneity. Simulating published literature, the earlier successful adult SRT studies show comparatively good index values, while the later failed studies do not. Those unsuccessful studies used smaller dose volumes with higher concentration mixtures, apparently assuming a well mixed compartment. The model shows that adult lungs are not well mixed in SRT due to the coating cost and gravity effects. Returning to the higher dose volume protocols could save many thousands of lives annually in the US.

¹Supported by NIH Grants HL85156, HL84370 and Agence Nationale de la Recherche, ANR no. 2010-BLAN-1119-05.
9:44AM G25.00009 Aerosol transport and deposition efficiency in the respiratory airways, Laura Nicolau, Imperial College London, Tamer Zaki, Johns Hopkins University — Prediction of aerosol deposition in the respiratory system is important for improving the efficiency of inhaled drug delivery and for assessing the toxicity of airborne pollutants. Particle deposition in the airways is typically described as a function of the Stokes number based on a reference flow timescale. This choice leads to significant scatter in deposition data since the velocity and length scales experienced by the particles as they are advected through the flow deviate considerably from the reference values in many sections of the airways. Therefore, the use of an instantaneous Stokes number based on the local properties of the flow field is proposed instead. We define the effective Stokes number as the time-average of the instantaneous value. Our results demonstrate that this average, or effective, Stokes number can deviate significantly from the reference value particularly in the intermediate Stokes number range. In addition, the effective Stokes number shows a very clear correlation with deposition efficiency, and is therefore a more appropriate parameter to describe aerosol transport.

9:57AM G25.00010 Respiratory flows during early childhood: Computational models to examine therapeutic aerosols in the developing airways.1, Janna Tenenbaum-Katan, Philipp Hofemeier, Josu Sznitman, Technion, Isreal institute of technology, Janna Tenenbaum-Katan Team — Inhalation therapy is the cornerstone of early-childhood respiratory treatments, as well as a rising potential for systemic drug delivery and pulmonary vaccination. As such, an indispensable understanding of respiratory flow phenomena, coupled with particle transport at the deep regions of children’s lungs is necessary to attain efficient targeting of aerosol therapy. However, fundamental research of pulmonary transport is overwhelmingly focused on adults. In our study, we have developed an anatomically-inspired computational model of representing pulmonary acinar regions at several age points during a child’s development. Our numerical simulations examine respiratory flows and particle deposition maps within the acinar model, accounting for varying age dependant anatomical considerations and ventilation patterns. Resulting deposition maps of aerosols alter with age, such findings might suggest that medication protocols of inhalation therapy in young children should be considered to be accordingly amended with the child’s development. Additionally to understanding basic scientific concepts of age effects on aerosol deposition, our research can potentially contribute practical guidelines to therapy protocols, and its’ necessary modifications with age.

1We acknowledge the support of the ISF and the Israel ministry of Science


8:00AM G26.00001 Shape of the human nasal cavity promotes retronasal smell, Sophie Trastour, Ecole Polytechnique and Harvard University, Simone Melchionna, Institute for Complex Systems, Rome, Shruti Mishra, David Zwicker, Daniel E. Lieberman, Efthimios Kaxiras, Michael P. Brenner, Harvard University — Humans are exceptionally good at perceiving the flavor of food. Flavor includes sensory input from taste receptors but is dominated by olfactory (smell) receptors. To smell food while eating, odor must be transported to the nasal cavity during exhalation. Olfactory performance of this retronasal route depends, among other factors, on the position of the olfactory receptors and the shape of the nasal cavity. One biological hypothesis is that the derived configuration of the human nasal cavity has resulted in a greater capacity for retronasal smell, hence enhanced flavor perception. We here study the air flow and resulting odor deposition as a function of the nasal geometry and the parameters of exhalation. We perform computational fluid dynamics simulations in realistic geometries obtained from CT scans of humans. Using the resulting flow fields, we then study the deposition of tracer particles in the nasal cavity. Additionally, we derive scaling laws for the odor deposition rate as a function of flow parameters and geometry using boundary layer theory. These results allow us to assess which changes in the evolution of the human nose lead to significant improvements of retronasal smell.

8:13AM G26.00002 How do mice follow odor trails?, David Zwicker, Harvard University, Sophie Trastour, Ecole Polytechnique and Harvard University, Shruti Mishra, Alexander Mathis, Venkatesh Murthy, Michael P. Brenner, Harvard University — Mice are excellent at following odor trails e.g. to locate food or to find mates. However, it is not yet understood what navigation strategies they use. In principle, they could either evaluate temporal differences between sniffs or they could use concurrent input from the two nostrils. It is unknown to what extent these two strategies contribute to mice performance. When mice follow trails, odors evaporate from the ground, are transported by flow in the air, and are then inhaled with the two nostrils. In order to differentiate between the two navigation strategies, we determine what information the mouse receives: first, we calculate the airflow by numerically solving the incompressible Navier-Stokes equations. We then determine the spatiotemporal odor concentration from the resulting advection-diffusion equations. Lastly, we determine the odor amount in each nostril by calculating the inhalation volumes using potential flow theory. Taken together, we determine the odor amount in each nostril during each sniff, allowing a detailed study of navigation strategies.

8:26AM G26.00003 Moths smell with their antennae, Thomas Spencer, Matthew Ballard, Alexander Alexeev, David Hu, Georgia Institute of Technology — Moths are reported to smell each other from over 5 miles away, locating each other with just 200 airborne molecules. In this study, we investigate how the structure of the antennae influences particle capture. We measure the branching patterns of over 40 species of moths, across two orders of magnitude in weight. We find that moth antennae have 3 levels of hierarchy, with dimensions on each level scaling with body size. We perform lattice-Boltzmann simulations to determine optimal flow patterns around antennae branches allowing for capture of small particles.

8:39AM G26.00004 A model for insect tracheolar flow1, Anne Staples, Virginia Tech, Krishnashish Chatterjee, Virginia Tech — Tracheoles are the terminal ends of the microscale tracheal channels present in most insect respiratory systems that transport air directly to the tissue. From a fluid dynamics perspective, tracheolar flow is notable because it lies at the intersection of several specialized fluid flow regimes. The flow through tracheoles is creeping, microscale gas flow in the rarefied regime. Here, we use lubrication theory to model the flow through a single microscale tracheole and take into account fluid-structure interactions through an imposed periodic wall deformation corresponding to the rhythmic abdominal compression found in insects, and rarefaction effects using slip boundary conditions. We compare the pressure, axial pressure gradient, and axial and radial velocities in the channel, and the volumetric flow rate through the channel for no-slip, low slip, and high slip conditions under two different channel deformation regimes. We find that the presence of slip tends to reduce the flow rate through the model tracheole and hypothesize that one of the mechanical functions of tracheoles is to act as a diffuser to decelerate the flow, enhance mixing, and increase the residency time of freshly oxygenated air at the surface of the tissue.

1This work was funded by the NSF under grant no. 1437887
8:52AM G26.00005 Fluid Mechanics of Taste, ALEXIS NOEL, NITESH BHATIA, TAREN CARTER, DAVID HU, Georgia Institute of Technology — Saliva plays a key role in digestion, speech and tactile sensation. Lack of saliva, also known as dry mouth syndrome, increases risk of tooth decay and alters sense of taste; nearly 10% of the general population suffer from this syndrome. In this experimental study, we investigate the spreading of water drops on wet and dry tongues of pigs and cows. We find that drops spread faster on a wet tongue than a dry tongue. We rationalize the spreading rate by consideration of the tongue microstructure, such as as papillae, in promoting wicking. By investigating how tongue microstructure affects spreading of fluids, we may begin to how understand taste receptors are activated by eating and drinking.

9:05AM G26.00006 Liquid-feeding strategy of the proboscis of butterflies1, SEUNG CHUL LEE, SANG JOON LEE, Pohang Univ of Sci & Tech, CENTER FOR BIOFLUID AND BIOMIMIC RESEARCH TEAM — The liquid-feeding strategy of the proboscis of butterflies was experimentally investigated. Firstly, the liquid uptake from a pool by the proboscis of a nectar-feeding butterfly, cabbage white (Pieris rapae) was tested. Liquid-intake flow phenomenon at the submerged proboscis was visualized by micro-particle image velocimetry. The periodic liquid-feeding flow is induced by the systaltic motion of the cibarial pump. Reynolds number and Womersley number of the liquid-intake flow in the proboscis are low enough to assume quasi-steady laminar flow. Next, the liquid feeding from wet surfaces by the brush-tipped proboscis of a nymphalid butterfly, Asian comma (Polygonia c-aureum) was investigated. The tip of the proboscis was observed especially brush-like sensilla styloconica. The liquid-feeding flow between the proboscis and wet surfaces was also quantitatively visualized. During liquid drinking from the wet surface, the sensilla styloconica enhance liquid intake rate with accumulation of liquid.

1This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP) (No. 2008-0061901).

9:18AM G26.00007 Modeling of Transient Nectar Flow in Hummingbird Tongues, ALEJANDRO RICO-GUEVARA, TAI-HSI FAN, MARGARET RUBEGA, University of Connecticut — We demonstrate that hummingbirds do not pick up floral nectar via capillary action. The long believed capillary rise models were mistaken and unable to predict the dynamic nectar intake process. Instead, hummingbird’s tongue acts as an elastic micropump. Nectar is drawn into the tongue grooves during elastic expansion after the grooves are squeezed flat by the beak. The new model is compared with experimental data from high-speed videos of 18 species and tens of individuals of wild hummingbirds. Self-similarity and transitions of short-to-long time behaviours have been resolved for the nectar flow driven by expansive filling. The transient dynamics is characterized by the relative contributions of negative excess pressure and the apparent area modulus of the tongue grooves.

9:31AM G26.00008 How dogs lap: open pumping driven by acceleration, SEAN GART, JOHN SOCHA, Virginia Tech, PAVLOS VLACHOS, Purdue University, SUNGHWAN JUNG, Virginia Tech — Dogs drink by lapping because they have incomplete cheeks and cannot suck fluids into the mouth. When lapping, a dog’s tongue pulls a liquid column from a bath, which is then swallowed, suggesting that the hydrodynamics of column formation are critical to understanding how dogs drink. We measured the kinematics of lapping from nineteen dogs and used the results to generate a physical model of the tongue’s interaction with the air-fluid interface. Through the use of an accelerating rod help to how dogs exploit the fluid dynamics of the column generation. The results suggest that effects of acceleration govern lapping frequency, and that dogs curl the tongue ventrally (backwards) and time their bite on the column to increase fluid intake per lap. Comparing lapping in dogs and cats reveals that though they both lap with the same frequency scaling with respect to body mass and have similar morphology, these carnivores lap in different physical regimes: a high-acceleration regime for dogs and a low-acceleration regime for cats.

9:44AM G26.00009 A musculo-mechanical model of esophageal transport based on an immersed boundary-finite element approach1, WENJUN KOU, Theoretical and Applied Mechanics Program, Northwestern University, BOYCE E. GRIFFITH, Departments of Mathematics and Biomedical Engineering, University of North Carolina at Chapel Hill, JOHN E. PANDOLFINO, PETER J. KAHRLAS, Feinberg School of Medicine, Northwestern University, NEELESH A. PATANKAR, Department of Mechanical Engineering, Northwestern University — This work extends a fiber-based immersed boundary (IB) model of esophageal transport by incorporating a continuum model of the deformable esophageal wall. The continuum-based esophagus model adopts finite element approach that is capable of describing more complex and realistic material properties and geometries. The leakage from mismatch between Lagrangian and Eulerian meshes resulting from large deformations of the esophageal wall is avoided by careful choice of interaction points. The IB approach that is capable of describing more complex and realistic material properties and geometries. The leakage from mismatch between Lagrangian and Eulerian meshes resulting from large deformations of the esophageal wall is avoided by careful choice of interaction points. The IB approach is then used to form the esophageal transport model. Cases of esophageal transport with different esophagus models are analyzed and compared.

1Support from NIH grant R01 DK56033 and R01 DK079902 is gratefully acknowledged. BEG is supported by NSF award ACI 1460334.

Monday, November 23, 2015 8:00AM - 10:10AM – Session G27 Biofluids: Propulsion: Interactions, Wakes and Jets  308 - Leah Mendelson, MIT

8:00AM G27.00001 Application of a discretized vortex impulse framework to fish maneuvering, LEAH MENDELSOHN, ALEXANDRA TECHET, MIT — In studies of biological propulsion, metrics for quantitative analysis of the vortex wake, including circulation, impulse, and their time derivatives, are a valuable indicator of performance. To better utilize volumetric PIV data in this type of analysis, a discretized method of deriving vortex impulse relying only on velocity data is developed. The impulse formulation is based on the geometry and distribution of circulation along the vortex core line, which can be detected using critical points in the velocity field. This analysis method is then applied to time-resolved velocity data of a turning giant danio (Devario aequipinnatus) and a jumping archer fish (Toctes microlopiis) obtained using Synthetic Aperture PIV (SAPIV). In the case of the danio, the vortex force vector derived from the impulse derivative shows good agreement with the kinematics of the fish tail during the turning maneuver. With the archer fish, the model is used to explore the relationship between the number of tail beats prior to the jump and the jump height.
8:13AM G27.00002 Experimental study of Strouhal number effects on the wake produced by a trapezoidal pitching panel1. JUSTIN KING, MELISSA GREEN, Syracuse University — Stereoscopic particle image velocimetry (PIV) was used to characterize the highly three-dimensional wake produced by a rigid, bio-inspired trapezoidal pitching panel. Previous work has demonstrated that one of the dominant parameters governing the wake structure of a pitching panel is the Strouhal number, and detailed analysis in terms of Strouhal number is the focus of the current work. Experiments were conducted over a range of Strouhal numbers from 0.17 to 0.56, and PIV data were collected in 55 planes across the spanwise extent of the wake. Examination of the spanwise vorticities and spanwise velocities found in the wake allow for an investigation into wake breakdown and spanwise wake compression behaviors. The results showed that increases in Strouhal number were associated with the movement of the wake breakdown location upstream and greater spanwise compression of the wake.

1This work was supported by the Office of Naval Research under ONR Award No. N00014-14-1-0418

8:26AM G27.00003 ABSTRACT WITHDRAWN —

8:39AM G27.00004 Synchronized Swimming of Two Fish1. PETROS KOUMOUTSAKOS, GUIDO NOVATI, GABRIELE ABBATI, ETH Zurich, Computational Science, Switzerland, BABAK HEJAZIALHOSSEINI, Cascade Technologies, USA, WIM VAN REES, School of Engineering and Applied Sciences, Harvard University, USA — We present simulations of two, self-propelled, fish-like swimmers that perform synchronized moves in a two-dimensional, viscous fluid. The swimmers learn to coordinate by receiving a reward for their synchronized actions. We analyze the swimming patterns emerging for different rewards in terms of their hydrodynamic efficiency and artistic impression.

1European Research Council (ERC) Advanced Investigator Award (No. 2-73985-14).

8:52AM G27.00005 A dynamical system for interacting flapping swimmers1. ANAND OZA, SOPHIE RAMANANARIVO, LEIF RISTROPH, MICHAEL SHELLEY, New York University, Courant Institute — We present the results of a theoretical investigation into the dynamics of interacting flapping swimmers. Our study is motivated by the recent experiments of Becker et al. who studied a one-dimensional array of self-propelled flapping wings that swim within each other’s wakes in a water tank. They discovered that the system adopts certain “schooling modes” characterized by specific spatial phase relationships between swimmers. To rationalize these phenomena, we develop a discrete dynamical system in which the swimmers are modeled as heaving airfoils that shed point vortices during each flapping cycle. We then apply our model to recent experiments in the Applied Math Lab, in which two tandem flapping airfoils are free to choose both their speed and relative positions. We expect that our model may be used to understand how schooling behavior is influenced by hydrodynamics in more general contexts.

1Thanks to the NSF for its support.

9:05AM G27.00006 Vortex interaction between two tandem flexible propulsors1. SUNG GOON PARK, HYUNG JIN SUNG, Department of Mechanical Engineering, KAIST, FLOW CONTROL LABORATORY TEAM — Schooling behaviors of flying and swimming animals are widespread phenomena in nature. Inspired by schooling behaviors of swimming jellyfish, self-propelling flexible bodies with a paddling-based locomotion were modeled in a tandem configuration. The interactions between surrounding fluids and propulsors were considered by using the immersed boundary method. The hydrodynamic patterns generated by the interactions between tandem flexible propulsors were analyzed in the present study. As a result of the flow-mediated interactions between them, stable configurations were formed spontaneously in which the gap distance between propulsors increased and decreased during the contraction and relaxation phases of the upstream propulsor. The stable configuration was not affected by the initial gap distance but influenced by the phase difference in the flapping frequency between them. Both tandem propulsors benefited from the tandem configuration in terms of the locomotion as compared with an isolated propulsor.

1This study was supported by the Creative Research Initiatives (No. 2015-001828) program of the National Research Foundation of Korea (MSIP).

9:18AM G27.00007 Locomotion gaits of a rotating cylinder pair1. WIM M. VAN REES, School of Engineering and Applied Sciences, Harvard University, GUIDO NOVATI, PETROS KOUMOUTSAKOS, Chair of Computational Science and Engineering, ETH Zurich, L MAHADEVAN, School of Engineering and Applied Sciences, Harvard University — Using 2D numerical simulations of the Navier-Stokes equations, we demonstrate that a simple pair of rotating cylinders can display a range of locomotion patterns of biological and engineering interest. Steadily counter-rotating the cylinders causes the pair to move akin to a vortex dipole for low rotation rates, but as the rotational velocity is increased the direction of motion reverses. Unsteady rotations lead to different locomotion gaits that resemble jellyfish (for in-phase rotations) and undulating swimmers (for out-of-phase rotations). The small number of parameters for this simple system allows us to systematically map the phase space of these gaits, and allows us to understand the underlying physical mechanisms using a minimal model with implications for biological locomotion and engineered analogs.

9:31AM G27.00008 Bio-inspired Propulsion with Functionally Graded Materials1. WILLIAM SCHLEICHER, Lehigh University, DANIEL FLORYAN, TYLER VAN BUREN, ALEXANDER SMITS, Priceton University, KEITH MOORED, Lehigh University, LEHIGH UNIVERSITY TEAM, PRICETON UNIVERSITY TEAM — From an engineering perspective, biological swimmers are a composite material system with varying material properties across their propulsors. These material properties govern how the swimmer’s structure interacts with its surrounding fluid. A two dimensional boundary element fluid solver is strongly coupled to a direct, implicit, geometrically non-linear structure solver to study the effects of functionally graded materials. A zeroth order functionally graded material approximation is used, where a rigid material abruptly meets a flexible material. Thrust, input power, and propulsive efficiency are studied as functions of non-dimensional frequency, reduced frequency, Strouhal number, flexion ratio, and effective stiffness. The numerical results are compared to experimental results for zero attack angle cases, building confidence in the numerical model. The results are further compared to structurally rigid materials.

1Supported by the Office of Naval Research under Program Director Dr. Bob Brizollara, MURI grant number N00014-14-1-0533.
9:44AM G27.00009 Jet vectoring through nozzle asymmetry1. CHRISS ROH, California Institute of Technology. ALEXANDROS ROSAKIS, Brown University. MORTEZA GHARIB, California Institute of Technology — Previously, we explored the functionality of a tri-leaflet anal valve of a dragonfly larva. We saw that the dragonfly larva is capable of controlling the three leaflets independently to asymmetrically open the nozzle. Such control resulted in vectoring of the jet in various directions. To further understand the effect of asymmetric nozzle orifice, we tested jet flow through circular asymmetric nozzles. We report the relationship between nozzle asymmetry and redirecting of the jet at various Reynolds numbers.

1This material is based upon work supported by the National Science Foundation under Grant No. CBET-1511414; additional support by the National Science Foundation Graduate Research Fellowship under Grant No. DGE-1144469.

9:57AM G27.00010 Multi-jet propulsion organized by clonal development in a colonial siphonophore. JOHN COSTELLO, Providence College. SEAN COLIN, Roger Williams University. BRAD GEMMELL, University of South Florida. JOHN DABIRI, Stanford University. KELLY SUTHERLAND, University of Oregon — Physonect siphonophores are colonial cnidarians that are pervasive predators in many neritic and oceanic ecosystems. Physonects employ multiple, clonal medusan individuals, termed nectophores, to propel an aggregate colony. Here we show that developmental differences between clonal nectophores of the physonect Nanomia bijuga produce a division of labor in thrust and torque production that controls direction and magnitude of whole colony swimming. Although smaller and less powerful, the position of young nectophores near the apex of the nectosome allows them to dominate torque production for turning whereas older, larger and more powerful individuals near the base of the nectosome contribute predominantly to forward thrust production. The patterns we describe offer insight into the biomechanical success of an ecologically important and widespread colonial animal group, but more broadly, provide basic physical understanding of a natural solution to multi-engine organization that may contribute to the expanding field of underwater distributed propulsion vehicle design.

Monday, November 23, 2015 8:00AM - 10:10AM –
Session G28 Surface Tension Effects: Particles at Interfaces 309 - Shreyas Mandre, Brown University

8:00AM G28.00001 Hydrodynamics of a fixed camphor boat at the air-water interface1. DHIRAJ SINGH, SATHISH AKELLA, Okinawa Institute of Science and Technology. RAVI SINGH, SHREYAS MANDRE, Brown University. MAHESH BANDI, Okinawa Institute of Science and Technology — A camphor tablet, when introduced at the air-water interface undergoes sublimation and the camphor vapour spreads radially outwards across the surface. This radial spreading of camphor is due to Marangoni forces setup by the camphor concentration gradient. We report experiments on the hydrodynamics of this process for a camphor tablet held fixed at the air-water interface. During the initial transient, the time-dependent spread radius $R(t)$ of camphor scales algebraically with time $t$ ($R(t) \propto t^{1/2}$) in agreement with empirical scalings reported for spreading of volatile oils on water surface. But unlike surfactants, the camphor stops spreading when the influx of camphor from the tablet onto the air-water interface is balanced by the outflux of camphor due to evaporation, and a steady-state condition is reached. The spreading camphor however, shears the underlying fluid and sets up bulk convective flow. We explain the coupled steady-state dynamics between the interfacial camphor spreading and bulk convective flow with a boundary layer approximation, supported by experimental evidence.

1This work was supported by the Collective Interactions Unit, OIST Graduate University

8:13AM G28.00002 Hydrodynamics of a self-propelled camphor boat at the air-water interface1. SATISH AKELLA, DHIRAJ SINGH, Okinawa Institute of Science and Technology. RAVI SINGH, Brown University. MAHESH BANDI, Okinawa Institute of Science and Technology — A camphor tablet, when placed at the air-water interface undergoes sublimation and camphor vapour spreads radially outwards across the surface due to Marangoni forces. This steady camphor influx from tablet onto the air-water interface is balanced by the camphor outflux due to evaporation. When spontaneous fluctuations in evaporation break the axial symmetry of Marangoni force acting radially outwards, the camphor tablet is propelled like a boat along the water surface. We report experiments on the hydrodynamics of a self-propelled camphor boat at air-water interfaces. We observe three different modes of motion, namely continuous, harmonic and periodic, due to the volatile nature of camphor. We explain these modes in terms of ratio of two time-scales: the time-scale over which viscous forces are dominant over the Marangoni forces ($\tau_\eta$) and the time-scale over which Marangoni forces are dominant over the viscous forces ($\tau_M$). The continuous, harmonic and periodic motions are observed when $\tau_M/\tau_\eta \lesssim 1$. $\tau_\eta/\tau_M \geq 1$ and $\tau_M/\tau_\eta \gg 1$ respectively. Experimentally, the ratio of the time scales is varied by changing the interfacial tension of the air-water interface using Sodium Dodecyl Sulfate.

1This work was supported by the Collective Interactions Unit, OIST Graduate University

8:26AM G28.00003 PIV measurements of the streaming fluid flow due to the adsorption of particles1. HARSH PATEL, NAGA MUSUNURI, EDISON AMAH, IAN FISCHER, PUSHPENDRA SINGH, NJIT — The particle image velocimetry (PIV) technique is used to study the streaming flow that is induced when a spherical particle is adsorbed at a liquid surface. The flow causes powders sprinkled on a liquid surface to disperse on the surface. The dispersion can occur so quickly that it appears explosive, especially for small particles on the surface of mobile liquids like water. The measurements show that the adsorption of a spherical particle causes an axisymmetric streaming flow about the vertical line passing through the center of the particle. The fluid directly below the particle rises upward, and near the surface, it moves away from the particle. The flow, which develops within a fraction of second after the adsorption of the particle, persists for several seconds. The flow strength, and the volume over which it extends, decreases with decreasing particle size. The streaming flow induced by the adsorption of two or more particles is a combination of the flows which they induce individually.

1The work was supported by National Science Foundation
8:39AM G28.00004 Particle-laden interfaces: direct calculation of interfacial stress from a discrete particle simulation of a pendant drop1, CHUAN GU, LORENZO BOTTO, Queen Mary University of London — The adsorption of solid particles to fluid interfaces is exploited in several multiphase flow technologies, and plays a fundamental role in the dynamics of particle-laden drops. A fundamental question is how the particles modify the effective mechanical properties of the interface. Using a fast Eulerian-Lagrangian model for interfacial colloids, we have simulated a pendant drop whose surface is covered with spherical particles having short-range repulsion. The interface curvature induces non-uniform and anisotropic interfacial stresses, which we calculate by an interfacial extension of the Irving-Kirkwood formula. The isotropic component of this stress, related to the effective surface tension, is in good agreement with that calculated by fitting the drop shape to the Young-Laplace equation. The anisotropic component, related to the interfacial shear elasticity, is highly non uniform: small at the drop apex, significant along the drop sides. The reduction in surface tension can be substantial even below maximum surface packing. We illustrate this point by simulating phase-coarsening of a two-phase mixture in which the presence of interfacial particles “freezes” the coarsening process, for surface coverage well below maximum packing.

1This work is supported by the EU through the Marie Curie Grant FLOWMAT (618335)

8:52AM G28.00005 Marangoni-Driven Flow Oscillations during the Dissolution of Surfactant Powders, OREST SHARDT, HYOUNGSOO KIM, HASSAN MASOUD, HOWARD STONE, Princeton University — When particles of surface-active compounds are deposited on a liquid surface, they exhibit a variety of motions, such as the classic erratic movement of camphor on water. We investigate the unsteady motion of a water surface covered with surfactant particles and find that a rapid longitudinal oscillation occurs as the particles dissolve. This phenomenon happens with several common surfactant powders, but it is particularly striking with calcium propionate, an organic salt that decreases the surface tension of water. We examine the effects of several parameters on the characteristics of the oscillation by using particle image velocimetry (PIV). Due to the short period of the oscillation (of the order of 0.1 s) compared to the timescale of surfactant diffusion, we neglect diffusion and model this phenomenon by considering the evolution of variations in surfactant concentration along a liquid surface. This surfactant concentration is advected by the flow that is driven by Marangoni stress due to non-uniform surfactant and therefore surface tension distributions. We examine the critical conditions for and characteristics of the oscillation in this model through theory and simulations.

9:05AM G28.00006 Evaporation effects in elastocapillary aggregation, DOMINIC VELLA, ANDREAS HADJITTOFIS, KIRAN SINGH, Mathematical Institute, University of Oxford, JOHN LISTER, DAMTP, University of Cambridge — We consider the effect of evaporation on the aggregation of a number of elastic objects due to a liquid’s surface tension. In particular, we consider an array of spring–block elements in which the gaps between blocks are filled by thin liquid films that evaporate during the course of an experiment. Using lubrication theory to account for the fluid flow within the gaps, we study the dynamics of aggregation. We find that a non-zero evaporation rate causes the elements to aggregate more quickly and, indeed, to contact within finite time. However, we also show that the number of elements within each cluster decreases as the evaporation rate increases. We explain these results quantitatively by comparison with the corresponding two-body problem and discuss their relevance for controlling pattern formation in carbon nanotube forests.

9:18AM G28.00007 Elastocapillary-mediated interfacial assembly, ARTHUR EVANS, University of Wisconsin-Madison — Particles confined to an interface are present in a large number of industrial applications and ubiquitous in cellular biophysics. Interactions mediated by the interface, such as capillary effects in the presence of surface tension, give rise to rafts and aggregates whose structure is ultimately determined by geometric characteristics of these adsorbed particles. A common strategy for assembling interfacial structures relies on exploiting these interactions by tuning particle anisotropy: either by constructing rigid particles with heterogeneous wetting properties or fabricating particles that have a naturally anisotropic shape. Less explored, however, is the scenario where the interface causes the particles to deform. In this talk I will discuss the implications for interfacial assembly using elastocapillary-mediated interactions. The competition between surface energy and elasticity can wrinkle and buckle adsorbed soft particles, leading to complicated (but programmable) aggregates.

9:31AM G28.00008 Stokesian Dynamic Simulations of Colloid Assembly at a Fluid Interface, ARCHIT DANI, CHARLES MALDARELLI, City College of New York — The collective dynamics and self-assembly of colloids floating at a gas/liquid or a liquid/liquid interface is a balance between deterministic lateral interaction forces, e.g. capillary attraction and dipolar electrostatic repulsion if the particles are charged, viscous resistance to colloid motion along the surface and thermal fluctuations. As the colloid size decreases, thermal (Brownian) forces become important and can affect the self assembly into ordered patterns and crystal structures that are the starting point for materials applications. Stokesian dynamics simulations are presented to describe the lateral organization of particles along the surface in Brownian dominated regimes that includes (using a pairwise approximation) capillary attraction and the hydrodynamic interaction between particles (incorporating the effect of the particle immersion depth) and thermal fluctuations. Clustering, fractal growth and particle ordering are observed at critically large values of the Peclet numbers, while smaller values yield states in which particles remain uncorrelated in space and more widely separated.

9:44AM G28.00009 Tilting and tumbling of Janus nanoparticles at sheared interfaces, SHAHAB SHOJAEI-ZADEH, HOSSEIN REZVANTLAB, Rutgers University — We use molecular dynamics simulations to investigate the response of individual Janus nanoparticles adsorbed at liquid-fluid interfaces to imposed symmetric shear. Depending on particle shape, amphiphilicity, and the applied shear rate, two distinct rotational dynamics toward a steady particle orientation are observed: smooth tilting, and tumbling. Particles adopt a steady orientation when imposing shear-induced torque is balanced with the opposing capillary-induced one. The dynamics can be perhaps determined based on the free energy of different states relative to the thermal energy. We construct phase diagrams correlating particle dynamics to shear rate, hydrophobicity, and shape. Our results have direct implication on shear-induced alignment and assembly of Janus particles at fluid interfaces.

9:57AM G28.00010 Surface Tension Mediated Under-Water Adhesion of Rigid Spheres on Soft, Charged Surfaces, SHAYANDEV SINHA, SIDDHARTHA DAS, Univ of Maryland-College Park — Understanding the phenomenon of surface-tension-mediated under-water adhesion is necessary for studying a plethora of physiological and technical phenomena, such as the uptake of bacteria or nanoparticle by cells, attachment of virus on bacterial surfaces, biofouling on large ocean vessels and marine devices, etc. This adhesion phenomenon becomes highly non-trivial in case the soft surface where the adhesion occurs is also charged. Here we propose a theory for analyzing such an under-water adhesion of a rigid sphere on a soft, charged surface, represented by a grafted polyelectrolyte layer (PEL). We develop a model based on the minimization of free energy that, in addition to considering the elastic and the surface-tension-mediated adhesion energies, also accounts for the PEL electric double layer (EDL) induced electrostatic energies. We show that in the presence of surface charges, adhesion gets enhanced. This can be explained by the fact that the increase in the elastic energy is better balanced by the lowering of the EDL energy associated with the adhesion process. The entire behaviour is further dictated by the surface tension components that govern the adhesion energy.
We perform a combined study of the Koopman modes and eigenvalues involved with each category. It will be shown that projecting onto a low order subspace of Koopman modes can capture features supported by normal forms. This behavior can be critical for systems of practical significance, such as oscillating airfoils under dynamic stall. In this study, normal form theory and spectral decompositions based on Koopman operators will be used to reveal transitions from limit cycle attractors to chaotic/quasi-periodic dynamics in the cylinder Hopf bifurcation flow. Koopman operator methods are used since each mode is associated with a single frequency which allows one to observe the evolution to more continuous spectral behavior with forcing, while approaches such as proper orthogonal decomposition may obfuscate this unfolding of singularity and bifurcation diagram.

8:26AM G29.00003 Bifurcation Analysis of 1D Steady States of the Bénard Problem in the Long Wavelength Limit  

CHENGHZE ZHOU, SANDRA TROIAN, California Institute of Technology, MC 128-95, Pasadena, CA 91125 — We investigate the character and stability of stationary states of the (1 + 1)D evolution equation \( \partial_t h + (h^3 h_{xxx} + h^2 \partial_x \gamma) = 0 \) describing the motion of an interface \( h(x,t) \) separating a thin warm viscous film from a thin cool inviscid layer where \( \gamma = \gamma(h) \) represents the interfacial tension. The phase diagram corresponding to all positive periodic steady states (PPSS) is specified by two variables - the global extrema of the equilibrium shape and a generalized mechanical interface pressure. The analytic forms describing the PPSS shapes, the minimal period, the average height and the generalized free energy are all confirmed numerically. We find there is at most one non-trivial PPSS for specified period and volume. We also find no stable perturbed PPSS near the critical point for volume conserving perturbations of identical period. A weakly non-linear analysis about the critical point yields bifurcations of the pitchfork-type. For all non-trivial PPSS, we verify the unstable nature of the PPSS by transforming the non-normal operator (resulting from the spatially inhomogeneous PPSS) to normal form, which we then solve by finite element computations.

8:39AM G29.00004 Local analysis and topological bifurcations of free surfaces1.  

NUGAR SURAMLISHVILI, JENS EGGERS, School of Mathematics, University of Bristol — The recently developed method of the local analyst is applied in order to determine generic singularities and topological bifurcations of free surfaces and interfaces in liquids and gases. In the presented physical examples the surface geometry is described by a family of smooth maps with the state variables and a set of control parameters. We determine the order of Taylor expansion about a point of interest defining the multiplicity of a singular germ, codimension, unfolding of singularity and bifurcation diagram.

1This work was supported by the Leverhulme Trust Grant


8:52AM G29.00005 bifurcations of driven cavity flow and their implications for mixing  

HASAN ARBAB, IGOR MEZIC, University of California, Santa Barbara — We use the Koopman operator theory to study the sequence of bifurcations in 2D driven cavity flow. By extracting the Koopman modes (analogous to normal modes of linear-systems theory), we identify the dominant flow structures at different subranges of Reynolds number. We also use the eigenvalues of the Koopman operator to study the structural dependence of the flow on the Reynolds number and classify the asymptotic state of the unsteady flow into periodic, quasi-periodic and possibly chaotic categories. Ultimately, we perform a combined study of the Koopman modes and eigenvalues involved with each category to study the effect of each bifurcation on mixing properties of the cavity flow.

9:05AM G29.00006 Koopman decompositions of periodically forced Hopf bifurcation flows and application to dynamic stall1.  

BRAD GLAZ, U.S. Army Research Laboratory, SOPHIE LOIRE, MARIA FONOBEROVA, Amdyn, Inc., IGOR MEZIC, University of California Santa Barbara — Periodically forced Hopf bifurcation flows, such as oscillating cylinders, can exhibit rich spectral content. Though lock-on dynamics of systems forced near resonances are well understood, the underlying chaotic or quasi-periodic dynamics when forcing away from a natural frequency are not. This behavior can be critical for systems of practical significance, such as oscillating airfoils under dynamic stall. In this study, normal form theory and spectral decompositions based on Koopman operators will be used to reveal transitions from limit cycle attractors to chaotic/quasi-periodic dynamics in the cylinder Hopf bifurcation flow. Koopman operator methods are used since each mode is associated with a single frequency which allows one to observe the evolution to more continuous spectral behavior with forcing, while approaches such as proper orthogonal decomposition may obfuscate this transition. It will be shown that projecting onto a low order subspace of Koopman modes can capture features supported by normal forms. Using this, we show a mechanism that leads to regimes in which the system seems to exhibit shear-induced chaos. The new framework is applied to dynamic stall studies to establish periodically forced Hopf bifurcation dynamics as an underlying feature.

1Support for M. Fonoberova, S. Loire, and I. Mezic under U.S. Army Research Office project W911NF-14-C-0102, and U.S. Air Force Office of Scientific Research project FA9550-12-1-0230 are acknowledged.
submesoscale instabilities. We use very high resolution direct numerical simulations (DNS) to explore the interaction and feedbacks between submesoscales and small scales where the earth's rotation plays a major role, and turbulent overturning scales (1-10 km) where the earth’s rotation has a major role, and turbulent overturning scales (100 km) where the earth’s rotation plays a major role. By using both analytical approach and numerical simulation we show that there exists a sub-class of models with elongated shell interactions that exhibits a statistically stable inverse energy cascade. Using also data from direct numerical simulations of helical Navier-Stokes equations we further support the idea that energy transfer mechanism in fully developed turbulence is the result of a strong entanglement among different triadic interactions possessing different transfer mechanisms. Using Particle-Image Velocimetry, and examine the wake dynamics throughout the response regime in terms of the phase-averaged vorticity fields. The Finite-Time Lyapunov exponent (FTLE) fields are also computed in backward- and forward-time in order to identify the Lagrangian Coherent Structures. We examine four distinct wake modes that occur at various points in the response regime. The roll up of the shear layers and the vortex formation process are examined using the FTLE fields. This analysis allows the fluid-structure interaction and dynamics in the near wake to be examined in much greater detail than would be possible using the vorticity fields alone. Particular attention is paid to the symmetric vortex-shedding mode, which is characteristic to streamwise VIV; the forward-time FTLE fields show that the wake is organised into discrete “vortex cells,” which enclose each vortex and define its boundary. Finally, the advection of tracers in the wake is studied in order to examine how the different wake modes promote/inhibit mixing. The alternate wake modes tend to promote mixing, particularly in the second response branch, but the symmetric shedding tends to reduce the lateral mixing across the wake.

In the 2-d Navier-Stokes Equations with spatially periodic and temporally steady forcing is addressed. At first, the Fourier Galerkin method is applied to the 2-d N-S equations to obtain a fifth order system of nonlinear ordinary differential equations (ODE) that approximates the behavior of these equations. The alternate wake modes tend to promote mixing, particularly in the second response branch, but the symmetric shedding tends to reduce the lateral mixing across the wake.

9:44AM G29.00009 Chaotic advection in 2D anisotropic porous media1. STEPHEN VARGHESE, MICHEL SPEETJENS, RUBEN TRIELING, FREDERICO TOSCHI, Eindhoven University of Technology — Traditional methods for heat recovery from underground geothermal reservoirs employ a static system of injector-producer wells. Recent studies in literature have shown that using a well-devised pumping scheme, through actuation of multiple injector-producer wells, can dramatically enhance production rates due to the increased scalar / heat transport by means of chaotic advection. However the effect of reservoir anisotropy on kinematic mixing and heat transport is unknown and has to be incorporated and studied for practical deployment in the field. As a first step, we numerically investigate the effect of anisotropy (both magnitude and direction) on (chaotic) advection of passive tracers in a time-periodic Darcy flow within a 2D circular domain driven by periodically reoriented diametrically opposite source-sink pairs. Preliminary results indicate that anisotropy has a significant impact on the location, shape and size of coherent structures in the Poincare sections. This implies that the optimal operating parameters (well spacing, time period of well actuation) may vary strongly and must be carefully chosen so as to enhance subsurface transport.

1This work is part of the research program of the Foundation for Fundamental Research on Matter (FOM), which is part of Netherlands Organisation for Scientific Research (NWO). This research program is co-financed by Shell Global Solutions International B.V.

9:57AM G29.00010 Controlling the Dynamics of the Five-Mode Truncation System of the 2-d Navier-Stokes Equations1. NEJIB SMAOUI, MOHAMED ZRIBI, Kuwait University — The dynamics and the control problem of the two dimensional 2-d) Navier-Stokes (N-S) equations with spatially periodic and temporally steady forcing is addressed. At first, the Fourier Galerkin method is applied to the 2-d N-S equations to obtain a fifth order system of nonlinear ordinary differential equations (ODE) that approximates the behavior of these equations. The alternate wake modes tend to promote mixing, particularly in the second response branch, but the symmetric shedding tends to reduce the lateral mixing across the wake.

This work was supported and funded by the Research Sector, Kuwait University under Grant No. SM 05/15
Further, a break is seen in the total energy (TE) spectrum at small scales with a transition from $k^{-3}$ to $k^{-5/3}$ reminiscent of the atmospheric spectra of Nastrom & Gage. For details see JFM 756, 965-1006.

8:26AM G30.00003 Submesoscale baroclinic instability and the Balance Equations. IAN GROOMS, University of Colorado, Boulder — Ocean submesoscale baroclinic instability is studied in the framework of the Balance Equations. The Balance Equations are an intermediate model that includes balanced ageostrophic effects with higher accuracy than the quasigeostrophic approximation, but rules out unbalanced wave motions; as such, they are particularly suited to the study of baroclinic instability in submesoscale ocean dynamics. The linear baroclinic Richardson number problem is developed in generality and then specialized to the case of constant vertical shear. The primary finding is that at low Richardson numbers the growth rate of some instability modes is increased compared to larger-scale quasigeostrophic dynamics, and that the increase can be attributed to both ageostrophic baroclinic production and shear production of perturbation energy. This suggests that the nonlinear development of submesoscale baroclinic instability will proceed more vigorously than mesoscale/quasigeostrophic, and may include a downscale/forward transfer of kinetic energy.

8:39AM G30.00004 Coupled evolution of near-inertial waves and quasigeostrophic flow\(^1\), GREGORY WAGNER, University of California, San Diego, WILLIAM YOUNG, Scripps Institution of Oceanography — We derive a model describing the coupled nonlinear evolution of three fields: near-inertial wave (NIW) amplitude, quasigeostrophic potential vorticity, and the NIW second harmonic. The model is derived by asymptotic reduction of the Boussinesq equations using the method of multiple scales. The model conserves two distinct quantities: wave action, and coupled energy. Wave action conservation implies energy exchange between NIW kinetic energy and energy in the NIW second harmonic. Coupled energy conservation implies energy exchange between NIW potential energy and quasigeostrophic flow. We find that the impulsive solutions of the model with two dimensional numerical solutions meant to approximate NIW evolution in non-uniform quasigeostrophic flow following storm-driven excitation. For this scenario we find good agreement between the model solutions and the full Boussinesq equations. Preliminary results show the initial transient evolution of the NIW field extracts energy from the quasigeostrophic flow. Further, the quasigeostrophic flow catalyzes an interaction between the NIW and the NIW second harmonic which ultimately leads to the generation of small NIW vertical scales.

\(^1\)We thank the National Science Foundation for support under OCE-1357047

8:52AM G30.00005 Lagrangian and Eulerian Statistics of Vorticity Dynamics in Turbulent Stratified Shear Flows. FRANK JACOBITZ, University of San Diego, KAI SCHNEIDER, Aix-Marseille Universit, MARIE FARGE, Ecole Normale Supérieure — The Lagrangian and Eulerian time-rate of change statistics of vorticity in homogeneous turbulence with shear and stable stratification are studied. Direct numerical simulations are performed, in which the Richardson number is varied from $Ri=0$, corresponding to unstratified shear flow, to $Ri=1$, corresponding to strongly stratified shear flow. The probability density functions (pdfs) of both Lagrangian and Eulerian time-rates of change show a strong influence on the Richardson number. The Lagrangian time-rate of change pdf has a stretched-exponential shape due to the vortex stretching and tilting term in the equation for fluctuating vorticity. The shape of the Eulerian time-rate of change pdf changes more slowly with the change in the Richardson number compared to the Eulerian time-rate of change pdf which changes more rapidly with the change in the Richardson number. The probability density function for the Eulerian time-rate of change pdf is less sensitive to the change in the Richardson number. The Lagrangian time-rate of change pdf of fluctuating density does not show a stretched exponential shape, while its Eulerian counterpart does due to the change in the in the density advection-diffusion equation.

9:05AM G30.00006 Flow dynamics at a river confluence on Mississippi River: field measurement and large eddy simulation\(^1\), TRUNG LE, ALI KHOSROnejad, Saint Anthony Falls Laboratory, University of Minnesota, NICOLE BARTELT, SOLOMON WOLDEAMLAK, BONNIE PETERSON, PETRONELLA DEWALL, Minnesota Department of Transportation, FOTIS SOTIROPOULOS, Saint Anthony Falls Laboratory, University of Minnesota, SAINT ANTHONY FALLS LABORATORY, UNIVERSITY OF MINNESOTA TEAM, MINNESOTA DEPARTMENT OF TRANSPORTATION TEAM — We study the dynamics of a river confluence on Mississippi River branch in the city of Minneapolis, Minnesota, United States. Field measurements by Acoustic Doppler Current Profiler using on-board GPS tracking were carried out for five campaigns in the summer of 2014 and 2015 to collect both river bed elevation data and flow fields. Large Eddy Simulation is carried out to simulate the flow field with the total of 100 million grid points for the domain length of 3.2 km. The simulation results agree well with field measurements at measured cross-sections. The results show the existence of wake mode on the mixing interface of two branches near the upstream junction corner. The mutual interaction between the shear layers emanating from the river banks leading to the formation of large scale energetic structures that leads to "switching" side of the flow coherent structures. Our result here is a feasibility study for the use of eddy-resolving simulations in predicting complex flow dynamics in medium-size natural rivers.

\(^1\)This work is funded by Minnesota Dept. Transportation and Minnesota Institute of Supercomputing

9:18AM G30.00007 Anisotropic shear dispersion parameterization for ocean eddy transport, SCOTT RECKINGER, Montana State University, BAYLOR FOX-KEMPER, Brown University — The effects of mesoscale eddies are universally treated isotropically in global ocean general circulation models. However, observations and simulations demonstrate that the mesoscale processes that the parameterization is intended to represent, such as shear dispersion, are typified by strong anisotropy. We extend the Gent-McWilliams/Redi mesoscale eddy parameterization to include anisotropy and test the effects of varying levels of anisotropy in 1-degree Community Earth System Model (CESM) simulations. Anisotropy has many effects on the simulated climate, including a reduction of temperature and salinity biases, a downscaling of the southern ocean mixed-layer depth, impacts on the meridional overturning circulation and ocean energy and tracer uptake, and improved ventilation of biogeochemical tracers, particularly in oxygen minimum zones. A process-based parameterization to approximate the effects of unresolved shear dispersion is also used to set the strength and direction of anisotropy. The shear dispersion parameterization is similar to drifter observations in spatial distribution of diffusivity and high-resolution model diagnosis in the distribution of eddy flux orientation.
9:31AM G30.00008 Mesoscale Ocean Large Eddy Simulations, BRODIE PEARSON, BAYLOR FOX-KEMPER, SCOTT BACHMAN, FRANK BRYAN, Brown University — The highest resolution global climate models (GCMs) can now resolve the largest scales of mesoscale dynamics in the ocean. This has the potential to increase the fidelity of GCMs. However, the effects of the smallest, unresolved, scales of mesoscale dynamics must still be parameterized. One such family of parameterizations are mesoscale ocean large eddy simulations (MOLES), but the effects of including MOLES in a GCM are not well understood. In this presentation, several MOLES schemes are implemented in a mesoscale-resolving GCM (CESM), and the resulting flow is compared with that produced by more traditional sub-grid parameterizations. Large eddy simulation (LES) is used to simulate flows where the largest scales of turbulent motion are resolved, but the smallest scales are not. LES has traditionally been used to study 3D turbulence, but recently it has also been applied to idealized 2D and quasi-geostrophic (QG) turbulence. The MOLES presented here are based on 2D and QG LES schemes.

9:44AM G30.00009 SOMAR-LES for multiscale modeling of internal tide generation, VAMSI KRISHNA CHALAMALLA, University of California San Diego, EDWARD SANTILLI, Philadelphia University, MASOUD JALALI, University of California San Diego, ALBERTO SCOTTI, University of North Carolina, Chapel Hill, SUTANU SARKAR, University of California San Diego — A novel modeling technique is developed to study baroclinic energy conversion when the barotropic tide oscillates over underwater topography. In SOMAR-LES, a Large Eddy Simulation (LES) model that resolves turbulence scales is coupled with a large-scale model, Stratified Ocean Model with Adaptive Refinement (SOMAR). Thus, we overcome the constraints posed by the wide range of temporal and spatial scales during tide and topographic interaction. Two-way coupling is developed: LES is driven with large scale forcing, and SOMAR receives feedback in the form of eddy viscosity and diffusivity. Numerical simulations are performed with SOMAR-LES for supercritical and subcritical ridges with ridge length scales of O(10 km) and barotropic forcing that corresponds to the regime of low outer excursion number, $Ex = U_0/ωl \approx 0.1$. Results from the coupled model are compared against ongoing high-resolution LES to ascertain the accuracy of this technique. The simulation data is analyzed to quantify baroclinic energy conversion and the change in modal composition from near to far field.

9:57AM G30.00010 Baroclinic mixed layer instability in the presence of convection, JOERN CALLIES, RAFFAELE FERRARI, Massachusetts Institute of Technology — It has recently been discovered that geostrophic turbulence in the upper ocean undergoes a seasonal cycle at submesoscales, the scales smaller than the most energetic mesoscale eddies. Observations and theory suggest that baroclinic mixed layer instabilities release potential energy stored in deep mixed layers, energizing the submesoscales in winter. In shallow summer mixed layers, there is no such energization. The oceanic mixed layer, besides being prone to baroclinic instabilities, is subject to atmospheric forcing, which drives convective overturns. We here study how this forced convection interacts with baroclinic instabilities in a set of idealized numerical simulations resolving both processes. A major question is whether baroclinic instabilities can be damped out by convection. Implications for the seasonal cycle in submesoscale turbulence will be discussed.

Monday, November 23, 2015 8:00AM - 10:10AM — Session G31 Drops: Electrowetting and Charge Effects

8:00AM G31.00001 Ion adsorption-induced wetting transition in oil-water-mineral systems, BIJOYENDRA BERA, ANDREA CAVALLI, DIRK VAN DEN ENDE, FRIEDER MUGELE, Univ of Twente — The relative wettability of a rock substrate to oil and water is a central issue in many technological applications, especially in the field of enhanced oil recovery. We here consider a salty water droplet deposited on a mica substrate inside an oil bath. By adding specific ions to the water phase, the wettability of the mineral substrate can be altered. Different cations can be arranged in an Hofmeister-like sequence, based on their effectiveness in changing the wettability of the mineral substrate. This phenomenon in the scope of a Poisson-Boltzmann model. The absorption of divalent ions at the mica interface generates a positive surface charge, and induces an attractive interaction to the negatively charged oil-water interface, which triggers the transition. We also observe that different cations can be arranged in an Hofmeister-like sequence, based on their effectiveness in changing the wettability of the mineral substrate. Finally, we show that adding small amounts of a polar surfactant to the oil phase synergistically enhances the wetting transition.

8:13AM G31.00002 Liquid bridges in complex geometries: Equilibrium shape metamorphosis using electrowetting, DAVOOD BARATIAN, ANDREA CAVALLI, DIRK VAN DEN ENDE, FRIEDER MUGELE, Univ of Twente — The equilibrium morphology of liquid drops exposed to geometric constraints can be rather complex. Even for simple geometries, analytical solutions are scarce. We investigate the equilibrium shape and position of liquid drops confined in the wedge between two solid surfaces. Using electrowetting, we control the contact angle and thereby manipulate the shape and the equilibrium position of aqueous drops in ambient oil. In the absence of contact angle hysteresis and buoyancy, we find that the equilibrium shape is given by a truncated sphere, prior to filling the wedge corner, at a position that is determined by the drop volume and the contact angle. At this position, the net force between drop and the surfaces vanishes. The effect of buoyancy gives rise to substantial deviations from this equilibrium configuration which we discuss it as well. We elegantly show how the geometric constraint and electrowetting can be used to position droplets inside a wedge in a controlled way, without mechanical actuation.

8:26AM G31.00003 Spreading of thin droplets of perfect and leaky dielectric liquids on inclined surfaces, ANDREW CORBETT, BO KYUNG RYU, SHYUNG LEE, SATISH KUMAR, University of Minnesota — The spreading of droplets may be influenced by electric fields, a situation that is relevant to applications such as coating, printing, and microfluidics. In this work we study the effects of an electric field on the gravity-driven spreading of droplets down an inclined plane. We consider both perfect and leaky dielectric liquids, as well as perfectly and partially wetting systems. Lubrication theory is applied to generate a set of coupled partial differential equations for interfacial height and charge, which are then solved numerically with a finite difference method. Electric fields tend to increase the height of the capillary ridge in both perfect and leaky dielectric droplets. In partially wetting liquids, the presence of large concentrations of surface charge can cause the main droplet to split into several smaller droplets. Although the model predicts that electric fields do not significantly change the long time spreading rate, flow visualization experiments reveal that electric fields can significantly alter the dynamics of droplet spreading outside of the regime in which the model is valid.
nano-thread.

DARSHAK BHUPTANI, SARITH SATHIAN, Indian Inst of Tech-Madras — The stability behavior of nano-scale molecule orientation, whipping instability and splaying are observed. The force experienced by the surface molecules. With increase in the electrical field strength, different phenomenon such as Taylor's cone formation, and the role of thermal fluctuation on the breakup in such cases are investigated. A uniform electric field is applied along the axial direction to investigate the effect of an additional external force in the form of an electric field on the stability and breakup. The effect of surface tension accurately. At nanoscale, thermal fluctuation and surface tension forces plays a critical role in the breakup mechanism. The main objective is of 0.2 to 0.5 whereas for a macroscale it is around 0.697. The classical viscid theory predict the first breakup time of nanoscale liquid thread molecular dynamics (MD) simulations. The Rayleigh instability predicts the non-dimensional wave number for nano-thread to be in the range stationary polar liquid (water) thread in vacuum and in environment under the action of an external constant electric field is investigated using microstructures [1, 2], while previous experiments showed that the effect of surface charge on dynamic wetting on a flat surface is minor. Here, we combine microstructuring and electrowetting to further enhance the controllability of the dynamic wetting. Microstructures are fabricated on silicon wafers and the spontaneous spreading of a droplet is imaged with a high-speed camera. We reveal that the spreading rate sensitivity to surface charge increases in the presence of microstructures. Furthermore, numerical simulations solving Cahn-Hilliard/Navier-Stokes equations are performed and the effect of surface modification is quantified in terms of the contact-line friction.

Acknowledgment is made to the Donors of the American Chemical Society Petroleum Research Fund for support of this research.

1This work was financially supported in part by the Japan Science and Technology Agency through CREST


8:52AM G31.00005 Analysis of the triple contact point in electrowetting. SAM BRZEZICKI, DARREN CROWDY, ELENA LOUCA, None — In electrowetting applications, it is often necessary to consider static conducting droplets sitting on substrates. In this talk we perform a detailed study of the triple contact point at which the droplet, the ambient medium and the substrate touch. A local analysis is performed to understand the nature of the field singularities at the contact point. We also present some global solutions obtained using a novel transform formulation and gain insights into how those solutions depend on the system parameters.

9:05AM G31.00006 Oscillating dynamics of a bubble immersed in an electrical field. HERVE CAPS, JEROME HARDOUN, GRASP-University of Liège, Belgium, GRASP TEAM — From the pioneer work of Millikan, it is known that adding electrical charges in a droplet causes both volume and surface forces. The first force allowed Millikan to determine the electrical charge of the electron, while Taylor paved the way into the second effect by evidencing capillary pressure variations. In the present study, we focus on the dynamics of a hemispheric bubble (1cm in diameter) deposed onto one of the two conducting plates of a capacitor. The bubble is observed to experience periodic electrical breakdowns followed by more quiescent periods. One of the important facts is that the bubble never explodes even electrical sparks are generated. The bubble dynamics has been followed by mean of a high-speed camera and allowed us to grasp the deformations of the bubble due to the electric field, from rest to the electrical breakdown. We propose a rather simple model accounting for the experimental parameters such as the applied electric field. By balancing the surface tension, viscous and electrical forces, this model mimics the bubble dynamics and evidences the charge dynamics inside the bubble.

9:18AM G31.00007 Nano-crater Formation on Electrodes during the Electrical Charging of Aqueous Droplets. ERIC ELTON, ETHAN ROSENBERG, WILLIAM RISTENPART, Dept. Chemical Engineering, University of California Davis — A water drop in an insulating fluid acquires charge when it contacts an electrode, but the exact mechanism of charge transfer has remained obscure. Previous work, dating back to Maxwell, has implicitly assumed that the electrode remains unaltered by the charging process. Here we demonstrate that, contrary to this assumption, water drops and other conducting objects create “nano-craters” on the electrode surface during the charging process. We used optical microscopy, SEM, and atomic force microscopy to characterize the electrode surfaces before and after water drops were electrically bounced on them. We show that each drop contact creates an approximately micron wide and 30-nm deep crater to form on the electrode surface. Given enough time, the drop will form enough nano-craters to effectively ‘eat through’ a sufficiently thin electrode. We discuss possible physical mechanisms for the nano-crater formation, including localized melting caused by Joule heating during the charge transfer event. The observations reported here are of particular interest in the development of microfluidic devices that use thin film electrodes to control the motion of aqueous drops.

9:31AM G31.00008 Interfacial Charge Effects on Sticky Bubble Morphology in a Microchannel. JONATHAN HUI, PETER HUANG, Binghamton Univ — Many multiphase fluidic processes in small conduits, such as petroleum extraction and biochemical analysis, can encounter disastrous flow blockages due to the lodging of immiscible bubbles or droplets. The complete drainage of a thin-film lubrication layer surrounding an adhered bubble demands a significantly higher threshold pressure gradient in order to reinitiate bulk flows. In this work, we investigate bubble morphology due to the lubrication layer drainage process that results in bubble adhesion and study how an electrostatically charged bubble interface and charged channel wall may affect bubble morphology in preventing bubble adhesion. We report on our multiphysics computational analysis of an oversized gas bubble in a water-filled microchannel under the influence of surface tension and interfacial electrostatic forces.

Acknowledgment is made to the Donors of the American Chemical Society Petroleum Research Fund for support of this research.

9:44AM G31.00009 The effect of electric field on the stability and breakup of liquid nano-thread. DARSHAK BHUPTANI, SARITH SATHIAN, Indian Inst of Tech-Madras — The stability behavior of nano-scale stationary polar liquid (water) thread in vacuum and in environment under the action of an external constant electric filed is investigated using molecular dynamics (MD) simulations. The Rayleigh instability predicts the non-dimensional wave number for nano-thread to be in the range of 0.2 to 0.5 whereas for a macroscale it is around 0.697. The classical viscid theory predict the first breakup time of nanoscale liquid thread accurately. At nanoscale, thermal fluctuation and surface tension forces plays a critical role in the breakup mechanism. The main objective is to investigate the effect of an additional external force in the form of an electric field on the stability and breakup. The effect of surface tension and the role of thermal fluctuation on the breakup in such cases are investigated. A uniform electric field is applied along the axial direction of thread. For lower values of field strength (0.1V/nm), no breakup is observed as the surface tension force is completely balanced by the electrical force experienced by the surface molecules. With increase in the electrical field strength, different phenomenon such as Taylor’s cone formation, molecule orientation, whipping instability and splaying are observed.
and numerical simulations. In this work, we explore the impact of solid particles on filament thinning and drop formation by using a combination of experiments and numerical simulations. Solid particles influence the process in two ways: they change the interfacial tension and they affect the contact-line dynamics. When solid particles are present, the structure of the interface is changed and it becomes more difficult for the fluid to be pulled through the gaps between the particles.

To fully resolve the movement of rigid particles, a mesh is used to track the moving boundaries of rigid particles. A phase-field method based on the same mesh is used to capture the multiphase flows where fluid interfaces, moving rigid particles, and moving contact lines coexist. Practical applications include Pickering emulsions, froth flotation, and biolocomotion at fluid interface. An ALE algorithm based on the finite element method and an adaptive moving mesh is used to track the moving boundaries of rigid particles. A phase-field method based on the same moving mesh is used to capture the fluid interfaces; meanwhile, the Cahn-Hilliard diffusion automatically takes care of the stress singularity at the moving contact line when a fluid interface intersects a solid surface. To fully resolve the diffuse interface, mesh is locally refined at the fluid interface. All the governing equations, i.e., equations for fluids, interfaces, and particles, are solved implicitly in a unified variational framework.

In this talk, we will present a hybrid Arbitrary-Lagrangian-Eulerian (ALE)-Phase-Field method for the direct numerical simulation of multiphase flows where fluid interfaces, moving rigid particles, and moving contact lines coexist. Practical applications include Pickering emulsions, froth flotation, and biolocomotion at fluid interface. An ALE algorithm based on the finite element method and an adaptive moving mesh is used to track the moving boundaries of rigid particles. A phase-field method based on the same moving mesh is used to capture the fluid interfaces; meanwhile, the Cahn-Hilliard diffusion automatically takes care of the stress singularity at the moving contact line when a fluid interface intersects a solid surface. To fully resolve the diffuse interface, mesh is locally refined at the fluid interface. All the governing equations, i.e., equations for fluids, interfaces, and particles, are solved implicitly in a unified variational framework. In the end we will present some recent results on the water entry problem and the capillary interaction between floating particles (a.k.a. the Cheerios effect), with a focus on the effect of contact-line dynamics.

1We acknowledge financial support by the Dutch Technology Foundation STW.
bath, allowing for the deformation of the latter, is developed and found to match the observed dynamics closely. This effect, causing it to be pulled downward briefly with the downward-accelerating bath. Thus, for a small time interval during each bounce, the drop's motion is significantly influenced by the air layer present above the substrate.

Drops can bounce indefinitely due to the presence of a thin air layer separating the drop from the bath. This is especially evident when the maximum spreading radius of the droplet is analyzed for varying thin film thickness and viscosity. It has been observed that the maximum spreading radius of the droplet is influenced by the air layer thickness and spreading radius over time. For low Weber number impacts on thick films, the drop spreads to its maximum extent, whereas for high Weber number impacts on thin films, the drop does not spread as much.

The adhesion of the droplet to the wall is delayed by the air depletion and liquid film drainage, whereas for high Weber number impacts, the drop detaches immediately. In this study, we systematically investigate the drop impact dynamics on thin liquid films on various substrates by varying the film thickness, viscosity, and impact velocity. High-speed imaging is used to track the droplet morphology and trajectory over time as well as observing instability developments at high Weber number impacts. Moreover, the air layer between the drop and substrate fully imbibes the droplet, the well-known “coffee ring” is suppressed. However, when a residual droplet forms upon the termination of the infiltration regime, the competing particle motion and evaporation regimes, $t_p$ and $t_d$, respectively, define the critical time scales for which the coffee ring will be formed ($t_p/t_d < 1$) or suppressed ($t_p/t_d > 1$).

1National Science Foundation under Grant No. CMMI-1401438.
8:26AM G33.00003 Surface topography measurements for pilot-wave hydrodynamics\textsuperscript{1}, ADAM DAMIANO, MIT, DANIEL HARRIS, UNC, PIERRE-THOMAS BRUN, JOHN BUSH, MIT — We present the results of our attempt to refine the surface Schlieren technique originally developed by Moisy et al. (2009, 2012) to resolve the surface topography associated with capillary wave fields. Our technique is applied to infer the wave field that accompanies millimeter droplets self-propelling on the surface of a vibrating fluid bath. Apart from a shadow region on the order of the drops cross-sectional area, the waves are resolved to a micron scale, allowing for quantitative comparison with existing theoretical models of the wave field. The technique is used to yield insight into the interaction between walking droplets and submerged barriers.

\textsuperscript{1}Thanks to the NSF.

8:39AM G33.00004 Uncertainty Quantification on Entrapped Air in Droplet Impact Events\textsuperscript{1}, SEYEDSHAHABADDIN MIRJALILI, GIANLUCA IACCARINO, ALI MANI, Stanford University — Recent investigations have revealed that entrapment of air films under liquid-liquid impacts can lead to subsequent breakup processes forming many microbubbles per impact. In this work we consider a canonical setting in which individual liquid drops impact a deep flat pool as a model representative of this phenomena. We present an investigation of the uncertainty in the entrapped air associated with the angle of impact relative to the interface-normal direction. In practice, this uncertainty can be induced by surface waves or measurement errors; understanding this sensitivity might help in incorporating impact models as subgrid scale models in large-scale calculations. We have employed the direct numerical simulations of the Navier-Stokes equations in conjunction with a diffuse interface method to track the phase interface. For UQ analysis a quadrature-based and a regression-based non-intrusive polynomial chaos approach are compared. Using the same set of simulations, quadrature-based NIPC showed better convergence than regression-based NIPC. Our results indicate that even order 10 degree variability in the incident angle can lead to significant variability in the entrapped air film. Impact on various measures such as total entrapped volume and film thickness is discussed.

\textsuperscript{1}Supported by ONR

8:52AM G33.00005 Settling of a sphere through a fluid-fluid interface: influence of the Reynolds number\textsuperscript{1}, JEAN-LOU PIERSIEN, Institut de mcanique des fluides de Toulouse, JACQUES MAGNAUDET, Institut de mcanique des fluides de Toulouse - CNRS — When a particle sediments through a horizontal fluid-fluid interface (a situation frequently encountered in oceanography as well as in coating processes), it often tows a tail of the upper fluid into the lower one. This feature is observed in both inertia- and viscosity-dominated regimes. Nevertheless the tail evolution and the particle motion are found to highly depend on the ratio of the two effects, i.e. on the Reynolds number. In this work we study numerically the settling of a sphere through a horizontal fluid-fluid interface using an Immersed Boundary Method combined with a Volume of Fluid approach. To get some more insight into the underlying physical mechanisms, we combine this computational approach with a semi-analytical description based on the concept of Darwin “drift” which allows us to predict the interface evolution, hence the thickness of the film encapsulating the sphere, in the two limits of Stokes flow and potential flow.

\textsuperscript{1}This work was funded by DGA whose financial support is greatly appreciated

9:05AM G33.00006 High Velocity Droplet Rebound On Liquid Pools, WILLIAM DOAK, DANIELLE LAJACONA, PAUL CHIAROT, GUY GERMÁN, SUNY Binghamton — Rebound of high velocity, periodic droplet streams off viscous liquid pools is studied experimentally. Droplets, approximately 60 micrometers in diameter, impact the oil surface at velocities up to 13 m/s and at angles between 2–25 degrees. The oil surface does not degrade or lose its ability to provide rebound even after millions of droplet impacts. The oil was varied to examine the effect that surface tension and viscosity had on droplet rebound. Stable rebound is achievable on oils varying in dynamic viscosity in the range 13–970 Pa.s and surface tensions in the range 19–28 mN/m. When rebound occurs, a consistent 29% loss of droplet kinetic energy is observed. This is a surprising relationship due to the fact that it holds true for all cases of stable rebound regardless of the oil used. We further observe an upper inertial limit where droplets no longer provide stable rebound and instead become fully entrained in the oil pool. This limit is governed by the Rayleigh-Plateau instability and can be characterized and predicted using a modified version of the Weber number. The droplet rebound presented in this study is unique due to the size, velocity, and frequency of the droplets used. Another unique feature is that the rebound manifests itself as an effectively static phenomenon. No motion of the interface – oscillations, waves, or otherwise – was observed during rebound. The quasi-static nature of rebound enabled distinctions to be made regarding energy dissipation and the transition from droplet rebound to entrainment.

9:18AM G33.00007 Leave the seat down: The physics of droplet streams impacting a free surface, NATHAN SPEIRS, RANDY HURD, Utah State University, JESSE BELDEN, Naval Undersea Warfare Center, Newport, TADD TRUSCOTT, Utah State University — Even though toilets are designed for sitting, men often stand to urinate. This behavior can be unintentionally messy and encouraging to sit is often disregarded. In an effort to improve communication, we present a physics-based examination of the overwhelming benefits of sitting to pee. Single droplet impact onto a water surface has been studied intensely for over a century; however, very little is known about the effect of multiple droplet impacts. A single droplet impact predominantly forms a hemispherical subsurface cavity. In contrast, a multi-droplet impact creates a deep conical cavity, often 15 times deeper than the single droplet counterpart. A competition between the cavity formation time and droplet frequency maximize the cavity length and duration, implying that cavity size and shape is dependent on droplet diameter, speed and frequency. Upon collision, larger subsurface cavities result in the formation of larger jets capable of projecting droplets significantly further than single droplet cavities, emphasizing why even a sharp shooter can create a mess. We utilize high-speed imaging and controlled droplet experiments to unravel the key frequencies and parameters at play, offering several suggestions for those wishing to make the transition.

9:31AM G33.00008 A model for bouncing droplets: effects of obstacles and confine-ments, ASLAN KASIMOV, KAUST, LUIZ FARIA, KAUST, MIT — We propose a simple model for particle-wave interactions that captures many aspects of experiments with droplets bouncing on a vibrating bath. The model results from shallow water, small viscosity, and linear approximation to free surface hydrodynamics. The droplet motion is governed by an equation with a force that depends on the wave slope. We study a droplet motion in a wave guide, passing through a single/double slits, and other interactions that are known experimentally to exhibit quantum effects.
9:44AM G33.00009 Nonmonotonic Response of Drop Impacting Liquid Film, XIAOYU TANG, ABHISHEK SAHA, DELIN ZHU, Princeton University, CHAO SUN, Tsinghua University, CHUNG R. LAW, Princeton University — Drop impact on liquid film is ubiquitous in both natural phenomena and industrial applications. The dynamics of the gas layer trapped between the drop and the deformed liquid surface play a crucial role in determining the impact outcomes. However, a quantitative measurement of this gas layer dynamics is extremely challenging because it is hidden behind the deformed liquid film. In this study, high-speed white light interferometry enables the measurement of the gas layer dynamics during the drop impact with high resolutions and is complemented by side view shadowgraphy to observe the penetration process below the liquid surface. Drop impacting with different inertia onto liquid film with various thicknesses is systematically studied to obtain a phase diagram of different outcomes in the h/R-We space, where h/R is the liquid thickness normalized by drop radius, and We is the drop Weber number. It is observed that there exists a critical WeC beyond which the drop always merges with the liquid film. However, for 'subcritical' conditions, there exists a merging peninsula in otherwise globally bouncing region. Across this peninsula, as the liquid film thickness increases, the impact outcome transits from bouncing to merging and to bouncing again. The merging time within this peninsula is longer compared to its 'supercritical' counterpart, indicating different merging mechanisms. Based on scaling analysis, the boundaries between different zones are identified and compared with experiments.

9:57AM G33.00010 Deceleration of noncoalescing droplets, JACOB HALE, DePauw University — Free liquid droplets are observed to glide, or skirt along the surface of a bath of the same fluid, slowing at an exponential rate. The droplet deforms the bath surface creating a dimple which travels along the surface as a wave pulse. Viscous coupling of the droplet and bath surfaces through a thin air film causes a no-slip condition that leads to viscous drag on the bath and perturbs the wave motion of the otherwise free surface. Assuming a linear velocity profile in the bath near the no-slip zone, we show that viscous dissipation in the bath alone accounts for the loss in kinetic energy of the droplet.

Monday, November 23, 2015 8:00AM - 10:10AM
Session G35 Drops: Heat Transfer and Evaporation Ballroom B - Julien Landel, University of Cambridge

8:00AM G35.00001 Impact of internal transport on the convective mass transfer from a droplet into a submerging falling film1, JULIEN R. LANDEL, DAMTP, University of Cambridge, AMALIA THOMAS, UNCPBA, HARRY MCEVOY, DSTL, UK, STUART B. DALZIEL, DAMTP, University of Cambridge — We investigate the convective mass transfer of dilute passive tracers contained in small viscous drops into a submerging falling film. This problem has applications in industrial cleaning, domestic dishwashers, and decontamination of hazardous material. The film Peclet number is very high, whereas the drop Peclet number varies from 0.1 to 1. The characteristic transport time in the drop is much larger than in the film. We model the mass transfer using an analogy with Newton's law of cooling. This empirical model is supported by an analytical model solving the quasi-steady two-dimensional advection-diffusion equation in the film that is coupled with a time-dependent one-dimensional diffusion equation in the drop. We find excellent agreement between our experimental data and the two models, which predict an exponential decrease in time of the drop concentration. The transport characteristic time is related to the drop diffusion time scale, as diffusion within the drop is the limiting process. Our theoretical model not only predicts the well-known relationship between the Sherwood number and the external Reynolds number in the case of a well-mixed drop Sh ∼ Re1/3, it also predicts a correction in the case of a non-uniform drop concentration. The correction depends on Re, the film Schmidt number, the drop aspect ratio and the diffusivity ratio between the two phases. This prediction is in good agreement with experimental data.

8:13AM G35.00002 Droplet evaporation on a soluble substrate1, ALEXANDRA MAILLEUR, CHRISTOPHE PIRAT, JEAN COLOMBANI, Institut Lumière Matière, UMR 5306 Université Lyon 1 - CNRS, Université de Lyon 69622 Villeurbanne, France, CNES COLLABORATION — Stains left by evaporated droplets are ubiquitous in everyday life as well as in industrial processes. Whatever the composition of the evaporating liquid (colloidal suspensions, biological fluids...) the stains are mostly constituted by a deposit at the periphery of the dried drop, similar to a coffee stain (Deegan, 1997). All these studies have been carried with non-reacting solids. In this presentation, we focus on the behavior of a pure-water droplet evaporating on a soluble substrate which is more complex, since three phenomena are strongly interacting: the dissolution of the substrate, the diffusion/convection of the dissolved species into the drop and the evaporation of the liquid. NaCl and KCl single crystals have been chosen for this experimental study as they are fast-dissolving solids. We have observed that the dissolution induces a pinning of the triple line from the beginning of the evaporation, leading to a decrease of the contact angle in time. At the end of the evaporation, a peripheral deposit is always formed, proof of an outward flow inside the drop (coffee-ring effect).

8:26AM G35.00003 Evaporation of multi-component mixtures and shell formation in spray dried droplets, PEDRO VALENTE, IRIS DUARTE, TIAGO PORFIRIO, MÁRCIO TEMTEM, Hovione Farmacência SA — Drug particles where the active pharmaceutical ingredient (APIs) is dispersed in a polymer matrix forming an amorphous solid dispersion (ASD) is a commonly used strategy to increase the solubility and dissolution rate of poorly water soluble APIs. However, the formation and stability of an amorphous solid dispersion depends on the polymer/API combination and process conditions to generate it. The focus of the present work is to further develop a numerical tool to predict the formation of ASDs by spray drying solutions of different polymer/API combinations. Specifically, the evaporation of a multi-component droplet is coupled with a diffusion law within the droplet that minimizes the Gibbs free energy of the polymer/API/solvents system, following the Flory-Huggins model. Prior to the shell formation, the evaporation of the solvents is modelled following the simplified approach proposed by Abramzon & Sirignano (1989) which accounts for the varying relative velocity between solids. In this presentation, we focus on the behavior of a pure-water droplet evaporating on a soluble substrate which is more complex, since three phenomena are strongly interacting: the dissolution of the substrate, the diffusion/convection of the dissolved species into the drop and the evaporation of the liquid. NaCl and KCl single crystals have been chosen for this experimental study as they are fast-dissolving solids. We have observed that the dissolution induces a pinning of the triple line from the beginning of the evaporation, leading to a decrease of the contact angle in time. At the end of the evaporation, a peripheral deposit is always formed, proof of an outward flow inside the drop (coffee-ring effect).

8:39AM G35.00004 Numerical simulations of evaporative instabilities in sessile drops of ethanol on heated substrates, SERGEY SEMENOV, FLORIAN CARLE, MARC MEDALE, DAVID BRUTIN, Aix-Marseille University - IUSTI — The work is focused on numerical simulations of thermo-convective instabilities in evaporating pinnned sessile droplets of ethanol on heated substrates. Computed evaporation rate of a droplet is validated against parabolic flight experiments and semi-empirical theory presented here. To the best authors’ knowledge, this is the first study which combines theoretical, experimental and computational approaches in convective evaporation of sessile droplets. The influence of gravity level on evaporation rate and contributions of different mechanisms of vapor transport (diffusion, Stefan flow, natural convection) are shown. The qualitative difference (in terms of developing thermo-convective instabilities) between steady-state and unsteady numerical approaches is demonstrated.
provide a consensus to understand evaporation of ultrasmall capillary bridges. We present a feasible protocol to directly visualize femtoliter water bridges that evaporate in still air between a microsphere and a flat substrate by utilizing transmission X-ray microscopy. Precise measurements of evaporation kinetics for water bridges indicate that lower water pressure than 1 atm can significantly decelerate evaporation by suppression of vapor diffusion. This finding would provide a consensus to understand evaporation of ultrasmall capillary bridges.

9:05AM G35.00006 ABSTRACT WITHDRAWN

9:18AM G35.00007 Droplet Impingement Boiling on Heated Superhydrophobic Surfaces, JULIE CROCKETT, CRISTIAN CLAVIJO, DANIEL MAYNES, Brigham Young University — When a droplet impinges on a solid surface at a temperature well above the saturation temperature, vaporization of the liquid begins immediately after contact. Different boiling regimes may result depending on the surface temperature and volatility of the liquid. The nucleate boiling regime is characterized by explosive atomization, which occurs when vapor bubbles burst causing an extravagent shower of small micro droplets as well as the well-known “sizzling” sound. In this work, we show that the vapor is surprisingly re-directed during impingement on a superhydrophobic surface such that atomization is completely suppressed. We hypothesize that this results because vapor escapes through the superhydrophobic interface such that the top of the droplet remains free of bursting vapor bubbles. We explore a wide range of surface patterning with feature spacing of 8 to 32 microns and solid area fractions of 10 to 50 percent; surface temperatures from 100 C to 400 C; and Weber numbers of 1 to 100. Atomization is found to decrease with increasing feature spacing and decreasing solid fraction, and vanishes completely for large spacing. It may be that large feature spacing promotes early transition to the Leidenfrost regime.

9:31AM G35.00008 Enhanced condensation heat transfer with wettability patterning, PALLAB SINHA MAHAPATRA, ARITRA GHOSH, University of Illinois at Chicago, RANJAN GANGULY, Brigham Young University, CONSTANTINE MEGARIDIS, University of Illinois at Chicago — Condensation of water vapor on metal surfaces is useful for many engineering applications. A facile and scalable method is proposed for removing condensate from a vertical plate during dropwise condensation (DWC) in the presence of non-condensable gases (NCG). We use wettability-patterned superhydrophilic tracks (filmwise condensing domains) on a mirror-finish (hydrophilic) aluminum surface that promotes DWC. Tapered, horizontal “collection” tracks are laid to create a Laplace pressure driven flow, which collects condensate by the mirror-finish and sends it to vertical “drainage tracks” for gravity-induced shedding. An optimal design is achieved by changing the fractional area of superhydrophilic tracks with respect to the overall plate surface, and augmenting capillary-driven condensate-drainage by the mirror-finish and smooths it to vertical “drainage tracks” for gravity-induced shedding. The design facilitates pump-less condensate drainage and enhances DWC heat transfer on the mirror-finish regions. The study highlights the relative influences of the promoting and retarding effects of dropwise and filmwise condensation on the overall heat transfer improvement on the substrate. The study demonstrated ~ 34% heat transfer improvement on Aluminum surface for the optimized design.

9:44AM G35.00009 Microgrooves improve dew collection, PIERRE-BRICE BINTEN, LIED, UMR 8236 CNRS, Paris, HENRI LHUISIERS, IUSTI, UMR 7343 CNRS, Marseille, LAURENT ROYON, MSC, UMR 7057 CNRS, Paris, CLAIRE MANGENAY, ITODYS, UMR 7086 CNRS, Paris, ANNE MONGRUEL, DANIEL BEYSENS, PMMH, UMR 7636 CNRS, Paris — We present a study about condensation of water drops on inclined surfaces textured with microgrooves and cooled under the dew point temperature. Usual microfabrication techniques are employed to produce substrates (silicium wafer) with grooves of 30-500 micrometers in spacing and 50-150 micrometers in depth. Such patterns induce a faster growth of drops by coalescence, leading to earlier drainage and collection of water at the bottom of the plate. An additional grafting of hydrophilic polymer (Poegma) can even increase the efficiency of such condensation devices.

9:57AM G35.00010 Directional droplet transport at high temperature mediated by structural topography, JING LI, City University of Hong Kong, Hong Kong, YOUMIN HOU, The Hong Kong University of Science and Technology, Hong Kong, MANOJ CHAUDHURY, Lehigh University, Bethlehem, SHUHUAI YAO, The Hong Kong University of Science and Technology, Hong Kong, ZUANKAI WANG, City University of Hong Kong, Hong Kong — Controlling droplet dynamics on textured surfaces is of significant importance for a broad range of applications. Despite extensive advances, our ability to control droplet dynamics at high temperature remains limited, in part due to the emergence of complex wetting states complicated by the phase change process at the triple-phase interfaces. When the temperature of the surface is above a critical temperature, a continuous vapor layer separates the droplet from the hot surface, greatly reducing the heat transfer between the droplet and hot surface. In this work, we show that two concurrent wetting states (Leidenfrost and contact boiling) can be manifested in a single droplet by simply manipulating the structural topography. As a result, droplet vectors automatically move towards the boiling region that is associated with a large heat transfer efficiency between the liquid and solid. Coupled with a dynamic Leidenfrost model, we show experimentally and analytically that the droplet directional motion depends on the interplay between surface structure and its imposed thermal state. Our basic understanding and ability to control the droplet dynamics at high temperature would find many potential applications in high temperature systems such as spray cooling and fuel injection.

Monday, November 23, 2015 8:00AM - 10:10AM – Session G36: Bubbles: Surfactants and Foams

8:00AM G36.00001 Microstructural effects in foam fracture, PETER STEWART, Univ of Glasgow, STEPHEN DAVIS, Northwestern University, SASCHA HILGENFELDT, University of Illinois, Urbana Champaign — We examine the fracture of a quasi two-dimensional aqueous foam under an applied driving pressure, using a network modelling approach developed for metallic foams by Stewart & Davis (J. Rheol., vol. 56, 2012, p. 543). In agreement with experiments, we observe two distinct mechanisms of failure analogous to those observed in a crystalline solid: a slow ductile mode when the driving pressure is applied slowly, where the void propagates as bubbles interchange neighbours through the T1 process, and a rapid brittle mode for faster application of pressures, where the void advances by successive rupture of liquid films driven by Rayleigh–Taylor instability. The simulations allow detailed insight into the mechanics of the fracturing medium and the role of its microstructure. In particular, we examine the stress distribution around the crack tip and investigate how brittle fracture localizes into a single line of breakages. We also confirm that pre-existing microstructural defects can alter the course of fracture.
8:13AM G36.00002 Patterns, Instabilities, Colors, and Flows in Vertical Foam Films, SUBINUER YILIXIATI, EWELINA WOJCICK, YIRAN ZHANG, KRUPA SHAH, VIVEK SHARMA, Chemical Engineering, University of Illinois at Chicago — Understanding and controlling the drainage kinetics of thin films is an important problem that underlies the stability, lifetime and rheology of foams and emulsions. We follow the drainage kinetics of vertical foam films using imaging and color science. Interference between light reflected from two surfactant-laden surfaces that are 100 nm – 10 micron apart leads to thickness-dependent iridescent colors in the visible region. Below 50 nm the thin films appear as black. In this study, we utilize the thin film interference colors as markers for identifying patterns, instabilities and flows within vertical foam films. We study the emergence of thickness fluctuations near the borders (i.e., marginal regeneration) and within thinning films. Finally, we elucidate how buoyancy, capillarity, convection and gravity-driven instabilities and flows, are affected by the choice and concentration of constituents. We find fascinating examples of two-dimensional hydrodynamics and unexplained, if not unprecedented, drainage kinetics.

8:26AM G36.00003 Domain growth kinetics in stratifying foam films, YIRAN ZHANG, VIVEK SHARMA, Chemical Engineering, University of Illinois at Chicago — Baking bread, brewing cappuccino, pouring beer, washing dishes, shoveling, whipping eggs and blowing bubbles all involve creation of aqueous foam films. Typical foam films consist of two surfactant-laden surfaces that are ~5 nm – 10 micron apart. Sandwiched between these interfacial layers is a fluid that drains primarily under the influence of viscous and interfacial forces, including disjoining pressure. Interestingly, a layered ordering of micelles inside the foam films (thickness <100 nm) leads to a stepwise thinning phenomena called stratification, which results in a thickness-dependent variation in reflected light intensity, visualized as progressively darker shades of gray. Thinner, darker domains spontaneously grow within foam films. We show that the domain expansion dynamics exhibit two distinct growth regimes with characteristic scaling laws. Though several studies have focused on the expansion dynamics of isolated domains that exhibit a diffusion-like scaling, the change in expansion kinetics observed after domains contact with the Plateau border has not been reported and analyzed before.

8:39AM G36.00004 Gravitational Drainage of Foam: Planar Films, Stability and Foamability, SOUMYADIP SETT, RAKESH SAHU, ALEXANDER YARIN, Univ of Illinois - Chicago — Gravitational drainage from thin plane vertical surfactant films was studied experimentally by using microinterferometry. Anionic surfactant Sodium Dodecyl Sulfate (SDS), cationic surfactant Dodecyltrimethylammonium Bromide (DTAB), nonionic surfactant Tetraethyylene Glycol Monoctyl Ether (C8E4) and nonionic superspreader trisiloxane SILWET L-77 were used. The experiment results were used to measure the Gibbs surface elasticities of these surfactants, as well as the disjoining pressure of the superspreader. The interpretation of the experimental results was based on the theoretical model developed in the present work. Foamability and foam stability of foams generated from these surfactant solutions were studied experimentally in a settling column. Solutions and foams of SDS and the superspreader mixtures were also studied, and the resulting mechanism of drainage deceleration uncovered.

8:52AM G36.00005 Coupling thermocapillary and solutocapillary stress in 2D microfoam drainage, MARIE-CAROLINE JULLIEN, VINCENT MIRALLES, ESPCI/CNRS, EMMANUELLE RIO, LPS-Orsay, ISABELLE CANTAT, IPR-Rennes, ESPCI/CNRS TEAM, LPS/ORSAY TEAM, IPR-RENNES TEAM — The foam drainage dynamics is known to be strongly affected by the nature of the surfactants stabilising the liquid/gas interface. In the present work, we consider a 2D microfoam stabilized by both soluble (sodium dodecylsulfate) and insoluble (dodecanol) surfactants. The drainage dynamics is driven by a thermocapillary Marangoni stress at the liquid/gas interface [V. Miralles et al., Phys. Rev. Lett., 2014] and the presence of dodecanol at the interface induces stabilized by both soluble (sodium dodecylsulfate) and insoluble (dodecanol) surfactants. The drainage dynamics is driven by a thermocapillary Marangoni stress at the liquid/gas interface [V. Miralles et al., Phys. Rev. Lett., 2014] and the presence of dodecanol at the interface induces stabilization of the foam. We define a dimensionless permeability of the 2D foam in order to get insight into the relative contributions of the two surface stresses at play. We propose different surfactant transport scenarios.

9:05AM G36.00006 Disorder growth in a monodisperse foam in microfluidics, NICOLAS TACCOEN, LadHyX and Dept of mechanics, Ecole Polytechnique, 91128 Palaiseau, France, BENJAMIN DOLLET, Institut de Physique de Rennes, Université Rennes 1, CNRS (UMR 6251), Rennes 35042, France, CHARLES BARoud, LadHyX and Dept of mechanics, Ecole Polytechnique, 91128 Palaiseau, France — Monodisperse foam destabilisation is a complex problem and concerns various applications. For instance, the geometric structure of a foamed gel or concrete must be preserved until the matrix sets. Here we study experimentally this problem by observing, in microfluidics, the evolution of a monolayer of ~30’000 spherical bubbles (radius 0.1mm). We are able to individually track their positions and radii during 20h. We observe a transition from a highly ordered crystalline state (polydispersity=3%) to a completely disorder amorphous state (polydispersity=30%). This final state follows the scaling laws predicted by the classical LSW theory. To describe the transition, we define a geometric criterion that classifies the bubbles in disordered or ordered population. We observe the nucleation and growth of disorder zones, while large ordered zones remain. We show that the destabilisation of the foam is not a homogeneous process, but is the combination of two effects: (i) the quick desabilisation inside disordered zones, (ii) the growth in size of these zones, at the expense of the monodisperse ordered zones. Finally, we measure the volume variation rate of each bubble and show that while most of the gas transfer occurs in disordered zones, activity exists in ordered zones.

9:18AM G36.00007 Modeling Coarsening Induced Foam Drainage Using the Arbitrary Lagrangian Eulerian Method, ANDREW BRANDON, Lycoming College, RAMAGOPAL ANANTH, Naval Research Laboratory — In this presentation, we will explore coarsening induced foam drainage. Coarsening is the process by which a foams average bubble size increases over time due to diffusion of dissolved gas. Through bubble surface movement, coarsening induces drainage and these two processes are capable of altering the foams properties. Current models have explained some aspects of these coupled processes, but there remain questions that these foam-scale models cannot answer. To address some of these questions, we have created a bubble-scale Arbitrary Lagrangian Eulerian model of an idealized, coarsening foam. Drainage is captured by solving the Navier-Stokes equations over the liquid domain and bubble interface movement is described by equations derived to govern the exchange of gas between bubbles. With this model, we have studied the impact that assuming constant film thicknesses (the distance between bubbles) in the coarsening equations can have on drainage. This assumption is typical in current foam-scale models. In this presentation, we will show that allowing the film thicknesses to vary results in a better representation of coarsening induced drainage.
9:31AM G36.00008 Flow of foams in two-dimensional disordered porous media

Benjamin Dollet, Baudouin Geraud, Sian A. Jones, Institut de Physique de Rennes, Yves Meheust, Geosciences Rennes, Isabelle Cantat, Institut de Physique de Rennes, Institut de Physique de Rennes, GEOSCIENCES RENNES TEAM — Liquid foams are a yield stress fluid with elastic properties. When a foam flow is confined by solid walls, viscous dissipation arises from the contact zones between soap films and walls, giving very peculiar friction laws. In particular, foams potentially invade narrow pores much more efficiently than Newtonian fluids, which is of great importance for enhanced oil recovery. To quantify this effect, we study experimentally flows of foam in a model two-dimensional porous medium, consisting of an assembly of circular obstacles placed randomly in a Hele-Shaw cell, and use image analysis to quantify foam flow at the local scale. We show that bubbles split as they flow through the porous medium, by a mechanism of film pinching during contact with an obstacle, yielding two daughter bubbles per split bubble. We quantify the evolution of the bubble size distribution as a function of the distance along the porous medium, the splitting probability as a function of bubble size, and the probability distribution function of the daughter bubbles. We propose an evolution equation to model this splitting phenomenon and compare it successfully to the experiments, showing how at long distance, the porous medium itself dictates the size distribution of the foam.

9:44AM G36.00009 Foam-Driven Fractures of an Elastic Matrix

Ching-Yao Lai, Department of Mechanical and Aerospace Engineering, Princeton University, Sam Smiddy, Department of Chemical and Biological Engineering, Princeton University, Howard Stone, Department of Mechanical and Aerospace Engineering, Princeton University — We report an experimental study of foam-driven fractures in an elastic matrix. When a pressurized foam is constantly injected into a gelatin matrix with a constant flow rate, the foam generates a disc-like fracture which is commonly observed in liquid-driven fractures. Compare to liquid-driven fractures, foam-driven fractures grow faster with time. We investigate how the rheological behavior of foams affects the fracture characteristics by varying the air volume fraction of the foam, the types and concentration of surfactants in the foam. Foam-fracturing reduces the environmental costs of hydraulic fracturing, which inspires this laboratory study.

9:57AM G36.00010 Enhanced dissolution of particle-stabilized bubbles by cooling

Vincent Pouliquet, Valeria Garbin, Imperial College London — Foams and emulsions that are durable and stable under varying environmental conditions (e.g., temperature, humidity) are central in the food and personal care industry. Small bubbles (< 100 µm) need to be stabilized against dissolution even in a gas-saturated liquid, because the Laplace pressure drives diffusion across the curved gas-liquid interface. Solid particles adsorbed at the interface of microbubbles have been shown to prevent coalescence and also arrest bubble dissolution. We studied the effect of changes in temperature on the lifetime of particle-stabilized microbubbles. We report a mechanism of destabilization beyond dissolution arrest, driven by the cooling of the external liquid. We show that the dominant mechanism of destabilization is the increase in solubility of the gas in the liquid, leading to a condition of undersaturation, which drives gas diffusion. Control experiments show that indeed, at constant temperature and pressure, undersaturation alone is sufficient to cause particle-stabilized bubbles to dissolve.

Session G37 Minisymposium: Hydraulic Fracturing

Sheraton Back Bay A - Sungyon Lee, TAMU

8:00AM G37.00001 An Experimental Study of Penny-shaped Fluid-driven Cracks in an Elastic Matrix

Howard Stone1, Princeton University — When a pressurized fluid is injected into an elastic matrix, the fluid generates a fracture that grows along a plane and forms a fluid-filled disc-like shape. For example, such problems occur in various natural and industrial applications involving the subsurface of Earth, such as hydraulic fracturing operations. We report a laboratory study of such a fluid-driven crack in a gelatin matrix, study the crack shape as a function of time, and investigate the influence of different experimental parameters such as the injection flow rate, Young’s modulus of the matrix, and fluid viscosity. We find that the crack radius increases with time as a power law, which has been predicted both for the limit where viscous effects in the flow along the crack opening control the rate of crack propagation, as well as the limit where fracture toughness controls crack propagation. We vary experimental parameters to probe the physical limits and highlight that for our typical parameters both effects can be significant. Also, we measure the time evolution of crack shape, which has not been studied before. The rescaled crack shapes collapse at longer times, based on an appropriate scaling argument, and again we compare the scaling arguments in different physical limits. The gelatin system provides a useful laboratory model for further studies of fluid-driven cracks, some of which we will mention as they are inspired by the physics of hydraulic fracturing. This work is part of the PhD thesis of Ching-Yao Lai and is a collaboration with Drs. Zhong Zheng and Jason Wexler (Princeton University) and Professor Emilie Dressaire (NYU).

1Department of Mechanical and Aerospace Engineering

8:26AM G37.00002 Engineering Fracking Fluids with Computer Simulation

Eric Shaqfeh, Stanford University — There are no comprehensive simulation-based tools for engineering the flows of viscoelastic fluid-particle suspensions in fully three-dimensional geometries. On the other hand, the need for such a tool in engineering applications is immense. Suspensions of rigid particles in viscoelastic fluids play key roles in many energy applications. For example, in oil drilling the “drilling mud” is a very viscous, viscoelastic fluid designed to shear-thin during drilling, but thicken at stoppage so that the “cuttings” can remain suspended. In a related application known as hydraulic fracturing suspensions of solids called “proppant” are used to prop open the fracture by pumping them into the well. It is well-known that particle flow and settling in a viscoelastic fluid can be quite different from that which is observed in Newtonian fluids. For example, in a new well without a previous injection, the “fluid-particle split” at bifurcations is controlled by fluid rheology in a manner that is not understood. Second, in Newtonian fluids, the presence of an imposed shear flow in the direction perpendicular to gravity (which we term a cross or orthogonal shear flow) has no effect on the settling of a spherical particle in Stokes flow (i.e. at vanishingly small Reynolds number). Geosciences Rennes, ISabelle Cantat, Institut de Physique de Rennes, INSTITUT DE PHYSIQUE DE RENNES TEAM, GEOSCIENCES RENNES TEAM — Viscoelastic fluid-particle suspensions are commonly used as hydraulic fracturing fluids and are crucial to the success of hydraulic fracturing operations. Natural rock formations are typically heterogeneous and anisotropic, so flows in such formations are typically complex. Fluid-particle interactions play a crucial role in the evolution of such flows. To develop computer simulations of viscoelastic flow in suspensions of spheres to study these problems. These simulations allow us to understand the detailed physical mechanisms for the remarkable physical behavior seen in practice, and actually suggest design rules for creating new fluid recipes.
8:52AM G37.00003 Subcontinuum mass transport of hydrocarbons in nanoporous media and long-time kinetics of recovery from unconventional reservoirs\(^1\), LYDÉRIC BOCUQUET, Department of Physics, Ecole Normale Superieure — In this talk I will discuss the transport of hydrocarbons across nanoporous media and analyze how this transport impacts at larger scales the long-time kinetics of hydrocarbon recovery from unconventional reservoirs (the so-called shale gas). First I will establish, using molecular simulation and statistical mechanics, that the continuum description — the so-called Darcy law — fails to predict transport within a nanoscale organic matrix. The non-Darcy behavior arises from the strong adsorption of the alkanes in the nanoporous material and the breakdown of hydrodynamics at the nanoscale, which contradicts the assumption of viscous flow. Despite this complexity, all permeances collapse on a master curve with an unexpected dependence on alkane length, which can be described theoretically by a scaling law for the permeance. Then I will show that alkane recovery from such nanoporous reservoirs is dynamically retarded due to interfacial effects occurring at the material's interface. This occurs especially in the hydraulic fracking situation in which water is used to open fractures to reach the hydrocarbon reservoirs. Despite the pressure gradient used to trigger desorption, the alkanes remain trapped for long times until water desorbs from the external surface. The free energy barrier can be predicted in terms of an effective contact angle on the composite nanoporous surface. Using a statistical description of the alkane recovery, I will then demonstrate that this retarded dynamics leads to an overall slow — algebraic — decay of the hydrocarbon flux. Such a behavior is consistent with algebraic decays of shale gas flux from various wells reported in the literature.

\[^1\] This work was performed in collaboration with B. Coasne, K. Falk, T. Lee, R. Pellenq and F. Ulm, at the UMI CNRS-MIT, Massachusetts Institute of Technology, Cambridge, USA.

9:18AM G37.00004 Fracturing in granular media: the role of capillarity, wetting, and disorder, RUBEN JUANES, Massachusetts Institute of Technology — The advent of shale oil and shale gas into the energy landscape has relied on achieving vigorous stimulation of the rock by means of horizontal drilling and hydraulic fracturing. Traditionally, hydraulic fracturing is understood as a single-fluid-phase, pressure-driven process, in which the fluid (typically water with additives) is injected at a high-enough rate that the pressure builds up faster than it can dissipate by permeating into the rock, thereby fracturing it. However, the prevalent conditions for shale (ultra fine pore size, moderate overburden stress, and poor cementation) suggest that capillary forces could play an important role in the fracturing process. Here, we show the results of our recent experimental and theoretical studies on fracturing of granular media by means of injection of an immiscible fluid. We conduct carefully controlled injection experiments in a quasi-2D granular medium (a circular Hele-Shaw cell filled with glass beads), in an experimental set-up that allows us to systematically study the impact of capillarity (by varying injection rate, bead size, and fluid-fluid surface tension), wetting properties (by treating the beads and the cell plates by chemical vapor deposition of silane-based substances) and confinement (by varying the load on the cell). Our choice of defending and invading liquids and granular medium allows us to investigate a wide range of contact angles, from drainage to imbibition. We demonstrate that wettability exerts a powerful influence on the invasion/fracturing morphology of unfavorable mobility displacements. High time resolution imaging techniques and particle image velocimetry (PIV) allow us to quantitatively predict matrix displacement and fracture opening dynamics. Our results provide insights on fracture propagation, fracture length distribution and the fracture drainage area, parameters which are critically important to better understand long-term hydrocarbon production from shale.

9:44AM G37.00005 Particle laden fluids in hydraulic fracturing, BRICE LECAMPION\(^2\), EPFL — The aim of hydraulic fracturing is to create a highly conductive pathway in the reservoir formation of interest. This is typically achieved by “propping” the created fracture with solid particles (i.e. proppant) in order to prevent complete closure of the created fracture due to in-situ stresses when pumping stops. The placement of proppant is therefore the main goal of any fracturing treatment. It involves a number of interesting fluid dynamics problem (suspensions flow with settling, complex rheologies of the base fluid, effect of the fracture roughness etc.). In this talk, we will review the different class of fluids used to achieve proppant placement in fracture particularly focusing on their widely varied rheological properties. We will also discuss the different flow regimes that are typically encountered during a hydraulic fracturing job. In particular, we will notably present in details how recent advances in our understanding of dense suspensions flow [1,2] can improve predictions of proppant placement in the Stokesian regime.


\(^2\) Second author: Dmitry Garagash, Dalhousie University

Monday, November 23, 2015 8:00AM - 10:10AM – Session G38 Acoustics III: Aero-Acoustics

8:00AM G38.00001 Effect of an Adjacent Plate on Supersonic Jet Noise, EPHRAIM GUTMARK, PABLO MORA, FLORIAN BAIER, Univ of Cincinnati, KAILAS KAILASANATH, RYAN JOHNSON, KAMAL VISWANATH, Naval Research Laboratory, UNIVERSITY OF CINCINNATI COLLABORATION — A flat plate was installed parallel to \(M_d = 1.5\) circular and rectangular (\(AR=2\)) jets. Flow structures, from high speed shadowgraphs, and acoustic far-field at design, overexpanded and underexpanded conditions are compared between the free jets and the jets with the plate at different distances from the jet axis, 0.5-3De. The circular and rectangular jets had similar far field acoustics except that the latter had stronger screech tones. The free jet exhibited strong flapping mode and screech when overexpanded and broadband shock associated noise at all NPRs. When the plate was at the nozzle lip, the jet was stabilized and screech and BBSN were suppressed. Flapping and screech reappeared when the plate was moved away from the jet and at the largest stand off distance they were amplified. In the shielded region behind the plate, noise levels at all frequencies except the very low ones were significantly reduced for all plate positions. Conversely, reflection at the azimuthal angle above the plate enhanced OASPL magnitudes across all conditions. Mixing noise dominant in the downstream angle was affected by the plate location at the side azimuthal angle. The measurements were compared with LES computations of the SPL spectra and the OASPL and excellent agreement was shown.
remedying the shortcomings of linear reduced-order models. It has been shown in previous work that linear models correctly predict the evolution of axisymmetric wavepackets up to the end of the potential core of subsonic turbulent jets. Beyond this station linear models fail, and non-linearity is the likely missing piece. For better understanding such nonlinear process, the nonlinear interaction model (NIM) based on PSE solution and linear stability analysis was introduced by Chen and Colonius, using a simplified model of the critical layer, to predict the evolution of instability waves in subsonic jets.

The noise from high performance jet engines of both civilian and military aircraft is an area of active concern. To date, much of the work on noise reduction techniques has focused on axisymmetric circular nozzles. Asymmetric exhaust nozzle configurations, in particular rectangular, are likely to become more important in the future. In this study we validate the far field noise for ideally and over expanded supersonic jets issuing from a low aspect ratio rectangular nozzle geometry. Validation of the acoustic data is performed against experimentally recorded sound pressure level (SPL) spectra for the asymmetric nozzle. Data is shown for the cold jet case and two heated jets for all nozzle pressure ratios. It is shown that elevated operating temperatures result in elevated sound levels across the frequency spectra at all locations. Screech tones, that are present for certain cases, diminish in amplitude or cease completely as the jet is heated.

The essential underlying nonlinear mechanisms are unknown, and it remains unclear how these should be incorporated in a reduced-order model. The non-linear interactions are considered in this work as an “external” harmonic forcing added to the standard linear model. This modelling approach suggests that the critical layer may play a central role in the modelling of wavepackets in subsonic turbulent jets, and indeed may be the key to remedying the shortcomings of linear reduced-order models.
9:31AM G38.00008 Reduction of aerodynamic sound generated in a flow past an oscillating and a fixed cylinder in tandem, YUJI HATTORI, Institute of Fluid Science, Tohoku University — The aerodynamic sound generated in a two-dimensional flow past an oscillating and a fixed circular cylinder in tandem is studied. This flow can be regarded as a simplified model of the sound generation due to the interaction of rotating wings and a strut. The sound pressure is captured by direct numerical simulation of the compressible Navier-Stokes equations using the volume penalization method modified by the author. It is shown that synchronization plays a crucial role in sound reduction. When the frequency of the oscillating cylinder is smaller than that of vortex shedding of the fixed cylinder, the sound is significantly reduced due to synchronization as the frequency of vortex shedding is decreased. Sound reduction also depends on the distance between the cylinders. There are distances at which the forces exerted on the cylinders are in anti-phase so that the total force and thereby the resulting sound are significantly reduced.

9:44AM G38.00009 Large Eddy Simulation of Airfoil Self-Noise at High Reynolds Number1, JOSEPH KOCHIEMOOLAYIL, SANJIVA LELE, Stanford University — The trailing edge noise section (Category 1) of the Benchmark Problems for Airframe Noise Computations (BANC) workshop features five canonical problems. No first-principles based approach free of empiricism and tunable coefficients has successfully predicted trailing edge noise for the five configurations to date. Our simulations predict trailing edge noise accurately for all five configurations. The simulation database is described in detail, highlighting efforts undertaken to validate the results through systematic comparison with dedicated experiments and establish insensitivity to grid resolution, domain size, aleatory uncertainties such as the tripping mechanism used to force transition to turbulence and epistemic uncertainties such as models for unresolved near-wall turbulence. Ongoing efforts to extend the predictive capability to non-canonical configurations featuring flow separation are summarized. A novel, large-span calculation that predicts the flow past a wind turbine airfoil in deep stall with unprecedented accuracy is presented. The simulations predict airfoil noise in the near-stall regime accurately. While the post-stall noise predictions leave room for improvement, significant uncertainties in the experiment might preclude a fair comparison in this regime.

1We thank Cascade Technologies Inc. for providing access to the CharLES toolkit - a massively-parallel, unstructured large eddy simulation framework.

9:57AM G38.00010 Acoustic Scattering by a Vortex Dipole, ZHONGQUAN ZHENG, JUNJIAN ZHANG, Univ of Kansas — Acoustic scattering in vortical flow has been an interesting and practical topic, with applications in problems such as acoustic scattering of turbulent flow. In this study, the linearized Euler equation model is employed to investigate sound wave propagation over a subsonic counter-rotating vortex dipole. Both the stationary and moving due to mutual induction vortex dipoles are studied. The numerical scheme uses a high-order WENO scheme to accommodate the highly convective background flow at high Mach numbers. The simulation results are compared with analytical results and literature data. The theoretical study is focused on the effects of three characteristic length scales in this problem: the incident sound wave length, the vortex core size, and the vortex dipole size. The directivity and scaling laws related to the vortex scattering effects are discussed.

Monday, November 23, 2015 8:00AM - 10:10AM — Session G39 Flies: Pre-Mixed Flames and Flame Instabilities Sheraton Back Bay C - Hong G. Im, King Abdullah University of Science and Technology, Saudi Arabia

8:00AM G39.00001 Experimental observations of the development and growth of flame instabilities formed during vented deflagrations, C. REGIS BAUWENS, FM Global, Research Division, JEFFREY M. BERGTHORSON, McGill University, SERGEY B. DOROFEEV, FM Global, Research Division — The formation of instabilities on the surface of large expanding flames can significantly increase the rate of flame propagation and heat release. As the rate of heat release is the key parameter that determines the pressures that develop, the formation of these instabilities have a strong role in determining the consequences of accidental explosions. For this work, large-scale experiments of uniform propane-air mixtures in a 64 m enclosure were performed. The formation of hydrodynamic flame instabilities, the Darrieus-Landau and Rayleigh-Taylor instabilities, as well as strong flame-acoustic interactions, was observed. These instabilities were found to be the primary driver of the pressures that developed and were ultimately responsible for the overall maximum overpressure.

8:13AM G39.00002 Filtered chemical source term modeling for LES of high Karlovitz number premixed flames, SIMON LAPINTE, GUILLAUME BLANQUART, Caltech — Tabulated chemistry with the transport of a single progress variable is a popular technique for large eddy simulations of premixed turbulent flames. Since the reaction zone thickness is usually smaller than the LES grid size, modeling of the filtered progress variable reaction rate is required. Most models assume that the filtered progress variable reaction rate is a function of the filtered progress variable and its variance where the dependence can be obtained through the probability density function (PDF) of the progress variable. Among the most common approaches, the PDF can be presumed (usually as a β-PDF) or computed using spatially filtered one dimensional laminar flames (FLF). Models for the filtered source term are studied a priori using results from DNS of turbulent n-heptane/air premixed flames at varying Karlovitz numbers. Predictions from the optimal estimator and models based on laminar flames using a β-PDF or a FLF-PDF are compared to the exact filtered source term. For all filter widths and Karlovitz numbers, the optimal estimator yields small errors while β-PDF and FLF-PDF approaches present larger errors. Sources of differences are discussed.

8:26AM G39.00003 Bluff-body stabilized flame dynamics of lean premixed syngas combustion, HONG G. IM, YU JEONG KIM, BOK JIK LEE, King Abdullah Univ of Sci & Tech (KAUST), KAUST TEAM — Recently, syngas combustion has been actively investigated for the potential application to integrated gasification combined cycle (IGCC) systems. While lean premixed combustion is attractive for both reduced emission and enhanced efficiency, flame instability becomes often an issue. Bluff-bodies have been adopted as effective flame holders for practical application of premixed flames. In the present study, high-fidelity direct numerical simulations are conducted to investigate the dynamics of lean premixed syngas flames stabilized on a bluff-body, in particular at the near blow-off regime of the flame. A two-dimensional domain of 4 mm height and 20 mm length with a flame holder of a 1 mm-by-1 mm square geometry is used. For a syngas mixture with the equivalence ratio of 0.5 and the CO:H2 ratio of 1, several distinct flame modes are identified as the inflow velocity approaches to the blowoff limit. The sequences of extinction pathway and combustion characteristics are discussed.
Temperature is an essential parameter in turbulent combustion. The existence of turbulence significantly enhances the flame speed of a premixed flame mainly through the increase in the total flame surface area and modification of the flame structure. As of now, the highest turbulent flame speed reported is around 35 times those of the laminar flames, and there is no consensus if this is the upper limit or even if there exists one. In the present experimental work, we report highly enhanced turbulent flame propagation, with the ratio of turbulent flame speed to laminar flame speed reaching 200. Moreover, we demonstrated that such enhancements occur for extremely weak mixtures, where adiabatic flame temperatures are lower than 900 K and are commonly believed to be beyond the flammability of sustained one-dimensional laminar flame propagation. We further identified that such a strong enhancement effect occurs for mixtures with either extremely small Lewis number or large mass diffusivity of the deficient reactant and that such flames exhibit different morphology from previously observed turbulent flames, as fingering-like structures are developed on the flame fronts and local extinction and re-ignition are frequently observed. This work demonstrates the extension of flammability limit by turbulence and differential diffusion, enabling sustained flame propagation with extremely low burnt gas temperature (<1000 K), and the highest flame speed enhancement by turbulence so far.

9:05AM G39.00006 An Improved Flamelet-Based Model for Non-Premixed Supersonic Combustion, ZHIPENG LOU, FOLUSO LADEINDE, Stony Brook University, Stony Brook, NY 11794-2300, WIENHAI LI, TTC Technologies, Inc., Centereach, NY 11720 — The flamelet approach to turbulent reacting flows, though originally developed for essentially incompressible flows, has been used by many authors to simulate supersonic combustion, often without much justification other than that pressure scales in certain ways. In a compressible flow, pressure and temperature vary strongly, meaning that the use of a fixed value of pressure for generating flamelet libraries may be prone to errors in the flamelet modeling of supersonic combustion. We study the influence of static pressure on the flamelet solutions intended for use in modeling supersonic combustion. With various values of static pressure, we found significant differences in the values of the quenching stoichiometric scalar dissipation rate, reaction rate of species and progress variable, heat release rate and the temperature profile. As a result, at high static pressures, the flame is less likely to extinguish and the S-curve shows a steeper angle. We have experimented with the addition of pressure as an independent variable in the flamelet table, toward modeling pressure-sensitive properties and the variable quenching conditions. The effects of this kind of scheme on supersonic combustion will be discussed.

9:18AM G39.00007 An Investigation of a Hybrid Mixing Model for PDF Simulations of Turbulent Premixed Flames1, HUA ZHOU, SHAN LI, HU WANG, ZHUHAI LI, Tsinghua Univ — Predictive simulations of turbulent premixed flames over a wide range of Damkohler numbers in the framework of Probability Density Function (PDF) method still remain challenging due to the deficiency in current micro-mixing models. In this work, a hybrid micro-mixing model, valid in both the flamelet regime and broken reaction zone regime, is proposed. A priori testing of this model is first performed by examining the conditional scalar dissipation rate and conditional scalar diffusivity in a 3-D direct numerical simulation dataset of a temporally evolving turbulent slot jet flame of lean premixed H2-air in the thin reaction zone regime. Then, this new model is applied to PDF simulations of the Piloted Premixed Jet Burner (PPJB) flames, which are a set of highly shear turbulent premixed flames and feature strong turbulence-chemistry interaction at high Reynolds and Karlovitz numbers.

9:31AM G39.00008 Large Eddy Simulation of Radiation Effects on Pollutant Emissions in Diluted Turbulent Premixed Flames, A. CODY NUNNO, MICHAEL E. MUELLER, Princeton University — Radiation effects are examined in turbulent premixed flames using a detailed Large Eddy Simulation (LES) approach. The approach combines a tabulated premixed flamelet model (Flamelet Generated Manifolds) with an optically thin radiation model. Radiation heat loss is tracked using an enthalpy deficit coordinate. Heat loss in the flamelets is calculated by varying a coefficient on the radiation source term, ranging from zero (adiabatic) to unity (full optically thin heat loss). NOx emissions are modeled with an additional transport equation that is able to capture unsteady effects resulting from slow kinetics. The model is compared against experimental measurements of methane-air piloted turbulent premixed planar jet flames with increasing levels of water dilution that maintain a constant adiabatic flame temperature. The effects of water dilution on global flame structure and NO emissions resulting directly and indirectly from radiation are examined in detail.

9:44AM G39.00009 Lagrangian Analysis of Premixed Turbulent Flames, CLARISSA BRINER, PETER HAMILTON, Univ of Colorado - Boulder, ALEXEI POLUDNENKO, Naval Research Laboratory — Turbulent premixed combustion is a complicated problem that requires understanding of turbulence and chemistry, as well as their interactions. By contrast to the Eulerian approach, Lagrangian analyses track the evolution of chemical species and flow properties for an advecting fluid parcel. This approach permits detailed analysis of chemical reaction rates and validation of chemical reaction models. Lagrangian trajectories also allow changes in chemical species and flow properties to be examined locally and instantaneously through premixed flamelets. In this study, a Lagrangian analysis has been performed on data from direct numerical simulations of premixed H2-air flames for two different turbulence intensities, using a 8-species chemical reaction mechanism. The relative contributions of dynamical budget terms are calculated for both chemical species, including reaction and diffusion terms as well as viscous transport, dilatation, and strain. This approach allows for the examination of the motion throughout the flame are also characterized using multi-point correlations. The results reveal complicated dynamics, including non-monotonic behavior of temperature and fuel mass fractions along trajectories, as well as changes scales of motion through the flameout.
the shock Mach number reduces the induction time and eventually leads to deflagration-detonation transition. Ignition by a Ma wave directly leads to a detonation wave, driven by different reaction wave types, we performed DNS of RSBI for shock Mach numbers in the range of Ma = 2.13 - 2.50 at a constant initial pressure of p0 = 50 atm. Deflagration, dominated by H2O and OH production, is observed for a shock Mach number of Ma = 2.13. Increasing the shock Mach number reduces the induction time and eventually leads to deflagration-detonation transition. Ignition by a Ma = 2.50 shock wave directly leads to a detonation wave, driven by H2O2 and H2O2 high-pressure chemistry. Richtmyer-Meshkov instability, subsequent Kelvin Helmholtz instabilities, and bubble expansion are highly affected by the reaction wave. Mixing is significantly decreased by both reaction waves types. In particular detonation waves reduce the mixing distinctly.

8:13AM G40.00002 Mach induced ignition and DDT in the presence of mechanically driven fluctuations1. WENTIAN WANG, JAMES G. MCDONALD, MATEI I. RADULESCU, University of Ottawa — The present study addresses the problem of shock induced ignition and transition to detonation in the presence of mechanical and thermal fluctuations. These departures from a homogeneous medium are of significant importance in practical situations, where such fluctuations may promote hot-spot ignition and favor the flame transition to detonation. The problem is studied in 1D, where a piston-induced shock ignites the gas. The fluctuations in the shock-compressed medium are controlled by allowing the pistons speed to oscillate around a mean, with controllable frequency and amplitude. A Lagrangian numerical formulation is used, which allows to treat exactly the transient boundary condition at the piston head. The hydrodynamic solver is coupled with the reactive dynamics of the gas using Cantera. The code was verified by comparison with steady state ZND solutions and previous shock induced ignition results in homogeneous media. Results obtained for different fuels illustrate the strong relation of the DDT amplification length to mechanical fluctuations in systems with a high effective activation energy and fast rate of energy deposition, consistent with experiments performed on fast flame acceleration in the presence of strong mechanical perturbations.

1Financial support from NSERC and Shell, with A. Pekalski and M. Levin as technical monitors, are greatly acknowledged.

8:26AM G40.00003 Diffusion-flame ignition by shock-wave impingement on a supersonic mixing layer. ANTONIO L. SANCHEZ, CESAR HUETE, FORMAN A. WILLIAMS, University of California San Diego, JAVIER URZAY, Stanford University CTR — Ignition in a supersonic mixing layer interacting with an oblique shock wave is investigated analytically and numerically under conditions such that the post-shock flow remains supersonic. The study requires consideration of the structure of the post-shock hydrodynamic instabilities. Experimental data is derived from the reacting dynamics of the gas using Cantera. The code was verified by comparison with steady state ZND solutions and previous shock induced ignition results in homogeneous media. Results obtained for different fuels illustrate the strong relation of the DDT amplification length to mechanical fluctuations in systems with a high effective activation energy and fast rate of energy deposition, consistent with experiments performed on fast flame acceleration in the presence of strong mechanical perturbations.

8:39AM G40.00004 Acoustic timescale characterization of hot spot ignition in thermally stratified mixtures. FYNN REINBACHER, JONATHAN REGELE, Iowa State University — Thermal stratification and the formation of hot spots in reactive mixtures are of key interest to characterize the autoignition behavior of charges in internal combustion engines. Critical gradient conditions and local maximum sizes of a finite hot spot centers can be used to describe such a hot spot. In previous work, one- and two-dimensional hot spots consisting of a linear temperature gradient and constant plateau have been characterized on an acoustic timescale. In the present work, random one-dimensional temperature fields, derived from Fourier superposition for temperature fluctuations with a temperature spectrum similar to Passot-Pouquet kinetic energy spectrum, are analyzed. The linear gradient constant plateau model is compared to a more realistic hot spot temperature profile. Hot spots in the one-dimensional temperature fields are modeled with linear gradients and constant plateaus in order to be characterized with acoustic time scale analysis. Probability distributions for different excitation-to-acoustic timescale ratios are calculated for a range of engine conditions.
8:52AM G40.00005 Prediction of strong and weak ignition regimes in turbulent reacting flows with temperature fluctuations: A direct numerical simulation study\textsuperscript{1}. PINAKI PAL, Univ of Michigan - Ann Arbor, MAURO VALORANI, Sapienza University, Rome, Italy, HONG IM, King Abdullah University of Science and Technology, Thuwal, Saudi Arabia, MARGARET WOOLDRIDGE, Univ of Michigan - Ann Arbor — The present work investigates the auto-ignition characteristics of compositionally homogeneous reactant mixtures in the presence of thermal non-uniformities and turbulent velocity fluctuations. An auto-ignition regime diagram is briefly discussed, that provides the framework for predicting the expected ignition behavior based on the thermo-chemical properties of the reactant mixture and flow/scalar field conditions. The regime diagram classifies the ignition regimes mainly into three categories: weak (deflagration dominant), reaction-controlled strong and mixing-controlled strong (volumetric ignition/spontaneous propagation dominant) regimes. Two-dimensional direct numerical simulations (DNS) of auto-ignition in a lean thermally-stratified syngas/air turbulent mixture at high-pressure, low-temperature conditions are performed to assess the validity of the regime diagram. Various parametric cases are considered corresponding to different locations on the regime diagram, by varying the characteristic turbulent Damköhler and Reynolds numbers. Detailed analysis of the reaction front propagation and heat release indicates that the observed ignition behaviors agree very well with the corresponding predictions by the regime diagram.

\textsuperscript{1}U.S. DOE NETL award number DE-FE0007465; King Abdullah University of Science and Technology (KAUST)

9:05AM G40.00006 Numerical investigation of kinetic energy dynamics during auto-ignition of n-heptane/air mixture, PAULO LUCENA KREPPEL PAES, JAMES BRASSEUR, YUAN XUAN, The Pennsylvania State University — Many engineering applications involve complex turbulent reacting flows, where nonlinear, multi-scale turbulence-combustion couplings are important. Direct representation of turbulent reacting flow dynamics is associated with prohibitive computational costs, which makes it necessary to employ turbulent combustion models to account for the effects of unresolved scales on resolved scales. Classical turbulence models are extensively employed in reacting flow simulations. However, they rely on assumptions about the energy cascade, which are valid for incompressible, isothermal homogenous isotropic turbulence. A better understanding of the turbulence-combustion interactions is required for the development of more accurate, physics-based sub-grid-scale models for turbulent reacting flows. In order to investigate the effects of reaction-induced density, viscosity, and pressure variations on the turbulent kinetic energy, Direct Numerical Simulation (DNS) of autoignition of partially-premixed, lean n-heptane/air mixture in three-dimensional homogeneous isotropic turbulence has been performed. This configuration represents standard operating conditions of Homogeneous-Charge Compression-Ignition (HCCI) engines. The differences in the turbulent kinetic energy balance between the present reacting flow and incompressible, isothermal homogeneous isotropic turbulence are highlighted at different stages during the autoignition process.

9:18AM G40.00007 ABSTRACT WITHDRAWN —

9:31AM G40.00008 Characterization of Diesel and Gasoline Compression Ignition Combustion in a Rapid Compression-Expansion Machine using OH* Chemiluminescence Imaging\textsuperscript{2}. SUNDAR RAJAN KRISHNAN, KALYAN KUMAR SRINIVASAN, Mississippi State University, MATTHEW STEGMEIR, TSI, Inc. — Direct-injection compression ignition combustion of diesel and gasoline were studied in a rapid compression-expansion machine (RCEM) using high-speed OH* chemiluminescence imaging. The RCEM (bore = 84 mm, stroke = 110-250 mm) was used to simulate engine-like operating conditions at the start of fuel injection. The fuels were supplied by a high-pressure fuel cart with an air-over-fuel pressure amplification system capable of providing fuel injection pressures up to 2000 bar. A production diesel fuel injector was modified to provide a single fuel spray for both diesel and gasoline operation. Time-resolved combustion pressure in the RCEM was measured using a Kistler piezoelectric pressure transducer mounted on the cylinder head and the instantaneous piston displacement was measured using an inductive linear displacement sensor (0.05 mm resolution). Time-resolved, line-of-sight OH* chemiluminescence images were obtained using a Phantom V611 CMOS camera (20.9 kHz @ 512 x 512 pixel resolution, ~48 µs time resolution) coupled with a short wave pass filter (cut-off ~348 nm). The OH* distributions, which indicate high temperature flame regions within the combustion chamber, were used to discern the characteristic differences between diesel and gasoline compression ignition combusition.

\textsuperscript{2}The authors gratefully acknowledge facilities support for the present work from the Energy Institute at Mississippi State University.

9:44AM G40.00009 Ignition and propagation of premixed methane flame by successive laser-induced breakdowns, LYDIA WERMER, Worcester Polytechnic Institute, MOON SOO BAK, Sungkyunkwan University, SEONG-KYUN IM, Worcester Polytechnic Institute — The ignition and the propagation of premixed methane flame by two successive laser-induced breakdowns were investigated. The ignition and flame propagation were visualized using a high-speed schlieren imaging technique. Experiments were performed for various time intervals between the two pulses ranging from nanoseconds to milliseconds and were compared to the ignition by a single laser breakdown. For time intervals in the nanosecond range, the second pulse energy coupled with the first breakdown increasing energy absorption in the breakdown. For time intervals in the microseconds and milliseconds, the blast wave from the second breakdown interacted with the propagating flame induced by the first breakdown. The interaction triggered Richtmyer-Meshkov instability enhancing flame propagation. It is observed that there are time intervals inhibiting the second breakdown due to the heating either by the first breakdown or by combustion.

9:57AM G40.00010 Investigation of Laser Ignition Behavior of Iso-octane and Ethanol Blends, NATHAN PETERS, PATRINA BAILEY, DESHAWN COOMBS, BENJAMIN AKIH-KUMGEH, Syracuse University — Laser-induced ignition is a promising technology for combustion initiation in gas turbines and internal combustion engines. There is renewed interest in this technology in recent years due to its ability to ignite lean mixtures which are desirable for cleaner combustion. Research in this area has mainly focused on methane combustion. Effects of pressure, temperature, and ignition energy have been studied. Another fuel of practical interest which has not been studied as extensively is iso-octane. Due to the complexities of the laser ignition process, there is still a lot that to be understood, especially during the early stages of ignition. In this work we study the ignition of iso-octane and blends including ethanol, induced by focused light pulse from an Nd:YAG laser emitting at 532 nm. Experiments are carried out in a cylindrical stainless steel vessel, equipped with 6 optical accesses. Schlieren imaging and laser interferometry are used to image the ignition process. We seek to understand the multiphysics of the early stages of ignition including shock wave velocity, plasma to flame kernel transition, and flame kernel quenching under lean conditions.

10:35AM H1.00001 Uniaxial deformation of a soft porous material. CHRIS MACMINN, University of Oxford, ERIC DUFRESNE, JOHN WETTLAUFER, Yale University — Compressing a porous material will decrease the volume of pore space, driving fluid out. Similarly, injecting fluid into a porous material will drive mechanical deformation, distorting the solid skeleton. This poromechanical coupling has applications ranging from cell and tissue mechanics to geomechanics and hydrogeology. The classical theory of linear poroelasticity captures this coupling by combining Darcy’s law with linear elasticity and then further linearizing in the strain. This is a good model for very small deformations, but it becomes increasingly inappropriate as deformations grow larger, and moderate to large deformations are common in the context of phenomena such as swelling, damage, and extreme softness. Here, we compare the predictions of linear poroelasticity with those of a rigorous large-deformation framework in the context of two uniaxial model problems. We explore the error associated with the linear model in both steady and dynamic situations, as well as the impact of allowing the permeability to vary with the deformation.

10:48AM H1.00002 Dynamics of swelling and drying in a spherical gel. THIBAULT BERTRAND, Yale University, CHRISTOPHER W. MACMINN, Oxford University, SHOMEEK MUKHOPADHYAY, Yale University, JORGE PEIXINHO, CNRS & Université de Normandie — Swelling is a fundamental process in biology, engineering, and the earth sciences. Macroscopically, swelling is a volumetric-growth process in which a porous material expands due to the spontaneous imbibition of additional pore fluid. However, swelling is distinct from other growth processes because it is inherently poromechanical: Local expansion of the pore space requires that additional fluid be drawn from elsewhere in the material, or into the material from across the boundaries. Here, we study the swelling and subsequent drying of a sphere of hydrogel. Despite the apparent simplicity of this problem, no model has yet shown satisfying quantitative agreement with experiments in terms of the dynamics of swelling and drying. We develop a dynamic model based on large-deformation poromechanics and we compare the predictions of the model with a series of experiments performed with polyacrylamide spheres. We use the model and the experiments to study the dynamics of swelling and drying, and to highlight the fundamental differences between these two processes. Although we assume spherical symmetry, the model also provides insight into the transient patterns that form and then vanish during swelling, as well as the tendency of large spheres to fracture during drying.

11:01AM H1.00003 Boundary conditions between poro-elastic medium and pure fluid in multi-scale modelling. UGIS LACIS, SHERVIN BAGHERI, Linne Flow Centre, KTH Mechanics, 10044 Stockholm, Sweden — Accurate modelling of porous and poro-elastic media has been a long standing issue in geophysics, fluid mechanics, and biology. There has been a notable development of continuous models for both porous and poro-elastic materials, nevertheless there is still an on-going debate about the modelling of effective boundary conditions for different types of media, such as, poro-elastic medium and free fluid, porous medium and solid wall. Some recent works have rigorously treated interface between porous medium and free fluid, however, there have been no detailed investigation regarding the interface between poro-elastic medium and free fluid. We use the multi-scale modelling to arrive with averaged, effective macroscopic equations for description of a poro-elastic medium. Then we investigate the interface in detail and arrive with effective boundary conditions. To validate our model, we construct direct numerical simulations using an immersed boundary (IB) method. The IB method is beforehand validated with respect to theoretical predictions for Darcy’s flow in porous materials with a given pore structure.

11:14AM H1.00004 Capillary rise and swelling in cellulose sponges. JONGHYUN HA, JUNGHUL KIM, HO-YOUNG KIM, Seoul National University — A cellulose sponge, which is a mundane example of a porous hydrophilic structure, can absorb and hold a significant amount of liquid. We present the results of experimental and theoretical investigation of the dynamics of the capillary imbibition of various aqueous solutions in the sponge that swells at the same time. We find that the rate of water rise against the resistance caused by gravitational and viscous effects deviates from Washburn’s rule beyond a certain threshold height. We rationalize the novel power law of the rise height versus time by combining Darcys law with hydrosopic swelling equation and also predict the threshold height. The scaling law constructed through this work agrees well with the experimental results, shedding light on the physics of capillary flow in deforming porous media.

11:27AM H1.00005 Capillary displacement of viscous liquids. PETER WALLS, GREGOIRE DE-QUIDT, JAMES BIRD, Boston University — When a capillary tube is brought into contact with a wetting liquid, surface tension forces overcome gravity and the liquid spontaneously rises into the tube until an equilibrium height is reached. The early viscous dynamics of the rise typically follow the well-known Lucas-Washburn law, which is independent of gravity and neglects the displaced fluid. Here we explore the early viscous dynamics when the properties of displaced fluid are significant. Using a combination of experiments and theory, we show how the characteristic behavior of the Lucas-Washburn law is modified when the viscosity of the displaced fluid is comparable to or exceeds the wetting fluid. Additionally, we find that the effects of gravity reshape the dynamics of the capillary rise, not only in the late viscous regime, but also in the early viscous regime.

11:40AM H1.00006 Liquid spreading on ceramic-coated carbon nanotube films and patterned microstructures. HANGBO ZHAO, A. JOHN HART, Massachusetts Institute of Technology — We study the capillary-driven liquid spreading behavior on films and microstructures of ceramic-coated vertically aligned carbon nanotubes (CNTs) fabricated on quartz substrates. The nanoscale porosity and micro-scale dimensions of the CNT structures, which can be precisely varied by the fabrication process, enable quantitative measurements that can be related to analytical models of the spreading behavior. Moreover, the conformal alumina coating by atomic layer deposition (ALD) prevents capillary-induced deformation of the CNTs upon meniscus recession, which has complicated previous studies of this topic. Washburn-like liquid spreading behavior is observed on non-patterned CNT surfaces, and is explained using a scaling model based on the balance of capillary driving force and the viscous drag force. Using these insights, we design patterned surfaces with controllable spreading rates and study the contact line pinning-depinning behavior. The nanoscale porosity, controllable surface chemistry, and mechanical stability of coated CNTs provide significantly enhanced liquid-solid interfacial area compared to solid microstructures. As a result, these surface designs may be useful for applications such as phase-change heat transfer and electrochemical energy storage.

1Funding for this project is provided by the National Institutes of Health and the MIT Center for Clean Water and Clean Energy supported by the King Fahd University of Petroleum and Minerals.
11:53AM H1.00007 Switchable imbibition in nanoporous gold . PATRICK HUBER, Institute of Materials Physics and Technology, Hamburg University of Technology, JUERGEN MARKMANN, Institute of Materials Research, Materials Mechanics, Helmholtz-Zentrum Geesthacht, HUILING DUAN, State Key Laboratory for Turbulence and Complex Systems, Peking University, JOERG WEISSMUeller, Institute of Materials Physics and Technology, Hamburg University of Technology, YAHUI XUE, State Key Laboratory for Turbulence and Complex Systems, Peking University — Spontaneous imbibition enables the elegant propelling of nano-flows because of the dominance of capillarity at small length scales. The imbibition kinetics are, however, solely determined by the static host geometry, the capillarity, and the fluidity of the imbibed liquid. This makes active control particularly challenging. Here we show for aqueous electrolyte imbibition in nanoporous gold that the fluid flow can be reversibly switched on and off through electric potential control of the solid/liquid interfacial tension, that is, we can accelerate the imbibition front, stop it, and have it proceed at will. Simultaneous measurements of the multiphase flow and the electrical current allow us to document simple scaling laws for the imbibition kinetics, and to explore the charge transport in the metallic nanopores. Our findings demonstrate that the high electric conductivity along with the pathways for fluid/ionic transport render nanoporous gold a versatile, accurately controllable electrocapillary pump and flow sensor for minute amounts of liquids with exceptionally low operating voltages. (1) Yahui Xue, Juergen Markmann, Huilin Duan, Juerg Weissmueller, Patrick Huber, Nature Communications 5, 4237 (2014).

12:06PM H1.00008 Wettability control on fluid-fluid dispersions in patterned microfluidics . BENZHONG ZHAO, Massachusetts Institute of Technology, CHRISTOPHER MACMINN, University of Oxford, RUBEN JUANES, Massachusetts Institute of Technology — Two-phase flow in porous media is important in many natural and industrial processes. While it is well known the wetting properties of porous media can vary drastically depending on the media and the pore fluids, their effect continues to challenge our microscopic and macroscopic descriptions. We conduct experiments via radial displacement of silicon oil by water in microfluidic devices patterned with vertical posts. These devices allow for flow visualization in a complex but well-defined microstructure. Additionally, the surface energy of the devices can be tuned over a wide range of contact angles. We perform injection experiments with highly unfavorable mobility contrast at rates over four orders of magnitude. We focus on three wetting conditions: drainage $\theta = 120^\circ$, weak imbibition $\theta = 60^\circ$, and strong imbibition $\theta = 70^\circ$. We find that drainage is favored over imbibition and selective wetting is achieved. Moreover, compared to the experimental results obtained using x-ray tomography and CPSC measurement with the multi-particle collision dynamics numerical results.

12:19PM H1.00009 What controls the wettability of bidisperse sphere packings? . ROBABEH MOOSAVI, Max Planck Institute of Dynamics and Self-Organization, Göttingen, Germany, JULIE MURISON, C&I Ltd, Frankfurt, Germany, THOMAS HILLER, Max Planck Institute of Dynamics and Self-Organization, Göttingen, Germany — We report experiments on liquid two-phase flow in bidisperse sphere packings. The bidisperse bead pack consists of small and large beads which are either oil wetting or water wetting. The aim of our work is to understand what determines the average wettability of the sample and what effects the amount of remaining oil trapped in the bead pack invaded by water. The method is to measure the capillary pressure saturation curves (CPSC). The data indicates that segregation plays a role. Moreover, we compare the experimental results obtained using x-ray tomography and CPSC measurement with the multi-particle collision dynamics numerical results.

12:32PM H1.00010 Convective drying of a macroporous medium: a comparison of original porous asphalt geometry with randomized Kelvin cells1 . SREEYUTH LAL, ETH - Zurich, FRANCESCO LUCCI, THIIUS DEFAEUE, LYLE POULIKAKOS, MANFRED PARTL, DOMINIQUE DEROME, EMMA, JAN CARMELIJ, ETH - Zurich — Forced convective drying of a macroporous medium is a complex interplay of enhanced air-vapor mixing due to turbulent airflow at the air-solid interface and the momentum transfer resulting from air infiltration into the material. Such air infiltration is expected to have a non-trivial effect on the drying rate of a material like porous asphalt (PA), which is characterized by large, interconnected pores open to the air. Through a series of CFD simulations performed on an original PA geometry extracted from CT scans, we quantify the relative impacts of interior material resistance and boundary layer resistance on moisture transport in PA. At wind speeds below 1 m/s, the effect of material resistance on the total moisture transport is found to be high due to low air infiltration. At higher wind speeds, air infiltration increases by which the material resistance decreases. Similar simulations are performed on an idealized PA geometry made from randomized Kelvin cells (KC) since they are computationally less expensive, and thus ideal for parametric studies. However, in KC cells, drying from air infiltration is stronger than diffusive drying even at low wind speeds. This shows the need to fine-tune the pore connectivity of KC to better match the air infiltration observed in PA. 1This research was supported by a Swiss National Science Foundation (SNSF) Grant (200021-143651).


10:35AM H2.00001 Non-equilibrium tuning of attractive colloidal gels . ARMAN BORO-MAND, JOAO MAIA, Case Western Reserve University — In colloidal gel systems, the presence of multiple interactions in multiple length scales such as Van der Waals, depletion attractions, and electrostatic repulsions makes these systems challenging from both experimental and simulation aspects. Recently, there has been growing interest to tune and manipulate the structural and dynamics properties of those systems without adjusting interparticle interactions, just by taking them out of equilibrium. In this work, we used Core-Modified Dissipative Particle Dynamics (CM-DPD) with a modified depletion potential, as a coarse-grain model to address the gel formation process in short ranged-attractive bidisperse sphere packings. The bidisperse bead pack consists of small and large beads which are either oil wetting or water wetting. Aim of our work is to understand what determines the average wettability of the sample and what effects the amount of remaining oil trapped in the bead pack invaded by water. The method is to measure the capillary pressure saturation curves (CPSC). The data indicates that segregation plays a role. Moreover, we compare the experimental results obtained using x-ray tomography and CPSC measurement with the multi-particle collision dynamics numerical results.

10:48AM H2.00002 Constitutive upsampling of MR fluids . GRIGOR NIKA, BOGDAN VERNESCU, Worcester Polytech Inst — We consider a suspension of solid magnetizable particles in a viscous fluid with an applied external magnetic field. We assume the fluid to be electrically non-conducting. Thus, we use the quasi-static Maxwell equations coupled with the Stokes equations to capture the magnetorheological effect. We upscale using two scale asymptotic expansions to obtain the effective equations consisting of a coupled nonlinear system in a connected phase domain as well as the new constitutive laws. Qualitative properties of the solution of this nonlinear system are studied.
11:01AM H2.00003 Hydrodynamically interacting particles confined by a spherically cavity via dynamic simulation: a model for intracellular transport, CHRISTIAN APONTE-RIVERA, YU SU, ROSEANNA ZIA, Cornell University — We study the short- and long-time self-diffusion of hydrodynamically interacting colloids enclosed within a spherically cavity as a model for intracellular transport. Prior models of such behavior began with a single enclosed particle; attempts to enlarge such models to many particles have seen limited success owing to the challenges of accurately modeling many-body far-field and singular near-field hydrodynamic interactions. To overcome these difficulties we have developed a new set of hydrodynamic mobility functions to couple particle motion with hydrodynamic force moments which, when inverted and combined with near-field resistance functions form a complete coupling tensor that accurately captures both far-field and near-field physics, for an arbitrary number of particles enclosed by a spherical cavity of arbitrary relative size. The mobility functions are implemented into a Stokesian dynamics framework, and particle motion obtained via dynamic simulation. We present results for a range of volume fractions from dilute to concentrated, and a range of particle-to-cavity size ratios, where an interplay between entropic restriction and hydrodynamic entrainment give rise to novel diffusive behavior. Results are compared to experiments with excellent agreement.

11:14AM H2.00004 Mean and Fluctuating Force Distribution in a Random Array of Spheres, GEORGES AKIKI, THOMAS JACKSON, SIVARAMAKRISHNAN BALACHANDAR, Univ of Florida — In the first part of this talk, we present a novel two-phase continuum model for incompressible fluid-saturated granular flows. A direct forcing immersed boundary method is used to calculate the forces on individual particles for a volume fraction range of [0.1, 0.4] and a Reynolds number range of [10, 625]. The overall drag is compared to several drag laws found in the literature. As for the fluctuation of the hydrodynamic streamwise force among individual particles, it is shown to have a normal distribution with a standard deviation that varies with the volume fraction only. The standard deviation remains approximately 25% of the mean streamwise force on a single sphere. The force distribution shows a good correlation between the location of two to three nearest upstream and downstream neighbors and the magnitude of the forces. A detailed analysis of the pressure and shear forces contributions calculated on a ghost sphere in the vicinity of a single particle in a uniform flow reveals a mapping of those contributions. The combination of the mapping and number of nearest neighbors leads to a first order correction of the force distribution within a cluster which can be used in Lagrangian-Eulerian techniques. We also explore the possibility of a binary force model that systematically accounts for the effect of the nearest neighbors.

1Financial Support has been provided by SEDITRANS, an Initial Training Network of the European Commission’s 7th Framework Programme

11:27AM H2.00005 A continuum theory for two-phase flows of particulate solids: application to Poiseuille flows, DAVIDE MONSORNO, CHRISTOS VARSAKELIS, MILTIADIS V. PAPALEXANDRIS, Université catholique de Louvain — In the first part of this talk, we present a novel two-phase continuum model for incompressible fluid-saturated granular flows. The model accounts for both compaction and shear-induced dilatancy and accommodates correlations for the granular rheology in a thermodynamically consistent way. In the second part of this talk, we exercise this two-phase model in the numerical simulation of a fully-developed Poiseuille flow of a dense suspension. The numerical predictions are shown to compare favorably against experimental measurements and confirm that the model can capture the important characteristics of the flow field, such as segregation and formation of plug zones. Finally, results from parametric studies with respect to the initial concentration, the magnitude of the external forcing and the width of the channel are presented and the role of these physical parameters is quantified.

11:40AM H2.00006 The hydrodynamic lift on a slender, neutrally buoyant fiber in a wall bounded shear flow at small Reynolds number, JOHNSON DHANASEKARAN, DONALD KOCH, Cornell — The hydrodynamic lift velocity of a neutrally buoyant fiber in a simple shear flow near a wall is determined for small fiber Reynolds number. The generalized reciprocal theorem is used to relate the lift velocity to the Stokes flow generated by the fiber. This Stokes velocity field is determined using slender-body theory with the no slip velocity at the wall enforced using the method of images. This study is among the first accurate treatments of the lift on a non-spherical particle and illustrates how particle shape can contribute to separation methods such as those in microfluidic channels or cross-flow filtration processes. To leading order the lift velocity at distances large compared with the fiber length and small compared with the Oseen length is found to be $0.029 \rho \gamma \frac{(\rho_0 \gamma)^2 L^2 d}{\eta (\ln(2L/d))}$ where L and d are the half-fiber length and diameter, gamma is the shear rate and nu is the kinematic viscosity of the fluid. When the fiber separation from the wall is less than the fiber half-length a process of pole vaulting coupled with inertially induced changes of fiber orientation determine the lift velocity.

11:53AM H2.00007 Flipping and scooping of curved 2D fibers in simple shear: the Jeffery equations, DARREN CROWDY, Imperial College London — The dynamical system (or “Jeffery equations”) governing the orbits of a curved rigid two-dimensional fiber in simple shear is derived in analytical form. The study is motivated by the need to understand the dynamics of isolated non-axisymmetric rod-like particles in simple flows for use in suspension modeling. Solutions of the dynamical system are obtained from the “flipping” and “scooping” recently observed in computational studies of three-dimensional fibers using linked rigid rod and bead-shell models [Wang et al., Phys. Fluids, 24. (2012)]. Indeed the equations we derive are expected to be the same ones governing curved 3D slender fibers executing motions in the plane of shear.

12:06PM H2.00008 Periodic dynamics of pairs of sedimenting discs, RAHUL CHAJWA, NARAYANAN MENON1, SRIRAM RAMASWAMY2, TIFR Centre for Interdisciplinary Sciences, 21 Brundavan Colony, Osman Sagar Road, Narsingi, Hyderabad 500 075, India — We study the sedimentation in the Stokes regime of pairs of discs released with a variety of orientations relative to each other and to gravity. The orientation of a settling disc is coupled with the translational degree of freedom. Hydrodynamic interactions between settling disks produces richer dynamics than is possible with sedimenting spheres [S. Jung et al., PRE 74, 035302 (2006), S. Kim, Int J Multiphase Flow 11, 699 (1985), Goldfriend et al. http://arxiv.org/abs/1502.00221]. We demonstrate the classes of dynamics that follow from a variety of initial conditions, but focus on the periodic oscillations in position and orientation that result when two discs are released parallel to each other with their normals coaxial and in the horizontal plane. We report experiments that study the frequency, wavelength, and amplitude of the periodic fluctuation as a function of initial separation between the discs. We analyze the motions within a model that combines the hydrodynamics of single discs with a simplified model of their interaction that includes low order terms of appropriate symmetry. This allows us to examine the initial conditions that demarcate periodic from non-periodic dynamics.

1also at Dept of Physics and Astronomy, Univ of Massachusetts, Amherst MA 01003
2on leave from Department of Physics, Indian Institute of Science, Bangalore 560 012, India
12:19PM H2.00009 Brownian motion of a particle with arbitrary shape1, ELI GUSZ WAJNYRYB, Institute of Fundamental Technological Research, Polish Academy of Sciences, BOGDAN CICHOCKI, Faculty of Physics, University of Warsaw, MARIA L. EKIEL-JEZewsKA, Institute of Fundamental Technological Research, Polish Academy of Sciences — We consider a single Brownian particle of an arbitrary shape, in general non-axisymmetric. Starting from the Smoluchowski equation we develop a new formalism, which allows to determine the particle rotational and translational motion in a much simpler way as this which is based on the Euler angles and Wigner functions. Our approach makes use of the rotational matrix and irreducible tensors. The essential result of our presentation is that using our new formalism, we derive simple explicit analytical expressions for the cross-correlations of the Brownian translational and rotational displacements. The role of the particle mobility center is determined and discussed. No such formulas have been known yet - instead, numerical Brownian simulations have been extensively used. We compare our analytical results with low Reynolds number experiment and numerical simulations performed at the time scales comparable with the characteristic time of the rotational Brownian diffusion.

1E.W. and M.L.E.-J. are supported by the Polish National Science Centre under Grant No. 2012/05/B/ST8/ 03010

Monday, November 23, 2015 10:35AM - 12:45PM –
Session H3 Particle-Laden Flows: Particle Interactions 102 - Katherine Prestridge, Los Alamos National Laboratory

10:35AM H3.00001 Experimental drag histories of shocked spherical particles, KATHERINE PRESTRIDGE, GREG ORLICZ, ADAM MARTINEZ, Los Alamos National Laboratory — The horizontal shock tube (HST) facility at Los Alamos is used to investigate the drag forces on micrometer-sized particles dispersed in air when they are accelerated by a shock. Eight-frame, high-speed particle tracking velocimetry/accelerometry (PTVA) diagnostics are implemented to measure the trajectory of individual particles with high spatial and temporal resolution, and a shadowgraphy system is used to measure the shock position on each image. We present experiments over a range of Reynolds numbers, Mach numbers, particle sizes, and particle densities that explore the drag forces on solid, spherical, non-deforming particles. Experimental drag coefficients are calculated from eight dynamic measurements of particle position versus time, for Mach 1.3 and Mach 1.2 experiments. Experimental results show drag coefficients significantly larger than those predicted by the standard drag model for solid, spherical particles. These results are consistent with measurements made by Rudinger (1970) and Sommerfeld (1985). We will present experimental results and analysis of unsteady drag as a function of particle Reynolds number, Mach number and Stokes number.

1This work was supported by the European Research Council Grant No. ERC-2013-CoG-616186, TRITOS, and by the Swedish Research Council (VR).

10:48AM H3.00002 Interactions of non-spherical particles in simple flows1, MEHDI NIAZI, LUCA BRANDT, Linne FLOW Centre, KTH Mechanics, Stockholm, Sweden, PEDRÓ COSTA, WIM-PAUL BREUGEM, Lab. for Aero and Hydrodynamics, TU-Delft, Delft — The behavior of particles in a flow affects the global transport and rheological properties of the mixture. In recent years much effort has been therefore devoted to the development of an efficient method for the direct numerical simulation (DNS) of the motion of spherical rigid particles immersed in an incompressible fluid. However, the literature on non-spherical particle suspensions is quite scarce despite the fact that these are more frequent. We develop a numerical algorithm to simulate finite-size spherical particles in shear flows to gain new understanding of the flow of particle suspensions. In particular, we wish to understand the role of inertia and its effect on the flow behavior. For this purpose, DNS simulations with a direct-forcing immersed boundary method are used, with collision and lubrication models for particle-particle and particle-wall interactions. We will discuss pair interactions, relative motion and rotation, of two sedimenting spheroids and show that the interaction time increases significantly for non-spherical particles. More interestingly, we show that the particles are attracted to each other from larger lateral displacements. This has important implications for collision kernels.

1We would like to thank the center for compressible multiphase turbulence (CCMT) and acknowledge support from the U.S. Department of Energy, National Nuclear Security Administration, Advanced Simulation and Computing Program.

11:01AM H3.00003 Analytic expressions for first order correction to inviscid unsteady forces due to surrounding particles in a multiphase flow1, SUBRAMANIAN ANANMALAI, S. BALACHANDAR, YASH MEHTA, University of Florida — The various inviscid and viscous forces experienced by an isolated spherical particle situated in a compressible fluid have been widely studied in literature and are well established. Further, these force expressions are used even in the context of particulate (multiphase) flows with appropriate empirical correction factors that depend on local particle volume fraction. Such approach can capture the mean effect of the neighboring particles, but fails to capture the effect of the precise arrangement of the neighborhood of particles. To capture this inherent dependence of force on local particle arrangement a more accurate evaluation of the drag forces proves necessary. Towards this end, we consider an acoustic wave of a given frequency to impinge on a sphere. Scattering due to this particle (reference) is computed and termed “scattering coefficients.” The effect of the reference particle on another particle in its vicinity, is analytically computed via the above mentioned “scattering coefficients” and as a function of distance between particles. In this study, we consider only the first-order scattering effect. Moreover, this theory is extended to compressible spheres and used to compute the pressure in the interior of the sphere and to shock interaction over an array of spheres.

1We would like to thank the center for compressible multiphase turbulence (CCMT) and acknowledge support from the U.S. Department of Energy, National Nuclear Security Administration, Advanced Simulation and Computing Program.

11:14AM H3.00004 Validation of a Hertzian contact model with nonlinear damping1, ADAM SIERAKOWSKI, Johns Hopkins University — Due to limited spatial resolution, most disperse particle simulation methods rely on simplified models for incorporating short-range particle interactions. In this presentation, we introduce a contact model that combines the Hertz elastic restoring force with a nonlinear damping force, requiring only material properties and no tunable parameters. We have implemented the model in a resolved-particle flow solver that implements the Physalis method, which accurately captures hydrodynamic interactions by analytically enforcing the no-slip condition on the particle surface. We summarize the results of a few numerical studies that suggest the validity of the contact model over a range of particle interaction intensities (i.e., collision Stokes numbers) when compared with experimental data.

1This work was supported by the National Science Foundation under Grant Number CBET1335965 and the Johns Hopkins University Modeling Complex Systems IGERT program.
11:27AM H3.00005 Simulating immersed particle collisions: the Devil’s in the details  
EDWARD BIEGERT, BERNHARD VOWINCKEL, ECKART MEIBURG, Univ of California - Santa Barbara — Simulating densely-packed particle-laden flows with any degree of confidence requires accurate modeling of particle-particle collisions. To this end, we investigate a few collision models from the fluids and granular flow communities using sphere-wall collisions, which have been studied by a number of experimental groups. These collisions involve enough complexities—gravity, particle-wall lubrication forces, particle-wall contact stresses, particle-wake interactions—to challenge any collision model. Evaluating the successes and shortcomings of the collision models, we seek improvements in order to obtain more consistent results. We will highlight several implementation details that are crucial for obtaining accurate results.

11:40AM H3.00006 The collision efficiency of cloud droplets in a non-continuum gas  
ANUBHAB ROY, DONALD KOCH, Cornell University — The collision efficiency of bidisperse droplets in a non-continuum gas is determined, subject to the coupled driving forces of differential sedimentation and turbulent shear. A major source of uncertainty in predicting precipitation formation comes from the absence of reliable theoretical predictions for the collision efficiency. Since coalescence requires molecular contact between two drops, it is sensitive to the non-continuum gas flows and van der Waals (vdW) attractions occurring between colliding drops. As two drops interact, the disturbances to the velocity and pressure of the gas induced by the particle motion retard their rate of approach. An especially important aspect of the hydrodynamic interactions between drops (radial $a_1$ and $a_2$) is the lubrication interaction that occurs when the drop separation $r$ is such that $h=r-a_1-a_2<<1$. At such small separations, the relative velocity $v_r$ of the drops along their line-of centers induces a very large $O(w_0/h)$ force. Since the forces driving this relative motion remain finite, $w_0$ will vanish as $h \rightarrow 0$. This leads to a prediction that the collision efficiency would be zero if one considered the interaction of two drops in a continuum gas in the absence of attractive colloidal forces. Therefore, it is clearly essential to include an accurate description of all the relevant near field interactions to accurately predict the true collision efficiency. We will treat the coupled sedimentation and turbulent shear effects governing cloud droplets, treated independently in previous works. We show that it is the non-continuum effects rather than vdW that primarily allows finite collision efficiency for drop sizes $a_1>5 \mu m$ at atmospheric conditions.

11:53AM H3.00007 Is there solid-on-solid contact when spheres collide in a fluid?  
NARAYANAN MENON, U. Massachusetts, Amherst, USA and TCIS, Hyderabad, India, SUMIT BIRWA, G. RAJALAKSHMI, RAMA GOVINDARAJAN, TCIS, Hyderabad, India — A solid sphere colliding with another sphere or a wall within a fluid reverses its velocity and bounces back when it is launched with a Stokes number above a critical value, $St>10$. Previous experiments showed that $St$ is only weakly dependent on the material or roughness of the sphere, but did not have the spatial or temporal resolution to determine whether solid impact occurs in the collision. A calculation [1] in the lubrication approximation shows that it is possible for an elastic sphere to rebound under fluid forces alone, without contact between the solids. We report experiments which exploit electrical contact between a sphere and wall to study the collision with high temporal resolution. We find unambiguously that there is solid-on-solid contact when the sphere rebounds from a collision. Analysis of the time of contact, and the time between consecutive impacts, indicates that even when there is impact, fluid viscosity is the dominant dissipative mechanism. The exception is for very smooth spheres, at stokes numbers just above $St_c$. We present calculations with the incompressible Navier-Stokes equations to assess viscous dissipation and pressure effects in the collision.


12:06PM H3.00008 Experimental Exploration of Electrostatic Charge on Particle Pair Relative Velocity in Homogeneous and Isotropic Turbulence  
ADAM HAMMOND, ZHONG-WANG DOU, ANJAN TRIPATHI, ZACH LIANG, HUI MENG, University at Buffalo - SUNY — Study of droplet collision and cloud formation should consider the effects of both turbulence and electrostatic charge on particle dynamics. We present the first experimental observation of radial relative velocity (RV) of charged particles in homogeneous and isotropic turbulence (HIT). Charges on particles were generated through triboelectric effect between the inner wall of the chamber and the particles. To measure charge distribution, a particle-laden head-on impinging flow mimicking our HIT chamber conditions was built and holographic particle tracking was applied to quantify particle charges by measuring their displacements in an electric field. Particles were observed to have opposite charges. Next, in our HIT chamber, we measured particle RV by a novel 4-frame particle tracking velocimetry technique with and without charges on particles, wherein charges were neutralized by coating the interior of the HIT chamber with conductive carbon paint. We compared RV under the same turbulence conditions between charged particles and neutral particles and observed that when particles were oppositely charged, their mean inward RV increased at small separation distances. This result is consistent with recent theory and simulations (Lu and Shaw, Physics of Fluids, 2015).

12:19PM H3.00009 Correcting velocity and volume-fraction calculations in two-way-coupled, particle-laden flow simulations  
PETER IRELAND, Cornell University, JESSE CAPECCELATRO, University of Illinois Urbana-Champaign, RODNEY FOX, Iowa State University, OLIVIER DESJARDINS, Cornell University — In many flows, the motion of the carrier phase is altered by the presence of inertial particles. To alleviate the computational demands associated with resolving the boundary layers around these particles, volume-filtering is often applied to the underlying flow field, and model equations are solved for the forces on the particles. These model equations involve terms which depend on the fluid properties at the particle center in the absence of the disturbance induced by the particle (i.e., the ‘undisturbed fluid properties’). In a two-way-coupled simulation, however, we generally only have access to fluid properties after the particle-induced disturbance (i.e., the ‘disturbed fluid properties’). Using the disturbed fluid properties in the particle model equations leads to an under-prediction of the drag on the particles and an over-prediction of the particle settling velocity. We introduce analytical corrections to alleviate this issue for low particle Reynolds numbers, allowing us to recover undisturbed fluid properties from the disturbed fluid field, and thereby providing more accurate predictions of the particle velocity and drag. We show comparisons between the results with and without the corrections in both uniform Stokes flows and cluster-induced turbulent flows.

DENNIS DUNN, KYLE SQUIRES, Arizona State University — Modeling dispersions of particles in multiphase flows is especially challenging in gas-solid suspensions. Lagrangian methods are suitable for dilute particle mediums, but are not cost effective at denser concentrations and impose additional modeling challenges. A moderately dense particle phase is neither sufficiently dense for a continuum limit assumption (collisional equilibrium) nor sufficiently dilute for a Lagrangian method, and resides in the intermediate regime under consideration in the current work. A quadrature-based moment method (QBMM) is chosen to simulate a particle-laden turbulent channel flow considering inter-particle collision effects. In quadrature-based approaches similarly behaving particles may be grouped together and treated in a stochastic manner within an Eulerian framework. Specifically, the Conditional Quadrature Method of Moments (CQMO M) is implemented to discretize a fully 3-D velocity space and capture particle trajectory crossing (PTC). This has the potential for large computational savings as compared to Lagrangian methods, especially when dense collisions are prominent. The probability density function is discretized with a two-point-quadrature in each dimension – the minimum requirement to capture PTC and enforce collisions. Predictions of the channel flow demonstrate that the collision treatment leads to the expected effects (e.g., redistribution of kinetic energy) and also offer improved accuracy relative to simpler approaches.

This work was supported by the National Science Foundation through a Collaborative Research Grant CBET-0967407
1 An undergraduate lab on measuring fluid viscosity using a miniature ball drop device. JAY TANG, Brown University — I would like to describe measurement of fluid viscosity using a small ball drop device. It requires as little as 100 microliters of fluid. Each measurement can be performed in seconds. Through simple experimentation, students observe fluid flow confined in a thin cylindrical tube. They analyze forces and torques on a tiny ball falling and rolling down in an inclined tube. They gain practice in observing and identifying sources of errors and variability in their measurements beyond those indicated by standard error bars. The experiment is designed to yield reliable viscosity values by operating at properly chosen tilt angles and with calibration using well-characterized fluids such as mixtures of glycerol and water. The technique is also useful in research and technological applications as the device is easy to assemble and it allows the measurement of viscosity even when the fluid samples are too small to measure using most commercial viscometers or rheometers.

1 Work Partially Supported by the NSF Fluid Physics Program (Award number CBET 1438033).

10:48AM H4.00002 FlowGo: An Educational Kit for Fluid Dynamics and Heat Transfer, DOMINIC GURI, MERREDITH PORTSMORE, ERICA KEMMERLING, Tufts University — The authors have designed and prototyped an educational toolkit that will help middle-school-aged students learn fundamental fluid mechanics and heat transfer concepts in a hands-on play environment. The kit allows kids to build arbitrary flow rigs to solve fluid mechanics and heat transfer challenge problems. Similar kits for other engineering fields, such as structural and electrical engineering, have resulted in pedagogical improvements, particularly in early engineering education, where visual demonstrations have a significant impact. Using the FlowGo kit, students will be able to conduct experiments and develop new design ideas to solve challenge problems such as building plant watering systems or modeling water and sewage reticulation. The toolkit consists of components such as tubes, junctions, and reservoirs that easily snap together via a modular, universal connector. Designed with the Massachusetts K-12 science standards in mind, this kit is intended to be affordable and suitable for classroom use. Results and user feedback from students conducting preliminary tests of the kit will be presented.

11:01AM H4.00003 A formal derivation for the Blasius similarity solution for flat-plate boundary layer, HAO LIN, Department of Mechanical and Aerospace Engineering, Rutgers, The State University of New Jersey — The Blasius solution is a classical solution for a laminar boundary layer attached to a semi-infinite flat plate. The key of the solution strategy is to reduce the boundary layer equations, which are PDEs, to a set of ODEs, using a similarity variable transform. Conceptually, the similarity suggests that the velocity profile in each transverse cross-section appears “self-similar”. In many classical text books and typical classroom lectures on fluid mechanics, the existence of the similarity solution is argued heuristically. The similarity variable is defined a priori so as to collapse the PDEs. It appears somewhat mystical that the PDEs can be perfectly reduced via such an approach. Here we present a rigorous derivation for the existence of a similarity solution, which naturally arises from the fact that there is no apparent streamwise length scale for a semi-infinite plate. Conversely, a similarity solution cannot exist if the plate size is finite. This derivation can be useful in fluids education, in topics including similarity, scaling arguments, and boundary layer theory.

11:14AM H4.00004 Experimental Approach to Teaching Fluids, CATALINA STERN, Universidad Nacional Autonoma de Mexico — For the last 15 years we have promoted experimental work even in the theoretical courses. Fluids appear in the Physics curriculum of the National University of Mexico in two courses: Collective Phenomena in their sophomore year and Continuum Mechanics in their senior year. In both, students are asked for a final project. Surprisingly, at least 85% choose an experimental subject even though this means working extra hours every week. Some of the experiments were shown in this congress two years ago. This time we present some new results and the methodology we use in the classroom.

11:27AM H4.00005 Design and Testing of an Educational Water Tunnel, SRINIVAS KOSARAJU, Northern Arizona University — A new water tunnel is being designed and tested for educational and research purposes at Northern Arizona University. The university currently owns a wind tunnel with a test section of 12in X 12in X 24in. However, due to limited size of test section and range of Reynolds numbers, its application is currently limited to very few experiments. In an effort to expand the educational and research capabilities, a student team is tasked to design, build and test a water tunnel as a Capstone Senior Design project. The water tunnel is expected to have a test section of 8in X 8in X 18in. and be able to test up to Re = 30,000. The water tunnel will be designed to accommodate multiple experiments for drag and lift studies. It will also have dies of different colors to study the streamlines and vortex shedding from the surfaces. Numerical models will be used to optimize the flow field inside the test section before building the physical apparatus.

11:40AM H4.00006 Ups and downs of using “kitchen sink” experiments in an introductory fluid mechanics class, NIGEL KAYE, Clemson University — Both positive and negative experiences from two semesters of using take home “kitchen sink” experiments in an introductory civil engineering fluid mechanics class are reported. Four different experimental assignments were given each semester to groups of four students. The students were tasked with using common household equipment to measure various properties of fluids or fluid flows. These included the density of cooking oil, the exit velocity from a garden hose, and the mass flux of air from a compressed air can. Students were given minimal guidance on how to do the measurements and each measurement had to be done in at least two different ways. The labs were used to relate their course work to everyday situations and was also used as a platform for discussing experimental uncertainty and error propagation in calculations. In general the students successfully completed each task using at least one method. Finding a second method sometimes proved problematic. The presentation will discuss the logistics of running the program and the positive and negative aspects from the instructor viewpoint. A summary of student feedback on the labs will also be presented. Links to resources for those interested in implementing such a program will be provided.
11:53AM H4.00007 Geophysical Fluid Dynamics Laboratory Open Days at the Woods Hole Oceanographic Institution

The Geophysical Fluid Dynamics Laboratory at the Woods Hole Oceanographic Institution hosted 10 hands-on demonstrations and displays, with something for all ages, to share the excitement of fluid mechanics and oceanography. The demonstrations/experiments spanned as many fluid mechanics problems as possible in all fields of oceanography and gave insight into using fluids laboratory experiments as a research tool. The chosen experiments were ‘simple’ yet exciting for a 6 year old child and a physics professor. How does an avalanche happen? How does a bath tub vortex form? What happens to waves when they break? How does a hurricane move? Hands-on activities in the fluid dynamics laboratory helped students of all ages in answering these and other intriguing questions. The laboratory experiments/demonstrations were accompanied by ‘live’ videos to assist in the interpretation of the demonstrations. Posters illustrated the oceanographic/scientific applicability and the location on Earth where the dynamics in the experiments occur.

1Support was given by the WHOI Dוהל Chair in Education.

12:06PM H4.00008 Computational simulations of frictional losses in pipe networks confirmed in experimental apparatuses designed by honors students

Vlado Nedialkov, Martin Wosnik, University of New Hampshire — OpenFOAM has gained significant popularity in academe and industry, but is still not widely introduced to CFD novices e.g., undergraduate students. This is likely due to the steep learning curve of the software. A relatively short tutorial was developed to introduce students to the basic features of OpenFOAM, allowing them to modify and create simulations, and to better understand other online resources. The tutorial has been successfully presented to students working on undergraduate capstone projects at the University of New Hampshire and parts of it were presented at a tech-camp for K-12 students.

12:32PM H4.00010 The art of scientific writing

Mohamed Gad-el-Hak, Virginia Commonwealth University — The humanities teach students how to learn and communicate. Science teaches why everything works. Engineering teaches how to make things work. But scientists and engineers need to communicate their ideas amongst themselves as well as to everyone else. A newly developed technical writing course is outlined. In the class, offered to senior undergraduate and beginning graduate students, we read numerous short novels, essays, and op-eds. Some of the reading materials are technical but many are not. The students also have weekly writing assignments. When the first assignment is returned to the students with a grade of 20–30%, their first reaction is, “how come I did not receive my usual 80–90%?” I retort, “you reach that level only when your essay is ready to be published in The New York Times.” What is emphasized in the class is the process of creating something to write about, researching that something, expressing ideas coherently and comprehensibly, then endlessly editing the essay. The elective class has been offered three times thus far, all of its available seats are always filled, the students’ evaluations have been outstanding, and the improvements in the students’ ability to write by the end of the semester is quite impressive.

Monday, November 23, 2015 10:35AM - 12:45PM — Session H5 Jets II: Impinging

10:35AM H5.00001 On the effect of fractal geometric parameters on the heat transfer features of circular impinging jets

Tommaso Astarita, Universita degli Studi di Napoli Federico II — Several solutions have been proposed over the last decades to increase the heat transfer rate of impinging jets. In all cases the heat transfer enhancement is obtained by exciting/altering the structure and organization of large scale turbulence, which is widely recognized to be the main agent in heat and mass transfer mechanism of impinging jets. Tampering with the large coherent turbulent structures is the key to achieve a significant heat transfer enhancement. In a recent work we demonstrated the effectiveness of fractal turbulence in this sense. Its effect is such that the heat transfer rate increases up to 63% in the stagnation region with respect to the well-known circular jet under the same power input. However, a systematic analysis of the effect of the singular geometric parameters of the fractal grid (such thickness ratio and length ratio) onto the spatial distribution of the Nusselt number has not been proposed yet. In this work we propose the analysis of the heat transfer enhancement produced by a class of turbulence promoters located in correspondence of the nozzle exit section of a circular jet. The upward shift of the turbulence intensity profile due to the blockage effect induced by the growing shear layer is discussed in terms of heat transfer enhancement.

10:48AM H5.00002 The effect of confinement on the development of an axisymmetric wall-jet in confined jet impingement

Tianqi Guo, Matthew J. Rau, Pavlos P. Vlachos, Suresh V. Garimella, Purdue University — An experimental study of a confined developing axisymmetric wall-jet is reported. The wall-jet is formed downstream of a circular, confined, impinging jet of water. Stereo particle image velocimetry (SPIV) experiments are conducted at three different nozzle-to-plate spacings (2, 4 and 8 jet diameters) and across Reynolds numbers ranging from 1000 to 9000. Special attention is paid to the development of the wall-jet. The growth rate of the boundary layer thickness, decay rate of the local maximum velocity, and velocity profile scaling for both the inner- and outer-layer are investigated. Measurements are obtained with a maximum spatial resolution of 25 μm and a temporal resolution of 750 Hz. Both ensemble-averaged and instantaneous time-resolved three-component, two-dimensional (3C-2D) flow fields are obtained and analyzed. The upper confinement plate is found to limit the supply of ambient liquid for both the impinging-jet and wall-jet entrainment, and thus significantly influences the wall-jet development; the growth and decay rate of the wall-jet are shown to be greatest at the smallest confinement height. The influence of these confining effects on recirculation patterns and coherent-structure evolution is also reported. These flow field measurements and analyses will serve to inform a variety of practical applications that use impinging jets.
heat transfer in stagnation region due to combined effect of oscillation in impingement position caused by large vortical structures and strong dimensional channel height \( H/L \). Synthetic Jets nozzle-to-plate distances. At low nozzle-to-plate distance \( H/D \) the twin configuration and several nozzle-to-plate distances have been investigated. The time-averaged behavior of the all velocity components Particle Image Velocimetry (PIV) at Reynolds and Strouhal numbers equal to 5100 and 0.024, respectively. Different inter-axes distances for — The behavior of impinging single synthetic jet and twin circular synthetic jets in phase opposition is experimentally investigated by using wettability engineered surfaces. Using wettability engineered surfaces, we demonstrate pump-less and directional transport of capillary jet on a flat surface. Spatial contrast of surface energy and a wedge-shape geometry of the wettability confined track on the substrate facilitate formation of instantaneous spherical bulges upon jet impingement; these bulges are transported along the superhydrophobic tracks due to Laplace pressure gradient. Critical condition warranted for formation of liquid bulge along the varying width of the superhydrophobic track is calculated analytically and verified experimentally. The work throws light on novel fluid phenomena of unidirectional jet impingement on wettability confined surfaces and provides a platform for innovative liquid manipulation technique for further application. By varying the geometry and wettability contrast on the surface, one can achieve volume flow rates of \( \sim O(100 \mu L/sec) \) and directionally guided transport of the jet liquid, pumplessly at speeds of \( \sim O(10cm/sec) \).

11:01AM H5.00003 Large Eddy Simulation of a cooling impinging jet to a turbulent crossflow1. MICHAEL GEORGIOU, MLIATIDIS PAPAELXANDRIS, Universite catholique de Louvain — In this talk we report on Large Eddy Simulations of a cooling impinging jet to a turbulent channel flow. The impinging jet enters the turbulent stream in an oblique direction. This type of flow is relevant to the so-called “Pressurized Thermal Shock” phenomenon that can occur in pressurized water reactors. First we elaborate on issues related to the set-up of the simulations of the flow of interest such as, imposition of turbulent inflows, choice of subgrid-scale model and others. Also, the issue of the commutator error due to the anisotropy of the spatial cut-off filter induced by non-uniform grids is being discussed. In the second part of the talk we present results of our simulations. In particular, we focus on the high-shear and recirculation zones that are developed and on the characteristics of the temperature field. The budget for the mean kinetic energy of the resolved-scale turbulent velocity fluctuations is also discussed and analyzed.

11:14AM H5.00004 Drop Characteristics of non-Newtonian Impinging Jets at High Generalized Bird-Carreau Jet Reynolds Numbers1. PAUL E. SOJKA2, NEIL S RODRIGUES3, M.J. Zucrow Laboratories, Purdue University — The current study investigates the drop characteristics of three Carboxymethylcellulose (CMC) sprays produced by the impingement of two liquid jets. The three water-based solutions used in this work (0.5 wt.-% CMC-7MF, 0.8 wt.-% CMC-7MF, and 1.4 wt.-% CMC-7MF) exhibited strong shear-thinning, non-Newtonian behavior - characterized by the Bird-Carreau rheological model. A generalized Bird-Carreau jet Reynolds number was used as the primary parameter to characterize the drop size and the drop velocity, which were measured using Phase Doppler Anemometry (PDA). PDA optical configuration enabled a drop size measurement range of approximately 2.3 to 116.2 \( \mu m \). 50,000 drops were measured at each test condition to ensure statistical significance. The arithmetic mean diameter \( \bar{D}_{10} \), Sauter mean diameter \( \bar{D}_{s} \), and mass median diameter \( \bar{D}_{MD} \) were used as representative diameters to characterize drop size. The mean axial drop velocity \( U_{2-max} \), along with its root-mean square \( U_{2-rms} \) were used to characterize drop velocity. Indeed, measurements for all three CMC liquids and reference DI water sprays seemed to follow a single curve for \( \bar{D}_{10} \) and \( \bar{D}_{MD} \) drop diameters in the high generalized Bird-Carreau jet Reynolds number range considered in this work (9.21E+03 < Reجن-gen-BC < 2.81E+04).

11:27AM H5.00005 Proper Orthogonal Decomposition Analysis of Turbulent Jet Impingement on Rib-roughened Surface. PRASANTH ANAND KUMAR LAM, ARUL PRAKASH KARAIYAN, Indian Institute of Technology Madras, Chennai, THERMO-FLUID DYNAMICS LABORATORY TEAM — A Proper Orthogonal Decomposition (POD) analysis on turbulent flow dynamics of confined slot jet impinging on rib-roughened surface is numerically investigated. The data for POD analysis has been obtained by solving mass, momentum and energy equations in Cartesian framework using Streamline Upwind/Petrov-Galerkin Finite element method. Further, turbulent kinetic energy \( k \) and its dissipation rate \( \varepsilon \) are modeled using standard \( k-\varepsilon \) turbulence model with standard wall functions. POD is applied to computational data for a wide range of Reynolds number \( Re = 5000 - 30000 \) and non-dimensional channel height \( H/L = 0.5 - 4.0 \) to reveal large scale vortical structures in the flow field. The simulated results demonstrate a better understanding on effect of turbulence and its influence on individual vortical structures for enhancement of heat transfer. The enhancement of heat transfer in impingement region due to rib adverse pressure gradient strength and the potential core-like region extension. Indeed, the turbulence distribution shows a region characterized by low values of turbulence intensity in the potential core of the opposing small channels. A systematic set of experiments has been undertaken, to investigate the hydrodynamic characteristics, to develop predictive models, and enable comparisons with other contactors. Drop size distribution and mixing intensity will be investigated for liquid-liquid mixtures as a function of various parameters using high speed imaging and conductivity probes.
12:19PM H5.00009 Dynamics of Bouncing-vs.-Merging Responses in Jet Collision. MINGLEI LI, Tsinghua University, ABHISHEK SAHA, DELIN L. ZHU, Princeton University, CHAO SUN, Tsinghua University, CHUNG K. LAW, Princeton University — Collision of two fluid masses is a common natural and industrial phenomenon. Many kinds of noncoalescence phenomena of collisional fluid masses, such as droplet & droplet, droplet & liquid film, have been studied, and the dynamics of the gas layer between the colliding liquid surfaces was found to play a crucial role. However, many fluid mass collision processes are nonstationary, making it difficult to study this air layer dynamics in detail. Jet bouncing can be in a stationary state with a geometrically simple gas layer, providing an ideal system to investigate the dynamics of the air film between the colliding interfaces. In this work, we observe an entire suite of possible jet collision outcomes of (soft) merging, bouncing and (hard) merging with increasing impact inertia. These transitions between these different regimes are characterized through scaling analysis by considering the competing effects of impact inertia, surface tension and viscous thinning of the interfacial air-gap leading to activate the van der Waals force to effect merging.

12:32PM H5.00010 Characterization of an impinging jet into porous media1. CONG WANG, Caltech, SALWAN ALHANI, West LA college, MORTEZA Gharib, Caltech — In this work, characteristic behavior of a liquid jet into porous hydrophobic / hydrophilic particle media is investigated. In porous media, the capillary effect becomes significant, especially when the jet Reynolds Number is low. To analyze the cavity creation phenomena, the effect of jet’s diameter, speed and acceleration as well as particles’ size are carefully studied. Such knowledge of fluid behavior will provide guidance for medicine injection process.

1This work is supported by Caltech GALCIT STEM program.

Monday, November 23, 2015 10:35AM - 12:45PM – Session H6 CFD: Reactive Flows II

10:35AM H6.00001 A New Method for Large Eddy Simulation of Turbulent Premixed Combustion. SEUNG HYUN KIM, The Ohio State University — A new method for large eddy simulation (LES) of turbulent premixed combustion is presented. The method is based on the front propagation formulation (FPF) of filtered reaction rates. In premixed combustion LES where a filter scale is typically taken as grid spacing, the spurious propagation of filtered flame fronts can occur due to under-resolved reaction zones. The FPF method avoids this spurious propagation by discretely preserving the total reaction rates on computational grids. The method not only recovers the flamelet limit when turbulence is not strong enough to perturb inner structures of the flame fronts, but also allows for the broadening of filtered flame fronts by turbulence. The FPF method is applied to LES of laboratory flames. The analysis and validation of the method will be presented.

10:48AM H6.00002 Analysis of entropy generation in turbulent reacting flows using large eddy simulation. MEHDI SAFARI, Miami University — The entropy transport equation is considered in large eddy simulation (LES) of turbulent reacting flows. This equation includes irreversible losses by entropy production due to viscous dissipation, heat conduction, mass diffusion and chemical reaction, all of which appear as unclosed terms. The closure is provided by entropy filtered density function (En-FDF) which includes the effect of chemical reaction in a closed form. An exact transport equation is developed for the En-FDF. The transport equation for En-FDF is modeled by a set of stochastic differential equations. The modeled En-FDF transport equation is solved by a Lagrangian Monte Carlo method. The methodology is applied to turbulent nonpremixed jet flames to analyze local entropy generation effects. Various modes of entropy generation are obtained and analyzed.

11:01AM H6.00003 Subfilter scalar variance models for LES of premixed turbulent flames. GUILLAUME BLANQUART, SIMON LAPOINTE, TOMAS TUSSIE, Caltech — The subfilter scalar variance plays an important role in large eddy simulations of turbulent reacting flows. It is not available in the simulations and needs to be modeled. Subfilter scalar variance models often take the form of a constant coefficient multiplying the square of the filter width and the square of the gradient of the filtered progress variable (referred to as the mixing model). Variance models are first studied a priori using results from constant density DNS of scalar mixing in isotropic turbulence. Scalar variance models based on a generalized Taylor expansion are accurate for small filter widths but errors arise in the inertial subrange. Results suggest that a constant coefficient computed from an assumed Kolmogorov spectrum is often sufficient to predict the subfilter scalar variance in homogeneous isotropic turbulence. The analysis is then extended to variable density reacting flows using DNS of turbulent n-heptane/air premixed flames at varying Karlovitz numbers. Results from homogeneous isotropic turbulence still hold when taking into account the change in the Kolmogorov length scale across the flame. The optimal coefficient in the mixing model varies between the two limits of small filter widths and assumed Kolmogorov spectrum.

11:14AM H6.00004 Computational Study of the Effect of Compositional Inhomogeneity of Fuel Streams on Turbulent Jet Flames. MICHAEL E. MUELLER, BRUCE A. PERRY, Princeton University, ASSAAD R. MASRI, The University of Sydney — A new piloted turbulent jet burner has been developed at The University of Sydney to investigate how inhomogeneous partially premixed inlet conditions affect flame structure and stability characteristics [S. Meares, A.R. Masri, Combust. Flame 161 (2014) 484-495]. Compositional inhomogeneity at the inlet is achieved by recessing a central tube that separates the fuel stream and a surrounding annular air flow to allow for a controlled amount of mixing before the gases reach the nozzle exit. In this work, Large Eddy Simulation of the burner is performed using a conventional nonpremixed flamelet/progress variable model. The geometry is divided into three separately computed domains: fully developed pipe/annulus flow, pipe flow in the region of fuel/air mixing upstream of the nozzle, and the turbulent flame. The results for two recess distances of the central tube (inhomogeneous fuel inlet and effectively homogeneous fuel inlet) are compared to recent experimental measurements. Discrepancies between the simulation and experiment show that premixed combustion is dominant only for the inhomogeneous case at the base of the flame. Sensitivities to grid resolution in both the upstream mixing domain and the turbulent flame domain as well as pilot conditions are assessed.
11:27AM H6.00005 Flamelet modeling of differential molecular diffusion in CO/H2 and ethylene DNS flames\textsuperscript{1}. CHAO HAN, Purdue University, DAVID LIGNELL, Brigham Young University, EVATT HAWKES, University of New South Wales, JACQUELINE CHEN, Sandia National Laboratories, HAIFENG WANG, Purdue University — A class of consistent differential diffusion models suitable for flamelet modeling of turbulent non-premixed combustion is developed recently. In this work, these differential diffusion models are further validated in two DNS temporally evolving jet flames with two different fuels, Syngas and ethylene. The dependence of differential diffusion on the Reynolds number, which is missing in previous models, is incorporated into the new models, and is based on a limiting analysis of the behaviors of differential molecular diffusion at the limits of small and large Reynolds numbers. The performances of the models are thoroughly examined in the two DNS flames. The effect is studied of the Reynolds number, Damkohler number, and Lewis number on the differential molecular diffusion in the temporally evolving jet flames.

\textsuperscript{1}Acknowledgment is made to the Donors of the American Chemical Society Petroleum Research Fund for support of this research.

11:40AM H6.00006 A Two Mixture Fraction Flamelet Model for Large Eddy Simulation of Turbulent Jet Flames with Inhomogeneous Inlets, BRUCE A. PERRY, MICHAEL E. MUELLER, Princeton University, ASSAAD R. MASRI, The University of Sydney — A revised flamelet/progress variable (FPV) model in which two mixture fractions are defined has been developed to address the known limitation that single mixture fraction FPV models require there to be a single, compositionally uniform fuel stream. The revised model is applied in a large eddy simulation of a new turbulent jet burner with inhomogeneous partially premixed inlet conditions that was developed at the University of Sydney [S. Meares, A.R. Masri, Combust. Flame 161 (2014) 484-495]. Compositional inhomogeneity at the inlet is achieved by recessing a central tube that separates the fuel stream and a surrounding annular air flow to provide controlled mixing upstream of the nozzle. The first mixture fraction characterizes the mixing between the jet and surrounding air. The second mixture fraction tracks mixing of fuel and air upstream of the nozzle and defines the fuel side boundary condition for solution of the 1D flamelet equation in terms of the first mixture fraction. The predictions using both the single mixture fraction and the two mixture fraction FPV models are compared to recent experimental results. It is shown that use of a single mixture fraction is insufficient to accurately capture the structure of the flame with inhomogeneous inlet conditions.

11:53AM H6.00007 Numerical Study of Flame Stabilization Mechanism in a Premixed Burner with LES Non-adiabatic Flamelet Approach, YIHANG TANG, MALIK HASSANALY, VENKAT RAMAN, Department of Aerospace Engineering, University of Michigan — In the development of highly efficient gas turbine combustion systems, using high-hydrogen-content fuels is a new solution that limits pollutant emissions but also triggers flame stabilization issues. The revised model is applied in a large eddy simulation of a new turbulent jet burner with inhomogeneous partially premixed inlet conditions that was developed at the University of Sydney [S. Meares, A.R. Masri, Combust. Flame 161 (2014) 484-495]. Compositional inhomogeneity at the inlet is achieved by recessing a central tube that separates the fuel stream and a surrounding annular air flow to provide controlled mixing upstream of the nozzle. The first mixture fraction characterizes the mixing between the jet and surrounding air. The second mixture fraction tracks mixing of fuel and air upstream of the nozzle and defines the fuel side boundary condition for solution of the 1D flamelet equation in terms of the first mixture fraction. The predictions using both the single mixture fraction and the two mixture fraction FPV models are compared to recent experimental results. It is shown that use of a single mixture fraction is insufficient to accurately capture the structure of the flame with inhomogeneous inlet conditions.

12:06PM H6.00008 Comparison of Flamelet Models with the Transported Mass Fraction Approach for Supersonic Combustion, WENHAI LI, KEN ALABI, TTC Technologies, Inc., Centerearth, NY 11720 USA, FOLUSO LADEINDE, Stony Brook University, Stony Brook, NY 11794-2300 USA — In this study, two fully compressible RANS, LES, and combined RANS/LES flow solvers AEROFLO and VULCAN, both of which were originally developed by the United States Department of Defense but have since been significantly enhanced and commercialized by our organization, are used to investigate the accuracy of CFD: Computational Methods and Modeling of Multiphase Flows IV

12:19PM H6.00009 Modeling studies of a turbulent pulsed jet flame using LES/PDF\textsuperscript{1}. PEI ZHANG, HAIFENG WANG, Purdue University — The combustion field in a pulsed turbulent jet flame is studied using an advanced large eddy simulation (LES) / probability density function (PDF) method. Measurement data with a joint OH-PLIF/OH* chemiluminescence/LDV system are available including the temporal series of the axial velocity and planar OH images. A time-dependent inflow condition is specified based on the measurement data. A direct comparison of the mean and rms velocities from the calculations and from the measurement shows a satisfactory prediction of the flow fields by using the employed modeling methods. The predicted OH mass fractions are compared qualitatively with the measured OH images at selected temporal and spatial locations. The comparison shows a good agreement. Conditional quantities and flame index are extracted from the simulations to examine the bimodal and multi-regime combustion dynamics in the flame.

\textsuperscript{1}This paper is based upon work supported by the National Science Foundation under Grant No. CBET-1336075.

12:32PM H6.00010 Interaction of Thermodiffusive Instabilities and Turbulence in Lean Hydrogen/Air Mixtures using Tabulated Chemistry, JASON SCHLUIP, GUILLAUME BLANQUART, Caltech — The combustion of lean hydrogen mixtures is prone to thermodiffusive instabilities due to the strongly non-unity fuel Lewis number. Simulations of the combustion process can aid in designing new burners to reduce operating risks associated with thermodiffusive instabilities; however, direct numerical simulations of large scale burners with detailed chemistry mechanisms are prohibitively expensive. The significant simulation time required due to the high number of time steps is the main drawback of this approach. In this work, a chemistry table, created with one-dimensional flames, is used to reduce the simulation cost. Direct numerical simulations of turbulent combustion with lean hydrogen/air mixtures are performed. Both statistically planar and spherically expanding flames are considered, and the turbulence level varies from laminar to fully turbulent flow conditions. The chosen equivalence ratio displays thermodiffusive instabilities in the wrinkled flame front. The influence of turbulence intensity on the flame instabilities is explored, and the results are compared to previous studies to determine the adequacy of the tabulated chemistry method for this set of simulation parameters.

Monday, November 23, 2015 10:35AM - 12:45PM – Session H7 CFD: Computational Methods and Modeling of Multiphase Flows IV 107

- Dimitrios Pavlidis, Imperial College London
10:35AM H7.00001 Three-dimensional numerical simulations of three-phase flows. DIMITRIOS PAVLIDIS, ZHIZHUA XIE, PABLO SALINAS, CHRIS PAIN, OMAR MATAR, Imperial College London — The objective of this study is to investigate the fluid dynamics of three-dimensional three-phase flow problems, such as droplet impact on a gas-liquid interface and bubble rising through a liquid-liquid interface. An adaptive unstructured mesh modelling framework is employed here to study three-phase flow problems, which can modify and adapt unstructured meshes to better represent the underlying physics of multiphase problems and reduce computational effort without sacrificing accuracy. The numerical framework consists of a mixed control volume and finite element formulation, a volume of fluid type method for the interface capturing based on a compressive control volume advection method and second-order finite element methods, and a force-balanced algorithm for the surface tension implementation, minimising the spurious velocities often found in such flow simulations. The surface tension coefficient decomposition method has been employed to deal with surface tension pairing between different phases via a compositional approach. Numerical examples of some benchmark tests and the dynamics of three-phase flows are presented to demonstrate the ability of this method.

1EPSRC Programme Grant, MEMPHIS, EP/K0039761/1

10:48AM H7.00002 A Scalable Parallel Fast Marching Method, YAJING GALE, MARCUS HERRMANN, Arizona State University — The fast marching method is an efficient strategy to solve the Eikonal equation with broad applications in computational fluid dynamics. However, the traditional fast marching method is a purely sequential algorithm and thus not straightforward to parallelise. In this presentation, four parallel fast marching methods are discussed: the non-blocking parallel fast marching method (NB-PFMM), the blocking parallel fast marching method (B-PFMM), the extended domain-blocking parallel fast marching method (DB-PFMM), and the layered-blocking parallel fast marching method (LB-PFMM). When combined with proper domain decomposition approaches, these methods are not only efficient but can scale over a wide range of processor counts. The applicability and performance of the different parallel fast marching methods are presented and compared as applied to a variety of test cases.

1Support from grant NSF-CBET1054272 is gratefully acknowledged.

11:01AM H7.00003 A framework for embedding molecular-level information in continuum-scale simulations of interfacial flows, EDWARD SMITH, PANAGIOTIS THEODORAKIS, ERICH MÜLLER, RICHARD CRASTER, OMAR MATAR, Imperial College London — Molecular dynamics provides a means of resolving the contact-line paradox. The price to pay for this insight is computational, with droplet simulations limited to the nanoscale. In order to model problems of engineering interest, the molecular contact line must be abstracted and included as part of a continuum scale simulation. Coupling, using dynamic molecular simulation in place of empirical or approximate closure relations, provides a means of doing just this. Molecular simulation of two-phase Couette flow can reproduce the key features of the moving contact line. This sheared liquid bridge has the advantage that a steady state can be obtained, providing an unlimited source of data for statistical analysis. In this talk, we will present highlights from molecular dynamics simulation of the moving contact line. Using interface tracking, the dynamics of the contact line are examined, with results compared to published experimental studies. Good agreement is observed despite the difference in scale between the molecular model and experiments. Potential applications of this method are discussed, including coupled simulation which incorporates the molecular detail for surfactant-driven spreading.

1EPSRC Platform Grant (MACiPh) EP/L020564/1

11:14AM H7.00004 Attenuation of numerical artefacts in the modelling of fluid interfaces, FABIEN EVRARD, BEREND G.M. VAN WACHEM, FABIAN DENNER, Imperial College London — Numerical artefacts in the modelling of fluid interfaces, such as parasitic currents or spurious capillary waves, present a considerable problem in two-phase flow modelling. Parasitic currents result from an imperfect evaluation of the interface curvature and can severely affect the flow (Denner and van Wachem, , Numer. Heat Trans. B-Fund. 65, 218 (2014)), whereas spatially underresolved (spurious) capillary waves impose strict limits on the time-step and, hence, dictate the required computational resources for surface-tension-dominated flows [Denner and van Wachem, J. Comp. Phys. 285, 24 (2015)]. By applying an additional shear stress term at the fluid interface, thereby dissipating the surface energy associated with small wavelengths, we have been able to considerably reduce the adverse impact of parasitic currents and mitigate the time-step limit imposed by capillary waves. However, a careful choice of the applied interface viscosity is crucial, since an excess of additional dissipation compromises the accuracy of the solution. We present the derivation of the additional interfacial shear stress term, explain the underlying physical mechanism and discuss the impact on parasitic currents and interface instabilities based on a variety of numerical experiments.

1We acknowledge financial support from the Engineering and Physical Sciences Research Council (EPSRC) through Grant No. EP/M021556/1 and from PETROBRAS.

11:27AM H7.00005 A Multiphase Open Boundary Treatment for Interface Capturing Methods, MANUEL GALE, MARCUS HERRMANN, Arizona State University — Open or outflow boundaries are present in a number of fluid dynamics problems, both internal and external. The main characteristic of this type of boundary is that flow variables are not known and must be computed such that the resulting flow field yields are both stable and accurate. With a focus on incompressible flow, there have been numerous boundary treatments with relevance to finite-element and finite-volume methods. However, while single phase outflow conditions have been widely studied, the work in multiphase outflow boundaries is limited. In this work, the single phase boundary treatment approach of Dong et al. (2014) is extended to multiphase flows using a fractional-step method in combination with either a level-set or volume-of-fluid interface capturing method. The kinetic energy influx through the outflow boundary is locally balanced to improve stability under heavy reverse flow conditions that can arise from having two fluids with high density ratio. Presented is a detailed mathematical description of the outflow boundary condition and representative numerical tests for both single and multiphase flows.

1Support from grant NSF-CBET1054272 is gratefully acknowledged.
11:40AM H7.00006 Interface Surface Area Tracking for the Conservative Level Set Method

1. STEPHANIE FIREHAMMER, OLIVIER DESJARDINS, Sibley School of Mechanical and Aerospace Engineering, Cornell University — One key question in liquid-gas flows is how to model the interface between phases in a way that is mass, momentum, and energy conserving. The accurate conservative level set (ACLS) method of Desjardins et al. [O. Desjardins, V. Moureau, H. Petsch, An accurate conservative level set/ghost fluid method for simulating turbulent atomization, J. Comput. Phys. 227 (18) (2008) 8395-8416] provides a tool for tracking a liquid-gas interface with minimal mass conservation issues; however, it does not explicitly compute the interface surface area and thus nothing can be said a priori about the balance between kinetic energy and surface energy. This work examines an equation for the transport of interface surface area density, which can be written in terms of the gradient of the volume fraction. Furthermore this presentation will outline a numerical method for jointly transporting a conservative level set and surface area density. Finally, we will explore opportunities for energy conservation via the accurate exchange of energy between the flow field and the interface through surface tension, with test cases to show the results of our extended ACLS method.

1Funding from the National Science Foundation is gratefully acknowledged

11:53AM H7.00007 Consistent and conservative framework for incompressible multiphase flow simulations

2. MARK OWKES, Montana State University, OLIVIER DESJARDINS, Cornell University — We present a computational methodology for convection that handles discontinuities with second order accuracy and maintains conservation to machine precision. We use this method in the context of an incompressible gas-liquid flow to transport the phase interface, momentum, and scalars. Using the same methodology for all the variables ensures discretely consistent transport, which is necessary for robust and accurate simulations of turbulent atomizing flows with high-density ratios. The method achieves conservative transport by computing consistent fluxes on a refined mesh, which ensures all conserved quantities are fluxed with the same discretization. Additionally, the method seamlessly couples semi-Lagrangian fluxes used near the interface with finite difference fluxes used away from the interface. The semi-Lagrangian fluxes are three-dimensional, un-split, and conservatively handle discontinuities. Careful construction of the fluxes ensures they are divergence-free and no gaps or overlaps form between neighbors. We have tested and used the scheme for many cases and demonstrate a simulation of an atomizing liquid jet.

12:06PM H7.00008 Thermodynamically Consistent Physical Formulation and an Efficient Numerical Algorithm for Incompressible N-Phase Flows

3. SUCHUAN DONG, Purdue University — This talk focuses on simulating the motion of a mixture of N (N ≥ 2) immiscible incompressible fluids with given densities, dynamic viscosities and pairwise surface tensions. We present an N-phase formulation within the phase field framework that is thermodynamically consistent, in the sense that the formulation satisfies the conservation of mass/momentum, the second law of thermodynamics and Galilean invariance. We also present an efficient algorithm for numerically simulating the N-phase system. The algorithm has overcome the issues caused by the variable coefficient matrices associated with the variable mixture density, viscosity and surface tension among the (N-1) phase field variables and the flow variables. We compare simulation results with the Langmuir-de Gennes theory to demonstrate that the presented method produces physically accurate results for multiple fluid phases. Numerical experiments will be presented for several problems involving multiple fluid phases, large density contrasts and large viscosity contrasts to demonstrate the capabilities of the method for studying the interactions among multiple types of fluid interfaces.

1Support from NSF and ONR is gratefully acknowledged

12:19PM H7.00009 A minimally diffusive interface function steepening approach for compressible multiphase flows

4. JONATHAN REGELE, Iowa State University — Interface capturing methods for contacts and shocks are commonly used in compressible multiphase flows. Artificial diffusion is inherently necessary to maintain a sharp interface function. In this work, a modification to the sharpening approach used in Shukla, Pantano, and Freund [J. Comp. Phys, 229, 2010] is developed that minimizes the artificial diffusion across the interface while maintaining a monotonic interface function. The method requires fewer iterations for convergence and provides a steeper interface function. Examples in one and two dimensions demonstrate the method’s performance.

12:32PM H7.00010 Numerical simulation of compressible multiphase flows using the Parallel Adaptive Wavelet-Collocation Method

5. MOHAMAD ASLANI, JONATHAN REGELE, Iowa State University — Numerical simulation of compressible multiphase flows is a rapidly growing area of research. In this work, we present a parallel adaptive wavelet-collocation method for simulating compressible multiphase flows that utilizes a unified framework to model both the interface capturing and the compressible multiphase flow. The method is based on a combination of the wavelet-collocation method and the parallel adaptive wavelet-collocation method. The method is designed to handle complex flows with sharp interfaces, such as shock waves, and is capable of accurately simulating a wide range of multiphase flows.

Monday, November 23, 2015 10:35AM - 12:45PM
Session H8 Microscale Flows: Flow in Microchannels 108 - Panagiota Angeli, University College London

10:35AM H8.00001 Experimental investigation of liquid-liquid plug formation in a T-junction microchannel

6. PANAGIOTA ANGELI, MAXIME CHINAUD, EYNAGELIA-PANAGIOTA ROUMPEA, WEHELYE WEHELYE, Department of Chemical Engineering, University College London, Torrington Place, London, WC1E 7JE, OMAR. K. MATAR COLLABORATION, LYSE KYAHOUDJI COLLABORATION — Plug formation mechanism of two immiscible liquids was studied experimentally in a 200-µm microchannel using two innovative micro Particle Image Velocimetry (µPIV) techniques i.e. two-colour µPIV and high speed bright field µPIV. The aqueous phase was a water/glycerol solution whereas the organic phase was silicon oil with a range of viscosities from 5 to 155 cSt. Experiments were conducted for different fluid flow rate combinations in the T-junction inlet and it was observed that velocity profiles within the forming plugs depend on the flow rate ratios. The velocity field studies provided insight into the plug mechanism revealing that the interface curvature at the rear of the forming plug changes sign at the later stages of plug formation and accelerates the thinning of the meniscus leading to plug breakage. Results from the two-colour PIV show that the continuous phase resists the flow of the dispersed phase into the main channel at the rear of the plug meniscus and causes the change in the interface curvature.

1Department of Chemical Engineering South Kensington Campus Imperial College London SW7 2AZ
2Department of Chemical Engineering South Kensington Campus Imperial College London SW7 2AZ
10:48AM H8.00002 Experimental and numerical investigations of ionic liquid-aqueous flow in microchannel . QI LI, DIMITRIOS TSAOULIDIS, PANAGIOTA ANGELI, Department of Chemical Engineering, University College London, Torrington Place, London, WC1E 7JE — The hydrodynamic characteristics of plug flow of an ionic liquid-aqueous two-phase system in a microchannel were studied experimentally and numerically. A mixture of 0.2M N-octyl(phenyl)-N,N-diobutylcarbamoylmethylphosphine oxide (CMOP)-1.2 M Tri-n-butylphosphate (TBP) in room temperature ionic liquid 1-butyl-3-methylimidazolium bis[trifluoromethyl]sulfonamide ([C6mim][NTf2]), and a nitric acid solution of 1M were chosen. These fluids are relevant Eu(III) separation by extraction from nitric acid solutions. The two liquid phases were introduced into microchannels of 0.2 and 0.5mm internal diameter through a T-junction inlet. The flow pattern was visualized during plug formation at the inlet section and further downstream by means of bright field planar micro-Particle Image Velocimetry. Key features of plug flow, such as plug velocity, film thickness, plug length and recirculation intensity were measured under various experimental conditions. To gain further understanding of the 3-D flow field, Computational Fluid Dynamics (CFD) simulations approach were also conducted.

11:01AM H8.00003 Interaction of Particles with Recirculating Flow Regions inside Cavities of Inertial Microchannels , HAMED HADDADI, DINO DI CARLO, University of California Los Angeles — Confined inertial flow over cavities of a microfluidic device leads to formation of recirculating flow regions, i.e flow cells, inside cavities which can entrap particles from the free stream. Besides its significance as a fundamental problem in fluid mechanics of mixtures, understanding particle interaction with recirculating flow regions inside cavities is important in biomedical applications, such as Circulating Tumor Cell (CTC) separation and platelet deposition in arterial stenosis. In the present work, a lattice-Boltzmann model with resolved particle-corner interaction combined with microfluidic experiments enabled improved understanding of the particle exchange within flow cells in confined inertial flow. Formation of a limit cycle trajectory, observed in experiments and numerical simulations, is a key feature in particle accumulation. By varying the dimensions of the cavity and channel Reynolds number, the length and location of the limit cycle trajectory also varies, altering of the rate of particle exchange and level of accumulation with recirculating zones inside cavities.

11:14AM H8.00004 Experiment and computational simulations of liquid-liquid flow displacement in microchannels1 , YU LU, MARK SIMMONS, University of Birmingham — Microfluidics has great potential for tight process control in the generation of high value-added products and there is a requirement to understand how one fluid displaces another for either cleaning or control of the interfacial phenomena. Micro-Particle Image Velocimetry (μ-PIV) and shadowgraphy have been used to examine the injection of a fluid into a cavity or semi-circular microchannel (with diameters of 200 μm and 205 μm respectively) which is pre-filled with another fluid. Both immiscible and miscible Newtonian fluid pairs with varying viscosity ratio have been used. Flow instabilities and regimes have been observed which can be characterised using dimensionless flow maps. Displacement efficiency, residual liquid film thickness on the wall, velocity fields and the effect of wall conditions such as wall wettability are also studied. The flow phenomena observed have been modelled using the finite volume ANSYS Fluent CFD package and compared with the experiments.

1EPSC Programme Grant, MEMPHIS, EP/K0039761/1

11:27AM H8.00005 Numerical simulation of liquid-liquid plug formation in a T-shaped cylindrical micro-channel1 , MAXIME CHINAUD, EVANGELIA ROUMPEA, PANAGIOTA ANGELI, University College London, JALEL CHERGUI, DAMIR JURIC, LIMSI-CNRS, SEUNGWON SHIN, Hongik University, Republic of Korea, LYES KAOUDJ, ALI MAHATA, Imperial College London — We present experimental studies and three-dimensional simulations using the code BLUE for different fluid flow rate combinations in a tubular T-junction. All branches have internal diameters equal to 200 μm. The dispersed phase consists of a water/glycerol solution injected from the side branch of the junction, while the continuous phase is silicon oil injected along the main channel axis. BLUE is a new massively parallel Navier-Stokes solver for multiphase flows. Communication across process threads is handled by MPI message passing procedures. The method for the treatment of the fluid phase interfaces and, in particular, capillary forces uses a hybrid Front Tracking/Level Set technique which defines the interface both by a discontinuous density field as well as by a local triangular Lagrangian mesh. This structure allows the interface to undergo large deformations including rupture or coalescence of fluid interfaces.

1EPSC Programme Grant, MEMPHIS, EP/K0039761/1

11:40AM H8.00006 Numerical study of mixing viscous fluids in T-shaped microchannels with compressibility effects1 , JUNFENG YANG, OMAR MATAR, Imperial College London, CHRISTOPHER HARRISON, MATTHEW SULLIVAN, Schlumberger-Doll Research, Cambridge, MA — We study numerically the mixing processes of twomiscible fluids in T-shaped micro-channels in the presence of compressibility effects. Three mixing modes are considered: passive mixing, which relies on the molecular diffusion and chaotic advection; active mixing relies on external disturbances, e.g. due to periodic compression; and a combination of these modes. In all cases considered, one of the fluids, fluid A, is initially present in the dead-end region of the micro-channel. In the ‘passive mixing case, the other fluid, fluid B, flows through the open part of the channel at a constant flow rate. In the active case, this fluid is initially at rest but is then set in motion through pressure cycling. The combined case, involves the flow of fluid B in the presence of compression-decompression cycles. Numerical simulations are carried out for three different fluids, accounting for their compressibility, and their pressure-dependent e.g. density, viscosity, and diffusivity; a simple mixing rule is used to model the properties of the mixed fluids. Our results indicate that the vortices in the dead-end zone, engendered by the relative motion of the fluids leads to their mixing; the combination of mixing modes is shown to promote mixing efficiency significantly.

1Schlumberger-Doll Research

11:53AM H8.00007 Physical mechanisms of flow resistance in textured microchannels , SIMON GAME, DEMETRIOS PAPAGEORGIU, ERIC KEAVENY, Imperial College London, MARC NODES, Tufts University — Transport in microchannels can be enhanced by replacing flat, no-slip boundaries with boundaries etched with longitudinal grooves containing an inert gas, resulting in an effective slip flow. Various physical considerations which are often omitted from mathematical models play a significant role in the behaviour of this flow. Such considerations include: gas viscosity, meniscus curvature, finite channel cross-sections, molecular slip on the gas/liquid or gas/solid interfaces. Using a computationally efficient, multi-element, Chebyshev collocation method, we are able to quantify and combine each of these physical effects. We have shown that for physically realistic parameter values, including each of these effects significantly alters the volumetric flow rate, and hence these effects should not be ignored. Using this framework, we hope to manipulate these effects in order to minimise the flow resistance of the channel.
12:06PM H8.00008 Fluid flow and heat transfer in polygonal micro heat pipes, Sai Rao, Harris Wong, Louisiana State University — Micro heat pipes have been used to cool microelectronic devices, but their heat transfer coefficients are low compared with those of conventional heat pipes. We model heat and mass transfer in triangular, square, hexagonal, and rectangular micro heat pipes under small imposed temperature differences. A micro heat pipe is a closed microchannel filled with a wetting liquid and a long vapor bubble. When a temperature difference is applied across a micro heat pipe, the equilibrium vapor pressure at the hot end is higher than that at the cold end, and the difference drives a vapor flow. As the vapor moves, the vapor pressure at the hot end drops below the saturation pressure. This pressure drop induces continuous evaporation from the interface. Two dimensionless numbers emerge from the momentum and energy equations: the heat-pipe number $H$, and the evaporation exponent $S$. When $H > 1$ and $S > 1$, vapor-flow heat transfer dominates and a thermal boundary layer appears at the hot end, the thickness of which scales as $L/S$, where $L$ is the half-length of the pipe. A similar boundary layer exists at the cold end. Outside the boundary layers, the temperature is uniform. We also find a dimensionless optimal pipe length $S_{\text{opt}}=S_{\text{m}}H$ for maximum evaporative heat transfer. Thus, our model suggests that micro heat pipes should be designed with $H > 1$ and $S = S_{\text{m}}$. We calculate $H$ and $S$ for four published micro-heat-pipe experiments, and find encouraging support for our design criterion.

12:19PM H8.00009 Conjugate thermal creep flow with hydrodynamics and thermal slip conditions in a slit microchannel, Ian Monsivais, Facultad de Ingeniería, Universidad Nacional Autónoma de México, México, D. F., 04510, México, José Lizardi, Programa de Energía, Universidad Autónoma de la Ciudad de México, México, D. F., 06720, México, Federico Méndez, Facultad de Ingeniería, Universidad Nacional Autónoma de México, México, D. F., 04510, México — In this work, we study the conjugate heat transfer between a gas flow and the walls of the microchannel, when the laminar motion of the fluid is caused uniquely by the thermal creep effect on the lower wall. Taking into account that this can represent a microchip or a similar device over which occurs a well defined heat dissipation rate; in our case, we have assumed that the bottom face of this lower wall with finite thermal conductivity, is exposed to a uniform heat flux. On the other hand, the upper wall of the microchannel is subject to a well-known prescribed thermal boundary condition. The heat conduction equation for the lower wall and the mass, momentum and energy equations for the phase gas together with the corresponding boundary conditions are written in dimensionless form, assuming that the Reynolds number associated with the characteristic velocity of the thermal creep and the aspect ratio of the microchannel are both very small. The velocity and temperature fields for the gas phase and the temperature profiles for the lower solid wall are predicted as functions of the involved dimensionless parameters and the main results confirm that the phenomenon of conjugate thermal creep exists whenever the temperature of the lower wall varies linearly or nonlinearly.

12:32PM H8.00010 Rarefaction effects in microchannel gas flow driven by rhythmic wall contractions, Krishnashis Chatterjee, Anne Staples, Virginia Tech, Department of Biomedical Engineering and Mechanics, Virginia Tech Collaboration — Current state of the art microfluidic devices employ precise and timely operation of a complex arrangement of micropumps and valves for fluid transport. A much more novel flow transport mechanism is found in entomological respiratory systems, which involve rhythmic wall contractions for driving the fluid flow. The practical viability of using this technique in future microfluidic devices has been studied earlier. The present study investigates the incorporation of rarefaction effects in the above model of microscale gas flow by including slip boundary conditions. The Navier Stokes equations for gas flow in rectangular microchannel are solved analytically with microscale and lubrication theory assumptions. First order slip boundary conditions are incorporated to account for the rarefaction effects. The dependence of fluid velocities and pressure gradient on the slip boundary conditions is studied. Time averaged unidirectional fluid flow rates are plotted for different phase lags between the contractions, with and without slip in order to obtain an optimum range under different conditions.


10:35AM H9.00001 Ultra-high Burst Strength of CVD Graphene Membranes, Luda Wang, Michael Boutillier, Piran Kidambi, Rohit Karnik1, Massachusetts Institute of Technology, MICROFLUIDICS & NANOFLUIDICS RESEARCH LAB TEAM — Porous graphene membranes have significant potential in gas separation, water desalination and nanofiltration. Understanding the mechanical strength of porous graphene is crucial because membrane separations can involve high pressures. We studied the burst strength of CVD graphene membrane placed on porous support at applied pressures up to 100 bar by monitoring the gas flow rate across the membrane as a function of pressure. Increase of gas flow rate with pressure allowed for extraction of the burst fraction of graphene as it failed under increasing pressure. We also studied the effect of sub-nanometer pores on the ability of graphene to withstand pressure. The results showed that porous graphene membranes can withstand pressures comparable to or even higher than the >50 bar pressures encountered in water desalination, with non-porous CVD graphene exhibiting even higher mechanical strength. Our study shows that porous polycrystalline CVD graphene has ultra-high burst strength under applied pressure, suggesting the possibility for its use in high-pressure membrane separations.

1Principal Investigator

10:48AM H9.00002 Development of macroscopic nanoporous graphene membranes for gas separation, Michael Boutillier, Nicolas Hadjiconstantinou, Rohit Karnik, Massachusetts Institute of Technology — Nanoporous graphene membranes have the potential to exceed permeance and selectivity limits of existing gas separation membranes due to their atomic thickness and ability to support sub-nanometer pores for molecular sieving, while offering low resistance to flow. Gas separation by graphene nanopores has been demonstrated experimentally on micron-scale membranes, but scaling-up to larger sizes is challenging due to graphene imperfections and control of the selective nanopore size distribution. Using a model we developed for the inherent permeance of graphene, we designed a macroscopic graphene membrane predicted to be selectively permeable despite material imperfections. Micrometer-scale defects are sealed by interfacial polymerization and nanometer-scale defects are sealed by atomic layer deposition. The underlying support structure is tuned to further reduce the effects of leakage. Finally, ion bombardment followed by oxidative etching is used to create a high density of selective nanopores. SEM and TEM imaging are used to characterize the resulting membrane structure, and its performance is assessed by gas permeance and selectivity measurements. This work provides insight into gas flow through nanoporous graphene membranes and guides their future development.

1Principal Investigator

11:14 AM H9.00004 Application of Solution-blown 20-50 nm Nanofibers in Filtration of Nanoparticles: The Efficient van der Waals Collectors1, SUMIT SINHA-RAY, SUMAN SINHA-RAY, ALEXANDER YARIN, University of Illinois at Chicago, BEHNAM POURDEYHIMI, North Carolina State University — Filtration efficiency of commercially available filter media with fiber/pore sizes on the scale of 10 µm can be dramatically increased by adding a layer of ultrafine supersonically-blown 20-50 nm nanofibers. Different commercial filters were modified with (i) electrospun nanofibers alone, (ii) solution-blown 20-50 nm alone, and (iii) the dual coating with electrospun nanofibers deposited first and the solution-blown 20-50 nm nanofibers deposited on top of them. Detailed observations of nanoparticle removal revealed that the above-mentioned modified filters, especially those with the dual nanofiber coating with the 20-50 nm nanofibers deposited on top, are the most effective in removing the below-200 nm Cu nanoparticles/clusters from aqueous suspensions, in particular at the lowest concentrations of 0.2-0.5 ppm. The theory developed in the present work dealing with convective transport of nanoparticles in the fluid flow along with diffusion of nanoparticles and the van der Waals attraction explains and describes how the smallest solution-blown nanofibers introduce a novel physical mechanism of nanoparticle interception (the attractive van der Waals forces) and become significantly more efficient collectors compared to the larger electrospun nanofibers. The theory also elucidates the morphology of the nanoparticle clusters being accumulated at the smallest nanofiber surfaces, including the clusters growing at the windward side, or in some cases also on the leeward side of a nanofiber.

1This work is supported by the Nonwovens Cooperative Research Center (NCRC), grant No. 12-144SB.

11:27 AM H9.00005 Molecular level water and solute transport in reverse osmosis membranes1, RICHARD M. LUEPTOW, MENG SHEN, SINAN KETEN, Northwestern University — The water permeability and rejection characteristics of six solutes, methanol, ethanol, 2-propanol, urea, Na+, and Cl−, were studied for a polymeric reverse osmosis (RO) membrane using non-equilibrium molecular dynamics simulations. Results indicate that water flux increases with an increasing fraction of percolated free volume in the membrane polymer structure. Solute molecules display Brownian motion and hop from pore to pore as they pass through the membrane. The solute rejection depends on both the size of the solute molecule and the chemical interaction of the solute with water and the membrane. When the open spaces in the polymeric structure are such that solutes have to shed at least one water molecule from their solvation shell to pass through the membrane molecular structure, the water-solute pair interaction energy governs solute rejection. Organic solutes more easily shed water molecules than ions to more readily pass through the membrane. Hydrogen-bonding sites for molecules like urea also lead to a higher rejection. These findings underline the importance of the solute’s solvation shell and solute-water-membrane chemistry in solute transport and rejection in RO membranes.

1Funded by the Institute for Sustainability and Energy at Northwestern with computing resources from XSEDE (NSF grant ACI-1053575).

11:40 AM H9.00006 Water and Molecular Transport across Nanopores in Monolayer Graphene Membranes, DOOJOON JANG, SEAN O’HERN, PIRAN KIDAMBI, MICHAEL BOUTILIER, YI SONG, Massachusetts Institute of Technology, JUAN-CARLOS IDROBO, Oak Ridge National Laboratory, JING KONG, Massachusetts Institute of Technology, TAHAR LAOUI, King Fahd University of Petroleum and Minerals, ROHIT KARNIK, Massachusetts Institute of Technology — Graphene’s atomic thickness and high tensile strength allow it to outstand as backbone material for next-generation high flux separation membrane. Molecular dynamics simulations predicted that a single-layer graphene membrane could exhibit high permeability and selectivity for water over ions/molecules, qualifying as novel water desalination membranes. However, experimental investigation of water and molecular transport across graphene nanopores had remained largely explored due to the presence of intrinsic defects and tears in graphene. We introduce two-step methods to seal leakage across centimeter scale single-layer graphene membranes create sub-nanometer pores using ion irradiation and oxidative etching. Pore creation parameters were varied to explore the effects of created pore structures on water and molecular transport driven by forward osmosis. The results demonstrate the potential of nanoporous graphene as a reliable platform for high flux nanofiltration membranes.

11:53 AM H9.00007 Water Purification across MoS2 Nano-porous Membranes, MOHAMMAD HEIRANIAN, AMIR BARATI FARIMANI, NARAYANA R. ALURU, Univ of Illinois - Urbana — A 2D material, molybdenum disulfide (MoS2), is proposed as a nano-porous membrane for water desalination. By performing detailed molecular dynamics simulations, we find that salt ions are rejected efficiently across a single-layer MoS2 while water permeates at high rates. Depending on the pore area, which ranges from 20 to 60 Å2, the nanopore allows less than 12% of ions to pass through even at theoretically high pressures of 350 MPa. Water permeation across the MoS2 membrane is found to be as high as 12 L/cm2/day/MPa which is at least two orders of magnitude higher than that of other existing nano-porous membranes. Pore chemistry is shown to be one of the important factors leading to large water fluxes. MoS2 pore edges terminated with only molybdenum atoms result in higher fluxes which are about 70% higher than that of graphene nanopores. These findings are explained and supported by the permeation coefficients, energy barriers, water density and velocity distributions in the pores.
Rutgers University filtration systems relying on the active osmotic exchanger principle. We anticipate that our proof of principle will be a starting point for the development of new reverse osmosis systems, but can also work at much smaller pressures (of the order of the blood pressure, 0.13 bar, as compared to more than 30 bars for pressure-retarded osmosis systems). We first establish simple analytical results to derive general operating principles, based on coupled water permeable pores and osmotic pumps. The best filtration geometry, in terms of power inspiration from this osmotic exchanger design to propose new nanofiltration principles. We take the human kidney is capable of recycling about 200 liters of water per day, at the relatively low cost of 0.5 kJ/L (standard dialysis requiring at least 150 kJ/L). Kidneys are constituted of millions of parallel filtration networks called nephrons. The nephrons of all mammalian kidneys present a specific loop geometry, the Loop of Henle, that is believed to play a key role in the urinary concentrating mechanism. One limb of the loop is permeable to water and the other contains sodium pumps that exchange with a common interstitium. In this work, we take numerical and experimentally explore how inertial migration is affected by the presence of a permeate flow through the porous walls of a microchannel in order to develop a platform for further studies of particle transport in a TFF device. Numerically, we use COMSOL multiphysics to model the transport parameter space to forecast the underlying asymmetry with particle-size segregation. To investigate this, we develop a MEMS fabricated TFF device to confirm the results of the numerical model, where the permeate flow is controlled using multiple pumps and pressure transducers regulated by a feedback loop. Experimental and numerical results reveal interesting dynamics, including the competition between permeate and inertial forces and the consequences of this competition on particle trajectories and equilibrium location.


10:35AM H10.00001 Itokawa is not Brazil: granular segregation on asteroids, TROY SHINBROT, Rutgers University, PINKA CHAKRABORTY, TAPAN SABUWALA, Okinawa Institute of Science & Technology — Recent photographs of the asteroid Itokawa have revealed strong separation between regions populated almost entirely by sand and other regions consisting only of larger boulders. This size separation has been attributed to the Brazil Nut Effect (BNE), however we point out here that the BNE depends on conditions such as isotropic gravity, parallel sidewalls and periodic vertical shaking that are wholly absent on asteroids. On the other hand, surface areas of boulders and sand appear to be comparable on Itokawa, and in this situation it follows that the asteroid must have suffered many orders of magnitude more collisions with sand particles than with boulders. We observe that a sand particle will tend to bounce off of a boulder but will sink into a sea of similar sand particles, and so we predict that sand seas must grow on such asteroids. We carry out experiments and simulations to evaluate this and related predictions, and we demonstrate that this new mechanism of segregation based on simple counting of grains can produce the strong separation of sizes reported.

10:48AM H10.00002 Underlying Asymmetry with Particle-Size Segregation, PARMESH GAJJAR, The University of Manchester, KASPER VAN DER VAART, GAEL EPELY-CHAUVIN, NICOLAS ANDREINI, cole Polytechnique Fdrale de Lausanne, NICO GRAY, The University of Manchester, CHRISTOPHE ANCEY, cole Polytechnique Fdrale de Lausanne — Granular media have a natural tendency to self-organise when sheared, with different sized constituents counter-intuitively separating from each other. Not only does the segregation produce a rich diversity of beautiful patterns, but it can also have serious implications in both industrial and geophysical environments. Despite the universal importance, the individual particle dynamics during segregation are still poorly understand, with such an analysis proving to be difficult with conventional techniques such as binning and sidewall observation. This talk will present results of recent experiments that studied particle scale segregation dynamics during oscillatory shear. Reflective index matched scanning allowed examination of the interior of the flow, where it was observed that large and small particles have an underlying asymmetry that is dependant on the local particle concentration. Small particles were seen to segregate faster through regions of many large particles, whilst large particles rise slower through regions of many small particles. The asymmetry is quantified on both bulk and particle length scales, and is shown to have good agreement with a continuum model that uses a cubic segregation flux.

11:01AM H10.00003 Size and density segregation in granular mixtures, DEEPAK TUNUGUNTLA, THOMAS WEINHART, ANTHONY THORNTON, Multi-scale Mechanics group, University of Twente — In recent years, quite a few mixture theory continuum models, e.g., Tunuguntla et al. (2014), have attempted to, qualitatively and quantitatively, predict particle segregation in bidisperse mixture flows over inclined channels. This ongoing continuum approach incorporates percolation-driven segregation phenomenon into a continuum transport equation given in terms of particle volume fraction of a particular species. The key feature behind these models lies upon the fact on how the total bulk pressure is distributed among the two particle species. Thereby, indicating the need for suitable pressure scalings which help us determine the proportion of the bulk pressure to be carried by each type of particle species. To investigate this in detail, fully three dimensional discrete particle simulations (DPMs) are used. Further, we project the discrete data onto a continuum field using the novel coarse graining technique, see Tunuguntla et al. (2015). With these constructed macroscopic fields, such as the partial and bulk stresses, at hand, we arrive at suitable pressure scalings taking into the effects of both particle particle size and density. Hence, providing us with DPMs validated pressure scalings required to predict particle segregation more accurately.

1STW Grant #11039 “Polydisperse flows over inclined channels” and STW-VIDI #13472 “Shaping Segregation: Advanced Modelling of Segregation and its Application to Industrial Processes”
ceramic particles were also performed which reproduced the segregation patterns obtained in both the simulations and the theory.

This study aims to understand this effect in the context of bi-disperse roll waves in shallow granular free-surface flows. Experiments have been performed in a 3 meter long chute using several mixtures of spherical glass beads of diameter 75-150 and 400-600 microns flowing on a rough bed. These show that the waves propagate at constant speed that depends on the initial mixture composition. In addition, during their propagation, a higher concentration of large particles is localized at the front of the waves. A theoretical and numerical approach is presented that relies on arbitrary fitting parameters, our model uses parameters based on kinematics measured using discrete element method (DEM) simulations. The model depends on both the Péclét number, Pe, which we defined as the ratio of the segregation rate to the diffusion rate, and the relative segregation strength between particle species. At large Pe, segregation dominates and chute flow consists of distinct stratified regions of small (bottom), medium (center) and large (top) particles, whereas at small Pe, diffusion dominates, which results in a well mixed flow. As relative segregation strength between any two particle species is increased, the segregation between them becomes quicker. However, as relative segregation strength between them is decreased, they remain mixed with each other. Preliminary results from DEM simulations support our theoretical model.

11:40AM H10.00006 Continuum modeling of segregation for tridisperse granular materials in developing chute flow

Our model uses parameters based on kinematics measured using discrete element method (DEM) simulations. The model depends on both the Péclét number, Pe, which we defined as the ratio of the segregation rate to the diffusion rate, and the relative segregation strength between particle species. At large Pe, segregation dominates and chute flow consists of distinct stratified regions of small (bottom), medium (center) and large (top) particles, whereas at small Pe, diffusion dominates, which results in a well mixed flow. As relative segregation strength between any two particle species is increased, the segregation between them becomes quicker. However, as relative segregation strength between them is decreased, they remain mixed with each other. Preliminary results from DEM simulations support our theoretical model.

11:53AM H10.00007 Self Propelled particle systems: A study of the onset of organized motion

The objective of this study is to investigate the onset of organized motion in Self Propelled particle systems in confined two-dimensional spaces. A dynamic Vicsek model including particle inertia has been proposed. In this approach, the particles are modeled as soft disks with finite mass. The particles are required to align their local motion to their immediate neighborhood, similar to standard Vicsek model. We study the dynamics of organized motion and diffusion properties of the particles as a function of the local co-ordination coefficient (LCC) and thrust generation ability. Firstly, we observe a hysteretic phase transition from disorganized thermal motion to organized vortical motion as LCC is increased. In addition, we observe a sensitive dependence of the realized state of the system to the initial conditions for the particles near to the critical LCC. Finally the energy budget of the system — including potential and kinetic energies as well as dissipation with time — is used to understand the motivation for the phase transition.

12:06PM H10.00008 ABSTRACT WITHDRAWN —

12:19PM H10.00009 Pattern formation in triboelectrically charged binary packings

In this spirit, we study the process of self-assembling for vertically shaken granular materials. Our system consists from 1 to 400 plastic beads of 3mm size made from Teflon and Nylon in 2D and 3D geometries. We find self-organization in four, five and sixfold order which is due to charging of the system via triboelectric effects between the grains. We observe that the binary system solidifies on a time scale of a few minutes. Image processing is used to extract the structural and dynamical properties of the assemblies. The mixture ratio is tuned from 1.5 to 5:1 and the humidity level is varied between 10% and 90% leading to various transitions between the morphologies.

10:35AM H11.00001 Effects of Pr on Optimal Heat Transport in Rayleigh-Bénard Convection

DAVID SONDACK, MARKO BUDIŠIĆ, FABIAN WALEFFE, LESLIE SMITH, University of Wisconsin, Madison — Steady flows that optimize heat transport are obtained for two-dimensional Rayleigh-Bénard convection with no-slip horizontal walls for a variety of Prandtl numbers Pr and Rayleigh number up to $Ra = 10^9$. The presence of two local maxima of $Nu$ with different horizontal wavenumbers at the same $Re$ leads to the emergence of two different flow structures as candidates for optimizing the heat transport where the Nusselt number $Nu$ is a non-dimensional measure of the vertical heat transport. For $Pr \leq 7$, optimal transport is achieved at the smallest maximal wavenumber whereas for $Pr > 7$ at high-enough $Ra$ the optimal structure occurs at the larger maximal wavenumber. Three regions are observed in the optimal mean temperature profiles, $\bar{T}(y)$: (1) $\partial\bar{T}/\partial y < 0$ in the boundary layers, (2) $\partial\bar{T}/\partial y > 0$ ($Pr \leq 7$) or $\partial\bar{T}/\partial y < 0$ ($Pr > 7$) in the central region, and (3) $\partial\bar{T}/\partial y > 0$ between the boundary layers and central region. We also search for a signature of these optimal structures in a fully-developed turbulent flow by employing modal decompositions such as the proper orthogonal decomposition and the Koopman mode decomposition.

$1\text{Partial support from NSF-DMS grant 1147523 is gratefully acknowledged.}$

10:48AM H11.00002 Optimal heat transport

ANDRE SOUZA, CHARLES R. DOERING, University of Michigan — The transport of heat by buoyancy driven flows, i.e., thermal convection plays a central role in many natural phenomena and an understanding of how to control its mechanisms is relevant to many engineering applications. In this talk we will consider a variational formulation of optimal heat transport in simple geometries. Numerical results, limits on heat transport, and a comparison to Rayleigh-Bénard convection will be presented.

1Research supported by NSF Awards PHY-1205219, PHY-1338407, PHY-1443836, PHY-1535555 and DMS-1515161.

11:01AM H11.00003 A theory for optimal heat transfer in a partitioned convection cell

JUN CHEN, State Key Lab. for Turb. & Complex Sys., Dept. Mech. & Engg. Sci., College of Engg., Peking Univ., Beijing 100871, China, YUN BAO, Dept. Appl. Mech. & Engg. Sci., School of Engg., Sun Yat-Sen Univ., Guangzhou 510275, China, ZHEN-SU SHE, State Key Lab. for Turb. & Complex Sys., Dept. Mech. & Engg. Sci., College of Engg., Peking Univ., Beijing 100871, China — We report a theory explaining recent observation of significant enhancement of heat transfer in a partitioned Rayleigh-Bénard convection (RBC), where vertical adiabatic boundaries and the turbulent interior of the flow. Here, by tailoring the geometry of the upper boundary we manipulate this boundary layer interaction, and study the turbulent transport of heat in two-dimensional Rayleigh-Bénard convection with numerical simulations in fluid dynamics. The detailed description by which hot fluid rises and cold fluid descends focuses on the nature of the interaction between the boundary layers and turbulent interior of the flow. Here, by tailoring the geometry of the upper boundary we manipulate this boundary layer – interior flow interaction, and study the turbulent transport of heat in two-dimensional Rayleigh-Bénard convection with numerical simulations using the Lattice Boltzmann method. By fixing the roughness amplitude of the upper boundary and varying the wavelength $\lambda$, we find that the exponent $\beta = 0.038 - 0.083$ for $\Gamma = 1$, and an optimal spacing of the horizontal channel $b/H = 0.011$ for $\Gamma = 5$. The former (channel) yields a heat flux linearly proportional to $b$ for small $b$, whereas the latter (boundary layer) follows $-2/3$-law for large $b$. We suggest that the partitioned RBC provides a vehicle for heat enhancement with a wide range of industrial applications.

1This work was supported by National Nature Science Fund of China under Grant No. 11372362.

11:14AM H11.00004 Tailoring boundary geometry to optimize heat transport in turbulent convection

SRIKANTH TOPPALADODDI, Yale University, University of Oxford, SAURO SUCCI, Istituto per le Applicazioni del Calcolo “Mauro Picone” (C.N.R.), Rome, JOHN WETTTLAUFER, Yale University, University of Oxford, Nordita — Turbulent Rayleigh-Bénard convection between planar horizontal boundaries is a classical example of the challenge posed by multiple interacting scales in fluid dynamics. The detailed description by which hot fluid rises and cold fluid descends focuses on the nature of the interaction between the boundary layers and the turbulent interior of the flow. Here, by tailoring the geometry of the upper boundary we manipulate this boundary layer – interior flow interaction, and study the turbulent transport of heat in two-dimensional Rayleigh-Bénard convection with numerical simulations using the Lattice Boltzmann method. By fixing the roughness amplitude of the upper boundary and varying the wavelength $\lambda$, we find that the exponent $\beta = 0.038 - 0.083$ for $\Gamma = 1$, and an optimal spacing of the horizontal channel $b/H = 0.011$ for $\Gamma = 5$. The former (channel) yields a heat flux linearly proportional to $b$ for small $b$, whereas the latter (boundary layer) follows $-2/3$-law for large $b$. We suggest that the partitioned RBC provides a vehicle for heat enhancement with a wide range of industrial applications.

11:27AM H11.00005 Effects of Prandtl number on flow over a vertical heated cylinder

ABDULVAHAB SAMEEN, AJITHKUMAR S, IIT Madras, chennai, India, ANILLAL S, College of engg, Trivandrum, India — Flow over a two dimensional heated cylinder is analyzed numerically using a hybrid finite element-finite volume method. We assume the flow direction to be opposite to the direction of gravity. It is fundamental in fluid dynamics that the von Karman vortex street appears in the wake of the cylinder above the Reynolds number of approximately 47. On heating the cylinder surface, the Strouhal number (St), which is the non-dimensional representation of the vortex shedding frequency, increases. The gradual increase in St is followed by a sudden drop at a particular $\lambda < \lambda_{\text{max}}$ wavelength limits. The changes in the exponent originate in the nature of the coupling between the boundary layer and the turbulent interior. We present a simple scaling argument embodying this coupling, which describes the maximal convective heat flux.

11:40AM H11.00006 Heat-flux enhancement by vapour-bubble nucleation in Rayleigh-Bénard turbulence

CHAO SUN, Tsinghua University, China. University of Twente, Netherlands, DANIELA NAREJO-GUZMAN, University of Twente, Netherlands, UCBS, Santa Barbara, USA, YANBO XIE, YONGYUE CHEN, DAVID FERNANDEZ-REVAS, DETLEF LOHSE, University of Twente, Netherlands, GUENTER AHLERS, UCBS, Santa Barbara, USA — We report on turbulent convective heat transport enhancement and local temperature modifications in the bulk due to vapour-bubble nucleation at the bottom plate of a Rayleigh-Bénard cylindrical sample (aspect ratio 1.0, diameter of 8.8 cm) filled with liquid. Etched microcavities acted as nucleation sites. Only the central area of the bottom plate (diameter of 2.5 cm) with an array of microcavities was heated. The Nusselt-number $Nu$ was investigated as a function of the bottom plate superheat $T_b$, by varying the temperature of the bottom plate $T_b$ and keeping a fixed difference between $T_b$ and the top plate temperature $T_c - T_b = 16 \text{ K}$. Nusselt-number of both 1- and 2-phase flow for the same $T_b$ value was obtained; 2-phase-$Nu$ was increasingly enhanced relative to the 1-phase $Nu$ for increasing $T_b$. Varying the cavity density between 69 and 0.3 per mm$^2$ had only a small effect on the global $Nu$ enhancement; however $Nu$ per active site decreased as the cavity density increased. $Nu$ of an isolated nucleating site was found to be limited by the rate at which it could host a phase change.

1Work supported by an ERC-Advanced Grant and by NSF grant DMR11-58514.
11:53AM H11.00007 Azimuthal Decomposition of Wide Aspect-Ratio, Turbulent Rayleigh-Benard Convection in a Cylindrical Cell. PHILIP SAKIEVICH, YULIA PEET, RONALD ADRIAN, Arizona State University — Turbulent Rayleigh-Benard convection (RBC) is considered an ideal problem for studying the thermal convection that occurs in nature, and it is typically studied in finite cylindrical or rectangular domains. Cylindrical domains have an advantage because they prevent geometric effects from defining preferential horizontal directions in the flow. This allows the large scale patterns to drift azimuthally and mimic the dynamics of convection in applications where geometric constraints are minimal. The large scale pattern for RBC in small aspect-ratio (Γ) domains is a single roll-cell that spans the entire domain, and the azimuthal drift for this pattern can be fairly energetic. As Γ is increased the single-roll cell breaks into a multi-roll cell pattern, and the time scale for azimuthal motion increases substantially. In this presentation we investigate azimuthal properties of the velocity and temperature fields in a 6.3 Γ cell with a Rayleigh number of 1 × 10^5 and a Prandtl number of 6.7. Statistical independence in the azimuthal direction is investigated for each field, and a detailed decomposition of the multi-roll cell pattern is presented. These analyses are performed through temporal and spatial averaging techniques and Fourier decomposition.

12:06PM H11.00008 Variation of effective roll number on MHD Rayleigh-Benard convection confined in a small-aspect ratio box. YUIJI TASAKA, Hokkaido University, TAKATOSHI YANAGISAWA, JAMSTEC, TOBIAS VOGT, SVEN ECKERT, HZDR — MHD Rayleigh-Benard convection was studied experimentally using a box filled with liquid metal with five in aspect ratio and square horizontal cross section. Applying horizontal magnetic field organizes the convection motion into quasi-two dimensional rolls arranged parallel to the magnetic field. The number of rolls has tendency, decreases with increasing Rayleigh number Ra and increases with increasing Chandrasekhar number Q. To fit the box with relatively smaller aspect ratio, the convection rolls take regime transition accompanying variation of the roll number against variations of Ra and Q. We explored convection regimes in a ranges, 2 × 10^4 < Q < 10^4 and 5 × 10^3 < Ra < 3 × 10^5 using ultrasonic velocity profiling that can capture time variations of instantaneous velocity profile. In a range Ra/Q ∼ 10, we found periodic flow reversals in which five rolls periodically change the direction of their circulation with gradual skew of rolls. We performed POD analysis on the spatio-temporal velocity distribution obtained by UVP and indicated that that the periodic flow reversals consist of periodic emergence of 4-rolls mode in dominant 5-rolls mode. POD analysis also provided evaluation of effective number of rolls as a more objective approach.

12:19PM H11.00009 Roughness-triggered turbulent boundary layers in Rayleigh-Benard convection. JULIEN SALORT, CNRS / ENS Lyon, OLIVIER LIOT, ENS Lyon, ROBERT KAISER, RONALD DU PUITS, TU Ilmenau, FRANCESCA CHILL, ENS Lyon — We present an analysis of the velocity fields in a Rayleigh-Benard cell with a rough bottom plate. Beyond a critical Rayleigh number, the cell undergoes a transition towards a regime of enhanced heat transfer. The threshold is reached when the boundary layer thickness is smaller than the roughness size. We have obtained velocity fields using PIV near the obstacles, as well as the local heat-flux on the bottom plate. This has allowed us to test and improve our previous interpretation of the roughness-induced heat transfer enhancement mechanisms as a roughness-triggered transition to turbulent boundary layers, see Salort, et al., Phys. Fluids 26, 015112 (2014). The velocity profiles on the top of the obstacle are indeed quite different above and below the transition. Below the transition, the profile is fairly compatible with profiles obtained in the smooth case. Above the transition, for z^+ > 30, the velocity profile is closer to the logarithmic profile that one would expect in the case of a turbulent boundary layer, and the slope is close to the classical value of 2.40. The offset however is slightly lower than the classical 5.84, as can be expected on a rough surface.

1Access to Barrel of Ilmenau funded by EuHIT project (European Grant Agreement No 312778)

12:32PM H11.00010 Inertial effects on heat transfer in superhydrophobic microchannels. ADAM COWLEY, DANIEL MAYNES, JULIE CROCKETT, BRIAN IVERSON, Brigham Young University, BYU FLUIDS TEAM — This work numerically studies the effects of inertia on thermal transport in superhydrophobic microchannels. An infinite parallel plate channel comprised of structured superhydrophobic walls is considered. The structure of the superhydrophobic surfaces consists of square pillars organized in a square array aligned with the flow direction. Laminar, fully developed flow is explored. The flow is assumed to be non-wetting and have an idealized flat meniscus. A shear-free, adiabatic boundary condition is used at the liquid/gas interface, while a no-slip, constant heat flux condition is used at the liquid/solid interface. A wide range of Peclet numbers, relative channel spacing distances, and relative pillar sizes are considered. Results are presented in terms of Poiseuille number, Nusselt number, hydrodynamic slip length, and temperature jump length. Interestingly, the thermal transport is varied only slightly by inertial effects for a wide range of parameters explored and compares well with other analytical and numerical work that assumed Stokes flow. It is only for very small relative channel spacing and large Peclet number that inertial effects exert significant influence. Overall, the heat transfer is reduced for the superhydrophobic channels in comparison to classic smooth walled channels.

1This research was supported by the National Science Foundation (NSF) United States (Grant No. CBET-1235881).

Monday, November 23, 2015 10:35AM - 12:32PM — Session H12 Wind Turbines: Vertical Axis

10:35AM H12.00001 Optimization of wind farm performance using low-order models. JOHN DABIRI, IAN BROWNSTEIN, Stanford University — A low order model that captures the dominant flow behaviors in a vertical-axis wind turbine (VAWT) array is used to maximize the power output of wind farms utilizing VAWTs. The leaky Rankine body model (LRB) was shown by Araya et al. (JRESE 2014) to predict the ranking of individual turbine performances in an array to within measurement uncertainty as compared to field data collected from full-scale VAWTs. Further, this model is able to predict array performance with significantly less computational expense than higher fidelity numerical simulations of the flow, making it ideal for use in optimization of wind farm performance. This presentation will explore the ability of the LRB model to rank the relative power output of different wind turbine array configurations as well as the ranking of individual array performance over a variety of wind directions, using various complex configurations tested in the field and simpler configurations tested in a wind tunnel. Results will be presented in which the model is used to determine array fitness in an evolutionary algorithm seeking to find optimal array configurations given a number of turbines, area of available land, and site wind direction profile. Comparison with field measurements will be presented.
10:48AM H12.00002 Transition to bluff body dynamics in the wake of vertical axis turbines. DANIEL ARAYA, California Institute of Technology, JOHN DABIRI, Stanford University — A unifying characteristic among bluff bodies is a similar wake structure independent of the shape of the body. We present experimental data to demonstrate that the wake of a vertical axis wind/water turbine (VAWT) shares similar features to that of a bluff body, namely a circular cylinder. For a fixed Reynolds number (Re $\approx 10^4$) and variable tip-speed ratio, 2D particle image velocimetry (PIV) is used to measure the velocity field in the wake of three different laboratory-scale turbines: a 2-bladed, 3-bladed, and 5-bladed VAWT, each with similar geometry. From the PIV measurements, the time-averaged and dynamic characteristics of the wake are evaluated. In all cases, we observe three distinct regions in the VAWT wake: (1) the near wake, where periodic blade shedding dominates; (2) a transition region, where blade vortices decay and growth of a shear layer instability occurs; (3) the far wake, where bluff body wake oscillations dominate. We further characterize this wake transition with regard to turbine solidity and examine its relation to the mean flow, an important metric for power production within a wind farm.

10:01AM H12.00003 Patterns of 3D flow in a rotating cylinder array. ANNA CRAIG, JOHN DABIRI, JEFFREY KOSEFF, Stanford University — Experimental data are presented for large arrays of rotating, finite-height cylinders, which show that the three-dimensional flows are strongly dependent on the geometric and rotational configurations of the array. Two geometric configurations of the cylinders, each with two rotational configurations, were examined for a total of four arrays. 2D PIV was conducted in multiple intersecting horizontal and vertical sheets at a location far downstream of the leading edge of the array in order to build up a picture of the 3D developed flow patterns. It was found that the rotation of the cylinders drives the formation of streamwise and transverse flow patterns between cylinders. These horizontal flow patterns, by conservation of mass, drive vertical flows through the top of the array. As the array of rotating cylinders may provide insight into the flow kinematics of an array of vertical axis wind turbines, this planform flux is of particular interest as it would bring down into the array high kinetic energy fluid from above the array, thus increasing the energy resource available to turbines far downstream of the leading edge of the array.

11:14AM H12.00004 In Situ Particle Tracking around kW Sized Wind Turbines. IAN BROWNSTEIN, JOHN DABIRI, Stanford — Laboratory studies of model wind turbines are typically unable to match both the Reynolds number (Re) and tip speed ratio (TSR) of full-scale wind turbines. In order to match both relevant parameters, a quantitative flow visualization method was developed to take in situ measurements of the flow around full-scale wind turbines. The apparatus constructed was able to seed an approximately 9m x 9m x 5m volume in the wake of the turbine using artificial snow. Quantitative results were obtained by tracking the evolution of the snow using particle tracking algorithms. As a next step toward full 3D-PTV measurements, results will be presented in which a 2D measurement is taken with both a single camera positioned at the base of the turbine looking upward. The resulting tracking is therefore integrated in the span-wise direction. This method is demonstrated through a comparative study of a five-bladed VAWT producing power in different wind conditions at the Field Laboratory for Optimized Wind Energy (FLOWE) in Lancaster, CA. Future work to expand this method to 3D-PTV is also discussed.

11:27AM H12.00005 Computational Study of Savonius Wind Turbines with Stators. AARON ALEXANDER, ARVIND SANTHANAKRISHNAN, Oklahoma State University — The dynamics of a stator assembly that directs incoming wind into an internal cylindrical trapped flow that exits vertically has been previously studied using particle image velocimetry and computational fluid dynamics (CFD). The present study uses the commercial CFD package Star-CCM+ (CD-adapco) to investigate how a Savonius rotor is affected by the introduction of cylindrical flow trapped by a stator enclosure. The results are then compared with the flow field around an identical Savonius rotor without a stator assembly. The flow characteristics are investigated at Reynolds numbers on the order of one million to examine local flow effects around the rotor as well as downstream wake vorticity. Additionally, the minimum free stream wind velocity needed for rotor start-up and rotor output power will be compared with and without the use of a stator.

11:40AM H12.00006 Morphing blade design for vertical axis wind turbines. DAVID MACPHEE, University of Alabama, Tuscaloosa, ASFAW BEYENE, San Diego State University — Wind turbines operate at peak efficiency at a certain set of operational conditions. Away from these conditions, conversion efficiency drops significantly, requiring pitch and yaw control schemes to mitigate these losses. These efforts are an example of geometric variability, allowing for increased power production but with an unfortunate increase in investment cost to the energy conversion system. In Vertical-Axis Wind Turbines (VAWTs), the concept of pitch control is especially complicated due to a dependence of attack angle on armature azimuth. As a result, VAWT pitch control schemes, both active and passive, are as of yet unfeasible. This study investigates a low-cost, passive pitch control system, in which VAWT blades are constructed of flexible material, allowing for continuous shape-morphing in response to local aerodynamic loading. This design is analyzed computationally using a finite-volume fluid-structure interaction routine and compared to a geometrically identical rigid rotor. The results indicate that the flexible blade increases conversion efficiency by reducing the severity of vortex shedding, allowing for greater average torque over a complete revolution.

11:53AM H12.00007 The effects of Reynolds number, tip speed ratio, and solidity in VAWTs. COLIN PARKER, ALLEN SCHULT, MEGAN C. LEFTWICH, George Washington University — The wakes of several scale models of vertical axis wind turbines (VAWTs) are investigated in a wind tunnel using particle imaging velocimetry (PIV). The tip speed ratio, Reynolds number, and solidity (chord to diameter ratio) is varied to see effect each parameter. The solidity is changed by varying the chord length of a three blade turbine of constant diameter. The range of parameters (Reynolds number and tip-speed ratio) investigated, closely matches those of full size turbines. Time averaging behind the turbines shows the asymmetry in the wake. A more complete picture of the wake is seen using phase averaging by syncing the imaging to the position of the turbine. These results show a cycle of structures developing on the blades and then being shed into the wake. Imaging is done at the midplane of the turbine from upstream of the turbine back into the wake. Additionally a vertical plane behind the center of the turbine is used to measure the horizontal components in the wake.

12:06PM H12.00008 Wake Development of a Model Vertical Axis Wind Turbine. HAWWA KADUM, SASHA FRIEDMAN, ELIZABETH CAMP, RAUL CAL, Portland State University — At the Portland State University wind tunnel facility, an experiment is conducted to observe the downstream development of the wake past a model vertical axis wind turbine (VAWT). The flow domain is composed of streamwise-spanwise planes at mid-height of the VAWT rotor and data is obtained via particle image velocimetry (PIV). The flow field is assessed by analyzing contours of mean velocities and the full Reynolds stress tensor. Furthermore, profiles of the aforementioned quantities and flow parameters are discussed in the context of downstream evolution/flow development.
12:19PM H12.00009 Experimental Performance of a Novel Trochoidal Propeller. BERNARD ROESLER, BRENDEN EPPS, Thayer School of Engineering, Dartmouth College — In the quest for energy efficiency in marine transportation, a promising marine propulsor concept is the trochoidal propeller. We have designed and tested a novel trochoidal propeller using a sinusoidal blade pitch function. The main results presented are measurements of thrust and torque, as well as the calculated efficiency, for a range of advance coefficients. The experimental data show narrow 95% confidence bounds, demonstrating high accuracy and repeatability in the experimental method. We compare our results with those of other propellers, as well as a representative screw propeller. While the efficiency of our propeller exceeds that of the cycloidal-pitch trochoidal propeller, it is slightly lower than the efficiencies of the other propellers considered. We also present a theoretical model that can be used to further explore and optimize such trochoidal propellers, leading to new avenues for improvements in marine propulsion systems.


10:48AM H13.00002 Diving seabirds: the stability of a diving elastic beam. BRIAN CHANG, MATTHEW CROSON, SUNGHWAN JUNG, Virginia Tech — In this study, we examine the buckling stability of a beam attached to a cone plunging diving into a bath of water, which is inspired by diving birds. This beam-cone system initially experiences an impact force before the cone is completely submerged, followed by a hydrodynamic drag force. Using high speed imaging techniques, it was observed that the soft elastic beam exhibits either buckling (unstable) or non-buckling (stable) behaviors upon impact and submergence. Large cone angles, long throw models, and high impact velocities likely cause buckling in the beam-cone system. By varying geometric factors of the beam-cone system, changing the impact velocity, a transition from non-buckling to buckling is characterized through physical experiments and is verified by an analytical model. This study elucidates under which conditions diving birds may possibly get injured.

10:11AM H13.00003 Hydrodynamic forces during the initial stage of body lifting from water surface. PATRICIA VEGA-MARTINEZ, JAVIER RODRIGUEZ-RODRIGUEZ, Carlos III University of Madrid, A KO-ROBKIN, TATYANA KHABAKHPASHEVA, University of East Anglia — We consider the flow induced by a rigid flat plate, initially touching a horizontal water surface, when it starts to move upwards with constant acceleration. Negative hydrodynamic pressures on the wetted surface of the plate are allowed, thus the water follows the plate due to the resulting suction force. The acceleration of the plate and the plate length are such that gravity, surface tension and viscous effects can be neglected. Under these assumptions, the potential flow caused by the plate lifting is obtained by using the small-time expansion of the velocity potential. This small-time solution fails close to the plate edges, as it predicts there singular velocities and unbounded displacements of the free surface. It is shown that close to the plate edges the flow is non-linear and self-similar in the leading order. This nonlinear flow is computed by the boundary element method combined with a time-marching scheme. We also report high-speed high impact velocity flow visualization results for a large cone angle and varying horizontal forces as well as for a wide range of entry velocities. Negligible interface slope with a localized shock. Instead, a generalized inertial lubrication theory seems able to explain these behaviors, that we relate to finite slope effects at the free surface. [1] T. Bohr et al., JFM 254, 635 (1993). [2] A. Duchesne et al., EPL 107, 54002 (2014). [3] N Rojas et al., PRL 104, 187801 (2010).

11:14AM H13.00004 Numerical investigation of the water entry of cylinders with and without spin. ARETI KIARA, RUBEN PAREDES, DICK K.P. YUE, Massachusetts Institute of Technology — We perform laminar, weakly compressible, numerical simulations of water impacting cylinders with radius $R$, entry velocity $V$, and spin $\omega$ about their axis. We consider two Froude numbers $Fr = V/\sqrt{gR} = 0.5, 1.5$ and moderate spin ratios $\Omega = \omega R/V < 3$. Our numerical predictions are in agreement with experiment and identify the effects of $Fr$ and $\Omega$ on the separation points, flow dynamics, and body trajectory. We find that the separation points depend primarily on $Fr$ and observe two distinct regimes: for $Fr = 0.5$ quasi-static cavities are obtained, while for $Fr = 1.5$ the separation points approach a limit angle of $70^\circ - 80^\circ$ with respect to the negative vertical axis. For times $t/t_F > 0.1$ the total pressure force on the cylinder decreases with $Fr$, obtaining significantly larger values for $Fr = 0.5$. The corresponding drag reduces with $\Omega$, while lift is towards the windward side and increases with both $\Omega$ and time. As a consequence, free-falling spinning cylinders drop slightly faster, while at a given depth their lateral displacement increases with $\Omega$.

11:27AM H13.00005 Spray Formation during the Impact of a Flat Plate on Water Surface. AN WANG, JAMES H. DUNCAN, University of Maryland-College Park — Spray formation during the impact of a flat plate on a water surface is studied experimentally. The plate is mounted on a two-axis carriage that can slam the plate vertically into the water surface as the carriage moves horizontally along a towing tank. The plate is 122 cm by 38 cm and oriented with adjustable pitch and roll angle. The plate is clamped at the edges of the plate is positioned with a 3-mm gap from one of the tank walls. A laser sheet is created in a plane oriented perpendicular to the axis of the horizontal motion of the carriage. The temporal evolution of the spray within the light sheet is measured with a cinematic laser induced fluorescence technique at a frame rate of 800 Hz. Experiments are performed with a fixed plate trajectory in a vertical plane, undertaken at various speeds. Two types of spray are found when the plate has nonzero pitch and roll angles. The first type is composed of a cloud of droplets formed as the plate impacts the water surface during the initial impact. The second type is a thin sheet of water that grows from the starboard edge of the plate as it moves below the local water level. The geometrical features of the spray are found to be dramatically affected by the impact velocity.

1 The support of the Office of Naval Research under grant N00014131058 is gratefully acknowledged.
11:40AM H13.00006 Ringin’ the water bell: dynamic modes of curved fluid sheets. JOHN KOLINSKI, HILLEL AHARONI, JAY FINEBERG, ERAN SHARON, Racah Institute of Physics, Hebrew University of Jerusalem, Jerusalem 91904, Israel — A water bell is formed by fluid flowing in a thin, coherent sheet in the shape of a bell. Experimentally, a water bell is created via the impact of a cylindrical jet on a flat surface. Its shape is set by the splash angle (the separation angle) of the resulting cylindrically symmetric water sheet. The separation angle is altered by adjusting the height of a lip surrounding the impact point, as in a water sprinkler. We drive the lip’s height sinusoidally, altering the separation angle, and ringin’ the water bell. This forcing generates disturbances on the steady-state water bell that propagate forward and backward in the fluid’s reference frame at well-defined velocities, and interact, resulting in the emergence of an interference pattern unique to each steady-state geometry. We analytically model these dynamics by linearizing the amplitude of the bell’s response about the underlying curved geometry. This simple model predicts the nodal structure over a wide range of steady-state water bell configurations and driving frequencies. Due to the curved water bell geometry, the nodal structure is quite complex; nevertheless, the predicted nodal structure agrees extremely well with the experimental data. When we drive the bell beyond perturbative separation angles, the nodal locations surprisingly persist, despite the strikingly altered underlying water bell shape. At extreme driving amplitudes the water sheet assumes a rich variety of tortuous, non-convex shapes; nevertheless, the fluid sheet remains intact.

11:53AM H13.00007 Polygonal instabilities. MATTHIEU LABOUSSE, University Paris Diderot — The interaction of a vortex with a free surface is encountered in a series of experiments, the hydraulic jump [1], the hydraulic bump [2], the toroidal Leidenfrost experiment [3]. All these experiments share in common an unstable configuration in which azimuthal perturbations give rise to polygonal patterns. We propose a unified theoretical framework to model the emergence of this instability by investigating the stability of a liquid torus with a poloidal motion [4]. As simple as it is, we show that the model retains the necessary ingredients to account for the experimental observations. In this talk, I will first describe the model and compare it to the existing data. However this model is purely inviscid and reaches its limits when being applied to relatively moderate Reynolds flows. So in a second part, I will present a recent experimental and theoretical investigation in which polygonal patterns are now driven by Marangoni flows [5]. To our great surprise, it extends the range of validity of the initial proposed framework, much more than initially expected.

12:06PM H13.00008 ABSTRACT WITHDRAWN —


10:35AM H14.00001 Dielectric decrement effects in electrokinetics. BRUNO FIGLIUZZI, Ecole des Mines ParisTech, WAI HONG RONALD CHAN, Stanford University, CULLEN BUIE, JEFFREY MORAN, Massachusetts Institute of Technology — Understanding the nonlinear phenomena that occur in the electric double layer (EDL) that forms at charged surfaces is a key issue in electrokinetics. In recent studies, Nakayama and Andelman [J. Chem. Physics 2015] Hatlo et al. [EPL 2012], and Zhao and Zhai [JFM 2013] demonstrated that dielectric decrement significantly influences the ionic concentration in the electric double layer (EDL) at high zeta potential, leading to the formation of a condensed layer near the particle’s surface. In this presentation, we apply the dielectric decrement model to study two archetypal problems in electrokinetics, namely the electrophoresis of particles with fixed surface charges and the electrophoresis of ideally polarizable particles. Our aim is to rely on numerical simulations to incorporate nonlinear effects including crowding effects due to the finite size of ions, dielectric decrement in the EDL, surface conduction, concentration polarization and advection in the bulk solution. In parallel, we derive a simplified composite layer model that enables us to obtain analytical estimates of the physical quantities involved in the physical description of the problem.

10:48AM H14.00002 The influence of soft layer electrokinetics on bacterial electroporation. JEFFREY MORAN, NAGA NEEHAR DINGARI, CULLEN BUIE, Massachusetts Institute of Technology — Electroporation of mammalian cells has received a significant amount of theoretical attention over the last decade because of its ability to deliver biologically active molecules into cells using short and strong electric field pulses. However, application of the same theory to bacterial electroporation presents significant challenges because of the presence of charged soft layers around bacteria. The soft layer charge distribution has been found to significantly influence bacterial electrophoretic mobility and polarizability because it alters the electric potential spatial distribution around the cell envelope. In addition, it also affects the relaxation rate of the soft layer and in some cases the order of magnitude of the pore creation time scale. Therefore in this study, we investigate the influence of soft layer electrokinetics on the spatial pore distribution and the temporal pore evolution during bacteria electroporation, which are quantitative measures of a bacterium’s amenability to electroporation. The study will have significant impact on designing and optimizing bacteria electroporation platforms for gene and drug delivery applications.

11:01AM H14.00003 Frequency dispersion of electrokinetically activated Janus particles. ALICIA BOYMELGREEN, TOV BALLI, GILAD YOSSIFON, Technion - Israel Institute of Technology, TOUVIA MILOH, Tel Aviv University — We examine the influence of the applied frequency of the electric field on the induced-charge electroosmotic flow around a metallo-dielectric Janus particle. Previously, we have used three dimensional-two component micro-particle-image-velocimetry (3D-2C µPIV) to study a stagnant particle, to illustrate the presence of a number of competing effects including dielectrophoresis and electrohydrodynamic flow which distorts both the strength and shape of the frequency dispersion predicted for pure induced-charge effects. Here, we extend this work by examining the frequency dispersion of mobile Janus particles of different sizes (3–15µm in diameter) at different electrolyte concentrations. In all cases, towards the DC limit, and in the frequency domain where previously EHD flow was shown to dominate, the velocity of a mobile particle decays to zero. At the same time significant variations in the frequency dispersion, including its shape and the value for maximum velocity are recorded as a function of both electrolyte concentration and particle size. This work is of both fundamental and practical importance and may be used to further refine non-linear electrokinetic theory and optimize the application of Janus particles as carriers in lab-on-a-chip analysis systems.
Ra and chaotic induced electroconvection in both gravitationally stable and unstable configurations via direct numerical simulations of a model depending on the geometric orientation of electrokinetic systems. In this study we thoroughly examine the interplay of gravitational convection and electroconvective flows. Additionally, we show the formation of novel electroconvective patterns with multiple coexisting length scales due to nominal threshold of instability, patterning enhances transport by inducing flow, while at higher voltages they do so by regularizing the chaotic interactions using direct numerical simulation of the governing equations. We show that despite the reduced surface area available for transport, two sources of electroconvection at ion-selective surfaces, electrokinetic instability and surface-patterning with impermeable stripes and their interactions using direct numerical simulation of the governing equations. We show that despite the reduced surface area available for transport, patterned surfaces can lead to an up to 80% increase in current density relative to homogenous surfaces. At applied voltages below the nominal threshold of instability, patterning enhances transport by inducing flow, while at higher voltages they do so by regularizing the chaotic electroconvective flows. Additionally, we show the formation of novel electroconvective patterns with multiple coexisting length scales due to electrokinetic instability at the homogeneous surface.

This work has been supported by the research grants no. 220900 of Consejo Nacional de Ciencia y Tecnología (CONACYT) and 20150919 of SIP-IPN at Mexico. F. Méndez acknowledges also the economical support of PAPIIT-UNAM under contract number IN112215

1Supported by Netherlands Organization for Scientific Research, Rubicon Grant

Investigation of Current Hotspots on an Ion-Selective Membrane Subject to Chaotic Electroconvection, CLARA DRUZGALSKI, ALI MANI, Stanford University — We have performed a 3D direct numerical simulation (DNS) of chaotic ion transport associated with electroconvective instability near an ion-selective membrane. Data from the 3D DNS demonstrate that the chaotic fluid motion substantially influences the transport of ions and causes instantaneous hotspots of high current density on the surface. We present a comprehensive statistical analysis of surface current density, including probability density functions (PDFs) and joint-PDFs with other interfacial measures involving flow, conductivity and electric fields. These results provide new insights into the mechanism and characterization of current hotspots. Our results are relevant to industrial applications involving ion-selective interfaces such as electrodialysis for water purification, and emerging microfluidic devices that use ion-selective components for separation processes.

Enhancement of Overlimiting Current through an Ion-Selective Membrane via Surface Conductivity Patterning, SCOTT DAVIDSON, Stanford University, MATTHIAS WESSLING, RWTH Aachen University, ALI MANI, Stanford University — Electroconvection’s ability to enhance transport through ion-selective surfaces provides promising opportunities for improving many diffusion-limited electrochemical and microfluidic systems. We have investigated two sources of electroconvection at ion-selective surfaces, electrokinetic instability and surface-patterning with impermeable stripes and their interactions using direct numerical simulation of the governing equations. We show that despite the reduced surface area available for transport, patterned surfaces can lead to an up to 80% increase in current density relative to homogenous surfaces. At applied voltages below the nominal threshold of instability, patterning enhances transport by inducing flow, while at higher voltages they do so by regularizing the chaotic electroconvective flows. Additionally, we show the formation of novel electroconvective patterns with multiple coexisting length scales due to electrokinetic instability at the homogeneous surface.

Asymmetry induced electric current rectification in permselective systems, YOAV GREEN, YARON EDRI, GILAD YOSSIFON, TECHNION — Permselective systems are inherently asymmetric as they have preference to the transport of one charge carrier over the other. In this work we derive a solution for the concentration distribution, electric potential and current-voltage response for a four-layered system comprised of two microchambers connected by two permselective regions of varying property. We show that any additional asymmetry in the system, in terms of the geometry, bulk concentration, or surface charge property of the permselective regions, results in rectification of the current. Our work is divided in two parts, when both permselective regions have the same sign surface charge sign and the case of opposite signs. For the same sign charge we are able to show that the system behaves as Dialyctic battery. For the case of opposite signs, our system exhibits the bipolar behavior of a diode where the magnitude of the rectification can be of order $O(100-1000)$.

Direct Observation of Three-dimensional Electroconvective Vortices on a Charge Selective Surface, RHOKYUN KWAK, Korea Institute of Science and Technology, JONGYOON HAN, Massachusetts Institute of Technology, TAIKJIN LEE, Korea Institute of Science and Technology, HO-YOUNG KWAK, Chung-Ang University — We present a visualization of three-dimensional electroconvective vortices (EC) by ion concentration polarization (ICP) on a cation selective membrane. The vortices are initiated between two transparent Nafion membranes under no-shear/shear conditions with various applied voltages and flow velocities. Fluorescent imaging and spatial Fourier transform allow us to capture vortex structures. In this 3-D system, EC shows three distinguished structures: i) polygonal shapes with no-shear and ii) transverse and/or iii) longitudinal vortex rolls with shear flow, which is reminiscent of 3-D Rayleigh-Benard instability. Under shear flow, as flow velocity (Reynolds number: Re) increases or voltage (electric Rayleigh number: Ra) decreases, pure longitudinal vortices are presented; in the inverse case, transverse vortices are also formed. It is noteworthy that if we confine EC in quasi-2-D system with high Ra (>10,000), we obtain pure transverse vortices (Kwak et al., PRL, 110, 114501 (2013)), high Ra induces chaotic EC in this 3-D system, instead of 2-D stable transverse vortices. To the best of our knowledge, this is the first direct observation of 3-D EC, which will occur in realistic electrochemical devices, e.g. electrodialysis.

Non-isothermal electro-osmotic flow in a microchannel with charge-modulated surfaces, OSCAR BAUTISTA, Instituto Politecnico Nacional, SALVADOR SANCHEZ, FEDERICO MENDEZ, Universidad Nacional Autonoma de Mexico — In this work, we present an analytical solution of a nonisothermal electro-osmotic flow of a Newtonian fluid over charge-modulated surfaces in a microchannel. Here, the heating in the microchannel is due to the Joule effect caused by the imposition of an external electric field. The study is conducted through the use of perturbation techniques and is validated by means of numerical simulations. We consider that both, viscosity and electrical conductivity of the fluid are temperature-dependent; therefore, in order to determine the heat transfer process characteristics, the governing equations of continuity, momentum, energy and electric potential have to be solved in a coupled manner. The principal obtained results evidence that the flow patterns are perturbed in a noticeable manner in comparison with the isothermal case. Our results may be used for increasing microfluidics mixing by conjugating thermal effects with the use of charge-modulated surfaces.

1Supported by Netherlands Organization for Scientific Research, Rubicon Grant
12:32PM H14.00010 Uncertainty Quantification of Nonlinear Electrokinetic Response in a Microchannel-Membrane Junction, SHIMA ALIZADEH1, GIANLUCA IACCARINO, ALI MANI, Stanford University — We have conducted uncertainty quantification (UQ) for electrokinetic transport of ionic species through a hybrid microfluidic system using different probabilistic techniques. The system of interest is an H-configuration consisting of two parallel microchannels that are connected via a nanofluid junction. This system is commonly used for ion preconcentration and stacking by utilizing a nonlinear response at the nanofluid-nanofluid junction that leads to deionization shocks. In this work, the nanofluid medium is modeled as a parallel nano-pores where, the nano-pore diameter, nanofluid porosity, and surface charge density are independent random variables. We evaluated the resulting uncertainty on the ion concentration fields as well as on the deionization shock location. The UQ methods predicted consistent statistics for the outputs and the results revealed that the shock location is weakly sensitive to the nanofluid surface charge and primarily driven by nano-pore diameters. The present study can inform the design of electrokinetic networks with increased robustness to natural manufacturing variability. Applications include water desalination and lab-on-a-chip systems.

Monday, November 23, 2015 10:35AM - 12:45PM –
Session H15 Flow Control: Drag Reduction II 203 - Taraneh Sayadi, University of Illinois at Urbana-Champaign

10:35AM H15.00001 Optimization of dynamic roughness elements for reducing drag in a laminar boundary layer, TARANEH SAYADI, Department of Aerospace Engineering, University of Illinois at Urbana-Champaign, PETER SAYADI, Department of Mathematics, Imperial College London — Roughness elements can serve as controllers in both laminar and turbulent regimes to, for example, reduce the skin friction or drag. In this study, adjoint-based optimization is employed to extract the optimal shape of roughness elements for reducing drag, in a laminar setting, given an initial condition. The roughness elements considered here are of the dynamic type, varying both in space and time, which allows control over the spatial distribution of the roughness but also the inherent timescales of the flow. Dynamic roughness is modeled here using the linearized boundary conditions previously introduced by McKeon (2008), where the no-slip and impermeability boundary conditions are replaced by stream-wise and wall-normal distributions at the wall. The adjoint equation is then implemented using the discretized approach by Fosas et al. (2012). This approach is particularly efficient, since the linearized operators are computed simply by using the local differentiation technique, without explicitly forming the resulting matrices for both forward and adjoint operators. Using the described framework we investigate the effect of the initial condition on the spatial distribution of the roughness elements and their variation in time as the drag coefficient is minimized.

10:48AM H15.00002 Development of Drag Reducing Polymer of FDR-SPC, INWON LEE, HYUN PARK, HO HWAN CHUN, Pusan Natl Univ — In this study, a novel FDR-SPC (Frictional Drag Reduction Self-Polishing Copolymer) is first synthesized in this study. The drag reducing functional radical such as PEGMA (Poly(ethylene) glycol methacrylate) has been utilized to participate in the synthesis process of the SPC. The release mechanism of drag reducing radical is accounted for the hydrolysis reaction between the FDR-SPC and seawater. The types of the baseline SPC monomers, the molecular weight and the mole fraction of PEGMA were varied in the synthesis process. The resulting SPCs were coated to the substrate plates for the subsequent hydrodynamic test for skin friction measurement. A significant reduction in Reynolds stress was observed in a range of specimen, with the maximum drag reduction being 15.9% relative to the smooth surface for PRD3-1.

11:01AM H15.00003 AFRODITE – passive flow control for skin-friction drag reduction using the method of spanwise mean velocity gradient1, BENGT FALLENIUS, SOHRAB SATTARZADEH, ROBERT DOWNS, SHAHAB SHAHINFAR, JENS FRANSSON, KTH — Royal Institute of Technology — Over the last decade wind tunnel experiments and numerical simulations have shown that steady spanwise mean velocity gradients are able to attenuate the growth of different types of boundary layer disturbances. Within the AFRODITE research program different techniques to setup the spanwise mean velocity variations have been studied and their stabilizing effect leading to transition delay quantified. A successful boundary-layer modulator for transition delay has turned out to be the miniature-vortex generator and has been well documented during the past years. More recent ideas of setting up spanwise mean velocity gradients will be presented here. We show that, the non-linear interaction between a pair of oblique disturbance waves creating a streaky base flow, as well as the direct surface modulation by means of applying wavy surfaces in the spanwise direction, can both successfully be utilized for transition delay and hence skin-friction drag reduction.

11:14AM H15.00004 Streamwise shear stress driven compliant wall for drag reduction, TAMÁS ISTVÁN JÓZSA, IGNAZIO MARIA VIOLA, University of Edinburgh, ELIAS BALARAS, The George Washington University — The interaction between a viscous fluid and a solid wall in relative motion to each other leads to wall shear stress, which results in often undesirable friction drag. In fully turbulent flow, it has been shown that a compliant wall whose streamwise velocity is equal to the streamwise flow velocity fluctuation in the buffer layer can lead to drag reduction (Choi et al., JFM, 1994; 262:75-110). Practical exploitation of this mechanism would require knowledge of the instantaneous velocity fluctuations in the near-wall region and active control of the wall velocity. However, the near-wall fluid velocity can be approximated by the wall shear stress through a first-order Taylor expansion; therefore we propose a passively controlled compliant wall whose streamwise wall velocity is driven by the streamwise wall shear stress fluctuations. We show that this wall behaviour can be modelled with a damped harmonic oscillator, where the damping coefficient is related to the target distance of the flow fluctuation from the wall. Our results suggest that a passively controlled shear stress driven compliant wall can be developed for drag reduction. On-going works include the use of direct numerical simulation where the proposed slip condition is applied to quantify the potential drag reduction.

1The European Research Council is gratefully acknowledged (ERC-StG-2010- 285339)
11:27AM H15.00005 Turbulent boundary layer control through spanwise wall oscillation using Kagome lattice structures. JAMES BIRD, MATTHEW SANTER, JONATHAN MORRISON, Imperial College London — It is well established that a reduction in skin-friction and turbulence intensity can be achieved by applying in-plane spanwise forcing to a surface beneath a turbulent boundary layer. It has also been shown in DNS (M. Quadrio, P. Ricco, & C. Viotti; J. Fluid Mech; 627, 161, 2009), that this phenomenon is significantly enhanced when the forcing takes the form of a streamwise travelling wave of spanwise perturbation. In the present work, this type of forcing is generated by an active surface comprising a compliant structure, based on a Kagome lattice geometry, supporting a membrane skin. The structural design ensures negligible wall normal displacement while facilitating large in-plane velocities. The surface is driven pneumatically, achieving displacements of 3 mm approximately, at frequencies in excess of 70 Hz for a turbulent boundary layer at $Re_x \approx 1000$. As the influence of this forcing on boundary layer is highly dependent on the wavenumber and frequency of the travelling wave, a flat surface was designed and optimised to allow these forcing parameters to be varied, without reconfiguration of the experiment. Simultaneous measurements of the fluid and surface motion are presented, and notable skin-friction drag reduction is demonstrated.

11:40AM H15.00006 Lubricant retention for liquid infused surfaces exposed to turbulent flow. MATTHEW FU, MARCUS HULTMARK, Princeton University — Liquid infused surfaces have been proposed as a robust alternative to traditional, air-filled superhydrophobic surfaces. The mobility of the liquid lubricant facilitates a surface slip with the outer turbulent shear flow. However, shear driven drainage in turbulent flow has been found to be a primary failure mechanism for such surfaces, resulting in loss of lubricant and the associated slip effect. A turbulent channel flow facility is used to characterize shear-driven drainage behavior of liquid infused micro-patterned surfaces. Micro-manufactured surfaces can be mounted flush in the channel and exposed to turbulent flows. The retention of fluorescent lubricants is monitored to characterize how surface geometry and lubricant properties affect the steady state retention length. Results are compared with theoretical predictions and experiments for lubricant retention in laminar microchannels, where the shear driven drainage is balanced by a Laplace pressure gradient, to determine the additional drainage induced by turbulent fluctuations.

11:53AM H15.00007 A Turbulent Boundary Layer over Superhydrophobic Surfaces. HYUNWOOK PARK, JOHN KIM, University of California, Los Angeles — Direct numerical simulations of a spatially developing turbulent boundary layer (TBL) developing over superhydrophobic surfaces (SHS) were performed in order to investigate the underlying physics of turbulent flow over SHS. SHS were modeled through the shear-free boundary condition, assuming that the gas-liquid interfaces remained as non-deformable. Pattern-averaged turbulence statistics were examined in order to determine the effects of SHS on turbulence in no-slip and slip regions separately. Near-wall turbulence over the slip region was significantly affected by SHS due to insufficient mean shear required to sustain near-wall turbulence. SHS also indirectly affected near-wall turbulence over the no-slip region. In addition to the effects of the spanwise width of SHS on skin-friction drag reduction reported previously, spatial effects in the streamwise direction were examined. A guideline for optimal design of SHS geometry will be discussed.

12:06PM H15.00008 ABSTRACT WITHDRAWN –

12:19PM H15.00009 Turbulent flows over superhydrophobic surfaces with shear-dependent slip length. SOHRAB KHOSH AGHDAH, MEHDI SEDDIGHI, PIERRE RICCO, University of Sheffield — Motivated by recent experimental evidence, shear-dependent slip length superhydrophobic surfaces are studied. Lyapunov stability analysis is applied in a 3D turbulent channel flow and extended to the shear-dependent slip-length case. The feedback law extracted is recognized for the first time to coincide with the constant-slip-length model widely used in simulations of hydrophobic surfaces. The condition for the slip parameters is found to be consistent with the experimental data and with values from DNS. The theoretical approach by Fukagata (PoF 18:5: 051703) is employed to model the drag-reduction effect engendered by the shear-dependent slip-length surfaces. The estimated drag-reduction values are in very good agreement with our DNS data. For slip parameters and flow conditions which are potentially realizable in the lab, the maximum computed drag reduction reaches 50%. The power spent by the turbulent flow on the walls is computed, thereby recognizing the hydrophobic surfaces as a passive-absorbing drag-reduction method, as opposed to geometrically-modifying techniques that do not consume energy, e.g. riblets, hence named passive-neutral. The flow is investigated by visualizations, statistical analysis of vorticity and strain rates, and quadrants of the Reynolds stresses.

12:32PM H15.00010 Turbulent drag reduction by permeable coatings. RICARDO GARCIA-MAYORAL, NABIL ABDERRAHAMAN-ELENA, University of Cambridge — We present an assessment of permeable coatings as a form of passive drag reduction, proposing a simplified model to quantify the effect of the coating thickness and permeability. To reduce skin friction, the porous layer must be preferentially permeable in the streamwise direction, so that a slip effect is produced. For fixed permeability, the controlling parameter is the difference between streamwise and spanwise permeability lengths, scaled in viscous units, $\sqrt{x^+} - \sqrt{y^+}$. In this regime, the reduction in drag is proportional to that difference. However, the proportional performance eventually breaks down for larger permeabilities. A degradation mechanism is investigated, common to other obstructed surfaces in general and permeable substrates in particular, which depends critically on the geometric mean of the streamwise and wall-normal permeabilities, $\sqrt{x^+ y^+}$. For a streamwise-to-cross-plane permeability ratio of order $K_x^+ / K_y^+ = K_z^+ / K_z^+ \sim 10-100$, the model predicts a maximum drag reduction of order 15-25%.

Monday, November 23, 2015 10:35AM - 12:45PM –
Session H16 Aerodynamics: Fluid-Structure Interaction l 204 - Lucy Zhang, Rensselaer Polytechnic Institute
10:35AM H16.00001 The Effectiveness of the Perfectly Matched Layer in Fluid-Structure Interaction Problems. LUCY ZHANG, JUBIAO YANG, Rensselaer Polytechnic Institute — It is well recognized that spurious reflections on computational domain boundaries can have contamination of the flow field when solving fluid and/or wave equations. The effects are even more pronounced in fluid-structure interaction (FSI) problems, since the solid responses may be distorted due to the contaminated flow field. In this work, we implemented the perfectly matched layer (PML) technique and applied it in our fully-coupled immersed finite element method (IFEM), where Navier-Stokes equations are solved in the fluid domain with finite element method. With PML included as an absorbing layer it successfully absorbs outgoing waves from the interior of the computational domain and therefore keeps them from reflecting back from the computational boundary. Validation cases are shown to demonstrate the effectiveness of the PML in pure computational fluid dynamics cases, and then followed by FSI problems.

10:48AM H16.00002 Characterizing self-excited fluidic energy harvesters subjected to Vortex Induced Vibration by utilizing Griffin scaling1, NIELL ELVIN, VAHID AZADEH RANJBAR, YIANNIS ANDREOPOULOS, CUNY-CCNY — The present work has experimentally characterized energy harvesters consisting of a circular cylinder mounted at the tip of a flexible cantilever beam. VIV phenomena such as lock-in range, maximum amplitude of transverse oscillation and hysteresis effects have been studied by testing different physical parameters such as structural damping, mass ratio, and aspect ratio. Griffin plot generated by the experimental data of SDOF high aspect ratio circular cylinders have been used to validate VIV. As the harvester is a continuous system of low aspect ratio circular cylinders, three cases have been investigated: low aspect ratio effect of cylinders, effect of multiple modes or coupled transverse-torsional oscillation and non-linear effect due to large deformation of flexible cantilever beams. Griffin plot shows large variance in the case of aspect ratios less than 3. Coupled transverse-torsional oscillation affects VIV negatively. Results show that added structural damping due to piezoelectric patches attached to the cantilever beam decreases electrical power output as a non-linear function of mass ratio.

1Work supported by National Science Foundation under Grant No. CBET #1033117.

11:01AM H16.00003 Flow structure interaction between a flexible cantilever beam and isotropic turbulence1, ANDREW VOGEL, Columbia University, THOMAS MORVAN, Université de Nantes, OLEG GOUSSCHA, YIANNIS ANDREOPOULOS, NIELL ELVIN. The City College of New York — In the present experimental work we consider the degree of distortion of isotropy and homogeneity of grid turbulence caused by the presence of a thin flexible cantilever beam immersed in the flow aligned in the longitudinal direction. Beams of various rigidities and lengths were used in the experiments. Piezoelectric patches were attached to the beams which provided an output voltage proportional to the strain and therefore proportional to the beam’s deflection. The experiments were carried out in a large scale wind tunnel and hot-wires were used to measure turbulence intensity in the vicinity of the beams for various values of the ratio of aerodynamic loading to beam’s rigidity. It was found that the flow field distortion depends on the rigidity of the beam. For very rigid beams this distortion is of the order of the boundary layer thickness developing over the beam while for very flexible beams the distorted region is of the order of the beam’s tip deflection. Analysis of the time-dependent signals indicated some correlation between the frequency of beam’s vibration and flow structures detected.

1Supported by NSF Grant: CBET #1033117.

11:14AM H16.00004 An Immersed-Boundary method for deformable bodies at high Reynolds numbers. DARIO DE MARINI1, 2, Dipartimento di Meccanica, Matematica e Management – Politecnico di Bari, SREENATH KRISHNAN2, Mechanical Engineering - Stanford University, MARCO DONATO DE TULLIO3, MICHELE NAPOLITANO3, GIUSEPPE PASCAZIO2, Dipartimento di Meccanica, Matematica e Management – Politecnico di Bari, GIANLUCA IACCARINO3, Mechanical Engineering - Stanford University — With the aim of accurately simulate the flow-field through gas turbine blades a numerical approach is presented, that couples a massively parallel, finite volume Unsteady Reynolds Averaged Navier–Stokes Equations solver with an efficient structural solver describing the dynamics of deformable bodies, using an iterative coupled approach. The numerical strategy is based on a suitable version of the immersed boundary (IB) technique, which is able to handle rigid and deformable complex geometries in turbulent flows. The structures are discretized by a surface mesh of three-node triangular elements and modeled by means of a finite element method. The solution of the fluid-structure-interaction (FSI) problem produces detailed information of the flow patterns through realistic geometries subject to small deformations at high Reynolds and Mach numbers.

1Via Orabona 4 - 70125 - Bari, Italy
2488 Escondido Mall, Bld. 500, CA 94305, USA
3 Via Orabona 4 - 70125 - Bari, Italy
4 Via Orabona 4 - 70125 - Bari, Italy
5 Via Orabona 4 - 70125 - Bari, Italy
6488 Escondido Mall, Bld. 500, CA 94305, USA

11:27AM H16.00005 Dynamical behaviors of a thin plate under bypassing flow. ANGXIU NI, QIQI WANG, Massachusetts Inst of Tech-MIT — The interaction of air flow and flexible structure could lead to complex dynamic behaviors. Here the motion of a thin plate under a bypassing air flow is numerically investigated. Dimensional analysis is carried out and all pertinent dimensionless parameters are found. The appropriate levels of fidelity needed for modeling the structural and aerodynamic behaviors are studied. The effect of ratios between aerodynamic force, bending force and external compression force on the plate are investigated. In general, the aerodynamic force causes the plate to oscillate, the external compression causes it to buckle, while the bending force tries to keep the plate flat. When the bending force is relatively small and the other two forces are comparable, the airflow strongly couples with the nonlinear feedback of the structure, and the system undergoes chaotic motion. We compare this complex motion at different Mach numbers against the classic high-Mach number result by Dowell.
Ground Effect is chosen such that the ellipse reaches stable equilibrium at an angle of roughly 45°. Using Lattice-Boltzmann simulations, we study angular oscillations of an elliptical cylinder attached to a torsional spring, with the axis placed perpendicular to a uniform flow, at low Reynolds numbers (Re = 100 and Re = 200). The equilibrium angle and stiffness of the torsional spring is chosen such that the ellipse reaches stable equilibrium at an angle of roughly 45° with respect to the incoming flow. This configuration leads to large unsteady torque due to vortex shedding, which in turn can lead to large oscillations of the ellipse, with several frequency modes. Along with measuring the angular oscillations of the ellipse, we also measure the potential for power-extraction from this setup, by attaching an external angular damper to the axis of the ellipse. For low density ratios, the ellipse tends to oscillate within the first quadrant, while, for higher density ratios, the ellipse, due to its tendency to auto-rotate, undergoes very large oscillations. The ellipse locks on to primary and secondary vortex shedding modes over the range of density ratios studied here. The power output of this setup increases with increasing Reynolds number and density ratio, with peak efficiency of 1.7%.

Squirt flow is a phenomenon that typically occurs in porous structures with more than one length scale, e.g., in fractured rocks or multi-porosity organic materials. Due to a heterogeneous pore space, external compression induces fluid flow between the pores of different compressibility and finally causes a delayed and attenuated response. While this phenomenon is well understood in natural materials, little is known about how to trigger and control it in artificially architected materials. Here, we will first show that squirt flow can occur in highly deformable, fluid-filled artificial materials if overall fluid drainage is prevented and then we will demonstrate how this can be controlled. Interestingly, this viscous-flow mechanism opens avenues for the design of smart materials with delayed stress-strain response (e.g., for high-impact applications) or additional attenuation regimes (e.g., below frequencies of internal resonance).

**12:06PM H16.00008 Energy Extraction from Fluid Flow Via Vortex Induced Angular Oscillations**

Using Lattice-Boltzmann simulations, we study angular oscillations of an elliptical cylinder attached to a torsional spring, with the axis placed perpendicular to a uniform flow, at low Reynolds numbers (Re = 100 and Re = 200). The equilibrium angle and stiffness of the torsional spring is chosen such that the ellipse reaches stable equilibrium at an angle of roughly 45° with respect to the incoming flow. This configuration leads to large unsteady torque due to vortex shedding, which in turn can lead to large oscillations of the ellipse, with several frequency modes. Along with measuring the angular oscillations of the ellipse, we also measure the potential for power-extraction from this setup, by attaching an external angular damper to the axis of the ellipse. For low density ratios, the ellipse tends to oscillate within the first quadrant, while, for higher density ratios, the ellipse, due to its tendency to auto-rotate, undergoes very large oscillations. The ellipse locks on to primary and secondary vortex shedding modes over the range of density ratios studied here. The power output of this setup increases with increasing Reynolds number and density ratio, with peak efficiency of 1.7%.

**12:19PM H16.00009 Globally shed wakes for three distinct retracting foil geometries**

In quickly retracting foils at an angle of attack, the boundary layer vorticity along with the added mass energy is immediately and globally shed from the body into the surrounding fluid. The deposited vorticity quickly reforms into lasting vortex structures, which could be used for purposes such as manipulating or exploiting the produced flow structures by additional bodies in the fluid. The globally shed wake thus entrains the added mass energy provided by the initially moving body, reflected by the value of the circulation left in the wake. In studying experimentally as well as numerically this phenomenon, we find that the three different tested geometries leave behind distinct wakes. Retracting a square-ended foil is undesirable because the deposited wake is complicated by three-dimensional ring vorticity effects. Retracting a tapered, streamlined-tipped foil is also undesirable because the shape-changing aspect of the foil geometry actually induces energy recovery back to the retracting foil, leaving a less energetic globally shed wake. Finally, a retracting hollow foil geometry avoids both of these detrimental effects, leaving relatively simple, yet energetic, vortex structures in the wake.

**12:32PM H16.00010 POD Analysis of Flow Behind a Four-wing Vortex Generator**

Wing-tip vortices that persist long after the passage of large aircraft are of major concern to aircraft controllers and are responsible for considerable delays between aircraft take-off times. Understanding these vortices is extremely important, with the ultimate goal to reduce or eliminate delays altogether. Simple theoretical models of vortices can be studied experimentally using a four-wing vortex generator. The cross-stream planes are measured with a two-component Particle Image Velocimetry (PIV) system, and the resulting vector fields were analyzed with a Proper Orthogonal Decomposition (POD) via the method of snapshots. POD analysis will be employed both before and after removing vortex core meandering to investigate the meandering effect on POD modes for a better understanding of it.

**Monday, November 23, 2015 10:35AM - 12:45PM**

**Session H17 Flow Instability: Boundary Layers II**

10:35AM H17.00001 On role of kinetic fluctuations in laminar-turbulent transition in chemically nonequilibrium boundary layer flows

 Anatoli Tumin, The University of Arizona — Zavolskii and Reutov (1983), Lucini (2008, 2010), Fedorov (2010, 2012, 2014) explored potential role of kinetic fluctuations (KF) in incompressible and calorically perfect gas boundary layer flows. The results indicate that role of KF is comparable with other disturbance sources in flight experiments and in quiet wind tunnels. The analysis is based on the Landau and Lifshitz (1957) concept of fluctuating hydrodynamics representing the dissipative fluxes as an average and fluctuating parts. We are extending analysis of the receptivity problem to the fluctuating dissipative fluxes in chemically reacting nonequilibrium boundary layer flows of binary mixtures. There are new terms in the energy, and the species equations. The species conservation equation includes the dissipative diffusion flux and the species generation due to dissociation. The momentum equation includes fluctuating stress tensor. The energy equation includes fluctuating heat flux, energy flux due to diffusion of the species, and fluctuating dissipative flux due to viscosity. The effects are compared for the cases stemming from constraints of the HTV project (Kientzman and Tumin, AIAA Paper 2013-2882).

1 Supported by AFOSR
will allow us to investigate the destabilizing effects of acoustic feedback and baseflow changes. Sensitivity information can still be obtained from a weighted inner product between the optimal forcing and optimal response. This information followed here, the flow response is assessed by the calculation of the frequency response to optimal forcings for varying frequency at a given flow location with increasing the boundary layer scale is locked up to transition onset. Our measurements confirm previous results on the advancement of the transition location with increasing $\Lambda_\text{Tu}$ for low to moderate $\text{Tu}$ levels, but show the opposite effect for higher levels, i.e. a delay in the transition location for larger $\Lambda_\text{z}$, which to the knowledge of the present authors so far is unreported. In addition, the common belief that the FST length scales have a negligible effect on the transition location with regards to the $\text{Tu}$ level does not seem to be fully true.

The primary convective instability is of the same type as the instabilities one finds on a swept wing, called the crossflow instability. Here the development of this flow is investigated by direct numerical simulations (DNS) using both the linearised and fully nonlinear NSE. The main goal is to map out the instabilities and structures in the flow to investigate how the flow becomes turbulent. Linear simulations are already finalized and further nonlinear simulations allow investigation of the transition to turbulence of the realistic spatially-developing boundary layer, and these simulations can be directly compared with physical experiments of the same case. However, in contrast to experiments, the DNS provides an opportunity to eliminate certain instabilities in the flow field such that other instabilities can be investigated separately.

The majority of workers studying transition to turbulence in pipes have been interested in the flow response to perturbations in otherwise perfect pipes. Conversely, the fuzzy problem involving inlet disturbances, pipe imperfections, and pipe roughness has not attracted as much attention. Here, we investigate both experimentally and theoretically the transition to turbulence in imperfect millimeter-scale channels. For probing the flows, we use microcantilever sensors embedded in the channel walls. We perform experiments in two nominally identical channels. We quantify growing perturbations near the channel wall by their spectra and statistical properties, including probability densities and low- and high-order moments. The different sets of imperfections in the two channels result in two random flows in which the high-order moments of the near-wall fluctuations differ by orders of magnitude. Surprisingly, however, the lowest-order statistics in both cases appear to be qualitatively similar and can be described by a proposed noisy Landau equation for a slow mode. The noise, regardless of its origin, regularizes the Landau singularity of the relaxation time and makes transitions driven by different noise sources appear similar.

The primary convective instability is of the same type as the instabilities one finds on a swept wing, called the crossflow instability. Here the development of this flow is investigated by direct numerical simulations (DNS) using both the linearised and fully nonlinear NSE. The main goal is to map out the instabilities and structures in the flow to investigate how the flow becomes turbulent. Linear simulations are already finalized and further nonlinear simulations allow investigation of the transition to turbulence of the realistic spatially-developing boundary layer, and these simulations can be directly compared with physical experiments of the same case. However, in contrast to experiments, the DNS provides an opportunity to eliminate certain instabilities in the flow field such that other instabilities can be investigated separately.

The primary convective instability is of the same type as the instabilities one finds on a swept wing, called the crossflow instability. Here the development of this flow is investigated by direct numerical simulations (DNS) using both the linearised and fully nonlinear NSE. The main goal is to map out the instabilities and structures in the flow to investigate how the flow becomes turbulent. Linear simulations are already finalized and further nonlinear simulations allow investigation of the transition to turbulence of the realistic spatially-developing boundary layer, and these simulations can be directly compared with physical experiments of the same case. However, in contrast to experiments, the DNS provides an opportunity to eliminate certain instabilities in the flow field such that other instabilities can be investigated separately.

The primary convective instability is of the same type as the instabilities one finds on a swept wing, called the crossflow instability. Here the development of this flow is investigated by direct numerical simulations (DNS) using both the linearised and fully nonlinear NSE. The main goal is to map out the instabilities and structures in the flow to investigate how the flow becomes turbulent. Linear simulations are already finalized and further nonlinear simulations allow investigation of the transition to turbulence of the realistic spatially-developing boundary layer, and these simulations can be directly compared with physical experiments of the same case. However, in contrast to experiments, the DNS provides an opportunity to eliminate certain instabilities in the flow field such that other instabilities can be investigated separately.

The primary convective instability is of the same type as the instabilities one finds on a swept wing, called the crossflow instability. Here the development of this flow is investigated by direct numerical simulations (DNS) using both the linearised and fully nonlinear NSE. The main goal is to map out the instabilities and structures in the flow to investigate how the flow becomes turbulent. Linear simulations are already finalized and further nonlinear simulations allow investigation of the transition to turbulence of the realistic spatially-developing boundary layer, and these simulations can be directly compared with physical experiments of the same case. However, in contrast to experiments, the DNS provides an opportunity to eliminate certain instabilities in the flow field such that other instabilities can be investigated separately.

The primary convective instability is of the same type as the instabilities one finds on a swept wing, called the crossflow instability. Here the development of this flow is investigated by direct numerical simulations (DNS) using both the linearised and fully nonlinear NSE. The main goal is to map out the instabilities and structures in the flow to investigate how the flow becomes turbulent. Linear simulations are already finalized and further nonlinear simulations allow investigation of the transition to turbulence of the realistic spatially-developing boundary layer, and these simulations can be directly compared with physical experiments of the same case. However, in contrast to experiments, the DNS provides an opportunity to eliminate certain instabilities in the flow field such that other instabilities can be investigated separately.

The primary convective instability is of the same type as the instabilities one finds on a swept wing, called the crossflow instability. Here the development of this flow is investigated by direct numerical simulations (DNS) using both the linearised and fully nonlinear NSE. The main goal is to map out the instabilities and structures in the flow to investigate how the flow becomes turbulent. Linear simulations are already finalized and further nonlinear simulations allow investigation of the transition to turbulence of the realistic spatially-developing boundary layer, and these simulations can be directly compared with physical experiments of the same case. However, in contrast to experiments, the DNS provides an opportunity to eliminate certain instabilities in the flow field such that other instabilities can be investigated separately.
12:06PM H17.00008 Flat-plate boundary-layer receptivity to high amplitude, spanwise-oriented, vortical disturbances in the free-stream\textsuperscript{1}, RICHARD BOSWORTH, JONATHAN MORRISON, Imperial College London --- Extending previous experimental work on boundary-layer receptivity to harmonic free-stream forcing with roughness, the current study focuses on the high amplitude (Tu>1%), 2D case. It is shown that at these high amplitudes the boundary layer response is markedly different to the low amplitude case and that roughness is not required for this behaviour to appear. The behaviour within the boundary layer takes the form of low-frequency, streaky structures characteristic of `Klebanoff-modes' and the beginnings of bypass transition rather than 2D Tollmien-Schlichting waves. Traditionally, high turbulence levels are created with turbulence grids, however, the current setup uses qualitatively similar behaviour in a different manner allowing easy adjustment of the incoming disturbance amplitudes. This may provide a novel way to study different routes to transition - simply via the addition or subtraction of a roughness strip - in a single controllable experiment. The free-stream disturbances are created via a vibrating ribbon placed above and upstream of the plate leading edge and the disturbance signature is measured using a single cross-wire. Correlation measurements taken within the boundary layer provide details of the streaky structures present.

\textsuperscript{1}This work is carried out under the LFC-UK grant

12:19PM H17.00009 Frequency response of the swept-wing three-dimensional boundary layer, GIANLUCA MENEGHELLO, THOMAS BEWLEY, UC San Diego, UCSD FLOW CONTROL LAB TEAM --- Three dimensional boundary layers are well known for being subject to the growth of perturbations and transition to turbulence well below the linear stability threshold. Both their linear stability and the temporal transient growth have been analyzed in previous studies. In this work we focus on the response of the swept-wing attachment-line boundary layer to periodic external forcing e.g. incoming turbulence or wall roughness and we analyze the response amplitude in multiple system norms. In particular, the induced \( \| \cdot \|_2 \) norm (aka singular value norm) identifies the peak gain at a given forcing frequency. The input-output gain and the spatial structures associated with each frequency are identified by singular value decomposition and discussed.

12:32PM H17.00010 Non-parallel Flow Effects of Stationary Crossflow Vortices at their Genesis\textsuperscript{1}, ADAM BUTLER, XUESONG WU, Imperial College London --- We investigate the linear stability of stationary Crossflow vortices whose spanwise wavenumber is sufficiently small that non-parallel flow effects play a leading order role in determining their growth rate. The chordwise and spanwise variations of the baseflow and the perturbation are of equal importance, and so must both be accounted for. Neutral modes can occur in this regime, which lies close to the leading edge. If the effective pressure minimum occurs within this regime, a new scaling for the lower deck must be determined along with a new governing equation for the perturbation. When the mode from the non-parallel regime is continued through the pressure minimum, it passes into a critical layer in the form of a Cowley, Hocking, & Tutty instability. Downstream of the effective pressure minimum, this critical layer will eventually pass into the main body of the boundary layer. This CHT instability can occur in a more general setting, when the first three derivatives of the effective velocity profile are zero at the wall.

\textsuperscript{1}supported by a grant from LFC-UK

Monday, November 23, 2015 10:35AM - 12:45PM – Session H18 Flow Instability: Interfacial and Thin Films II 206 - Satish Kumar, University of Minnesota

10:35AM H18.00001 Electrohydrodynamic deformation of thin liquid films near surfaces with topography, SATISH KUMAR, ARUNA RAMKRISHNAN, University of Minnesota --- Motivated by the use of electrostatic assist to improve liquid transfer in gravure printing, we use theory and experiment to understand how electric fields deform thin liquid films near surfaces with cavity-like topographical features. Lubrication theory is used to describe the film dynamics, and both perfect and leaky dielectric materials are considered. For sinusoidal cavities, we apply asymptotic methods to obtain analytical results that relate the film deformation to the other problem parameters. For trapezoidal-like cavities, we numerically solve evolution equations to study the influence of steep topographical features and the spacing between cavities. Results from flow visualization experiments are in qualitative agreement with the theoretical predictions. In addition to being relevant to printing processes, the model problems we consider are also of fundamental interest and represent novel contributions to the areas of electrohydrodynamics and thin-liquid-film flows.

10:48AM H18.00002 Miscible viscous fingering involving production of gel by chemical reactions, YUICHIRO NAGATSU, KENICHI HOSHINO, Tokyo Univ of Agri & Tech --- We have experimentally investigated miscible viscous fingering with chemical reactions producing gel. Here, two systems were employed. In one system, sodium polyacrylate (SPA) solution and aluminum ion (Al\textsuperscript{3+}) solution were used as the more and less viscous liquids, respectively. In another system, SPA solution and ferric ion (Fe\textsuperscript{3+}) solution were used as the more and less viscous liquids, respectively. In the case of Al\textsuperscript{3+}, displacement efficiency was smaller than that in the non-reactive case, whereas in the case of Fe\textsuperscript{3+}, the displacement efficiency was larger. We consider that the difference in the change of the patterns in the two systems will be caused by the difference in the properties of the gels. Therefore, we have measured the rheological properties of the gels by means of a rheometer. We discuss relationship between the VF patterns and the rheological measurement.

11:01AM H18.00003 Sensitivity of Saffman–Taylor fingers to channel-depth variations, ANDRES FRANCO-GOMEZ, Manchester Centre for Nonlinear Dynamics and School of Physics and Astronomy, University of Manchester, ALICE THOMPSON, Department of Mathematics, Imperial College London, ANDREW HAZEL, Manchester Centre for Nonlinear Dynamics and School of Mathematics, University of Manchester, ANNE JUEL, Manchester Centre for Nonlinear Dynamics and School of Physics and Astronomy, University of Manchester --- We probe the sensitivity of Saffman–Taylor fingers to small controlled variations in channel depth by investigating the effect of a centred, rectangular occlusion on finger propagation in a Hele-Shaw channel. This geometry supports symmetric, asymmetric and oscillatory propagation states. A previously developed depth-averaged model is in quantitative agreement with laboratory experiments once the aspect ratio (width/height) of the tube's cross-section reaches a value of 40. We find that the multiplicity of solutions at a finite occlusion height arises through interactions of the single stable and multiple unstable solutions already present in the absence of the occlusion: the classic Saffman–Taylor viscous fingering problem. The sequence of interactions that occurs with increasing occlusion height is invariant for all aspect ratios investigated, but the occlusion height required for each interaction decreases with increasing aspect ratio. Thus, the system becomes more sensitive as the aspect ratio increases, in the sense that multiple solutions are provoked for smaller relative depth changes. We estimate that the required depth-changes become of the same order as the typical roughness of the experimental system (1 micron) for aspect ratios beyond 155.
11:14AM H18.00004 Influence of the contact line velocity on the finger formation of the liquid film expanding on an inclined plate. MASATOSHI YAMASHITA, MASATO NISHIKAWA, TAKAIRO ITO, YOSHIYUKI TSUJI, Nagoya University — When a liquid film flows down on an inclined solid surface, the contact line can be destabilized to finger shape. This phenomenon leads to the non-uniform height of the liquid surface or even to generation of dry patch, and then has a great effect on cooling of energy device and quality of coating. In previous studies, the final finger shapes have been discussed by relating the with capillary (Ca) number and the wetting properties of the liquid for the solid substrate, i.e. the contact angle. However, in the experimental studies, little attention has been paid on the difference between the static contact angle and the dynamic one, the latter which is actually observed when the finger is developing. In this study, we performed three-dimensional measurement of surface geometry of the liquid film to clarify how the dynamic contact angles and the Ca number influence the finger shape by optical method. We observed two different finger shapes depending on the volumes of the working fluid., and verified that the finger shapes depend on the contact angle scaled by Ca number. We found that the local dynamic contact angle and the contact line velocity on the trough part of the wavy contact line can be highly related with the final finger shape.

11:27AM H18.00005 The Role of Chemical Reactions in Fluid Instabilities: Step-Growth Polymerization¹, SIMONE STEWART, DANIELA MARIN, PATRICK BUNTON, Department of Physics, William Jewell College, Liberty, MÔ U.S.A. — Fingering is a fluid instability that occurs when a fluid of high mobility displaces a fluid of lower mobility. When the source of mobility difference is viscosity, viscous fingering occurs. Schlieren imaging is used to view viscous fingering during step-growth polymerization of various dithiol-di-acrylate systems in a Hele-Shaw cell. A dithiol is flowed into various diacylates of varying viscosity. Systems are characterized in terms of the Damkohler number (Da), which is the ratio of the chemical time scale to the hydrodynamic time scale. The reaction rate is tuned by varying the kind and amount of initiator and the flow rate is easily varied with a syringe pump. As a result of these variations, it is possible to gain some degree over control of the fingering that occurs. Results have shown the effects of a low flow rate on a low concentration monomer are comparable to the effects in high mobility differences has an effect on recovery and microfluidics. Mixing relies on the presence of concentration gradients. In this work, we determine the concentration field experimentally during the injection of a fluid into a circular Hele-Shaw cell to displace a more viscous fluid. We use a fluorescent tracer with the injected fluid to obtain high-resolution concentration fields, from which we determine the concentration gradients for different fluid injection rates and various viscosity ratios. Areas where gradients are present constitute the mixing zone, which can be characterized by its length and its thickness. We develop quantitative models of the dynamics of the interface length (previous work) and mixing zone thickness, and propose a scaling theory for the growth of the mixing zone and the overall impact of viscous fingering on mixing.

11:40AM H18.00006 Impact of Viscous Fingering on Fluid Mixing , JANE CHUI, MIT, PIETRO DE ANNA, Universite de Lausanne, RUBEN JUANES, MIT — Viscous fingering is a hydrodynamic instability that occurs when a less viscous fluid displaces a more viscous one. Instead of progressing as a uniform front, the less viscous fluid forms fingers that vary in size and shape to create complex patterns. Understanding how these patterns and their associated gradients evolve over time is of critical importance in characterizing the mixing of two fluids, which in turn is important to applications such as enhanced oil recovery and microfluidics. Mixing relies on the presence of concentration gradients. In this work, we determine the concentration field experimentally during the injection of a fluid into a circular Hele-Shaw cell to displace a more viscous fluid. We use a fluorescent tracer with the injected fluid to obtain high-resolution concentration fields, from which we determine the concentration gradients for different fluid injection rates and various viscosity ratios. Areas where gradients are present constitute the mixing zone, which can be characterized by its length and its thickness. We develop quantitative models of the dynamics of the interface length (previous work) and mixing zone thickness, and propose a scaling theory for the growth of the mixing zone and the overall impact of viscous fingering on mixing.

11:53AM H18.00007 Influence of fluid dispersion on transient behaviors of miscible viscous fingering , TAPAN KUMAR HOTA, MANORANJAN MISHRA, Indian Institute of Technology Ropar — We study the influence of fluid dispersion on miscible viscous fingering (VF) in the framework of non-modal stability theory. Miscible VF has traditionally been investigated by quasi-steady approximation (QSSA), followed by normal mode analysis. However, the results of QSSA poorly predicting the transient behavior and often stability analysis are obscured by the unsteady base flow. The system has been studied by coupling the convective characteristic scales are used. The long time behavior of the response to external excitations and the initial conditions are examined by the structure of pseudospectra and the optimal growth function, respectively. Particular attention is paid to the transient behavior rather than the long time behavior of eigenmodes predicted by modal analysis. The results show that there can be substantial transient growth of perturbations when the fluid disperses at a very slow rate.

12:06PM H18.00008 Fingering patterns induced by precipitation reactions , A. DE WIT, F. HAUDIN, P. SHUKLA, F. BRAU, Universite libre de Bruxelles (ULB), Nonlinear Physical Chemistry Unit, 1050 Brussels, Belgium — When reactants of a precipitation reaction are injected in a given porous medium, a fingering instability deforming the precipitation front can occur due to a change in permeability along the flow. We study the related precipitation patterns by combined experimental and theoretical work. Experiments are performed in confined geometries i.e. so-called Hele-Shaw cells consisting in two horizontal transparent plates separated by a thin gap containing a solution of one reactant B. The solution of the other reactant A is injected radially in the cell through a small hole. Upon displacement, a precipitation reaction between reactants A and B produce a solid phase C in the miscible reactive zone. We show that a wealth of different precipitation patterns (including spirals, flowers or filaments) can be observed depending on the flow rate and relative concentration of the two reactants. We discuss the relative effect of viscous fingering and of the cohesive properties of the precipitate in shaping the patterns. From a theoretical point of view, nonlinear simulations of the problem give insight into the similarities and differences between viscous fingering and precipitation-driven fingering.

12:19PM H18.00009 Comet-shape deformation and transition to viscous fingering of a miscible circular blob in porous media , MANORANJAN MISHRA, Indian Institute of Technology Ropar, India, ANNE DE WIT, Universite Libre de Bruxelles, Belgium, SATYAJIT PRAMANIK, Indian Institute of Technology Ropar, India — We numerically show that a miscible circular blob of viscosity larger than the ambient fluid features three different instability patterns – viscous fingering (VF), lump- and comet-shape instabilities. For a given Péclet number larger than the critical value, VF is observed in a finite range of viscosity contrast between the two fluids. This is in strong contrast to the displacement of a finite slice of miscible fluid in porous media in which instability enhances as viscosity contrast increases. Outside the finite interval of critical viscosity contrast for VF, the circular blob features comet- and lump-shape instabilities. These new dynamics are attributed to competition among the diffusive, convective and viscous forces. Our findings can be very important to understand many physico-chemical dynamics, CO₂ sequestration, reactive dissolution of porous rocks, etc.
The displacement flow in a porous media is remarkably influenced by the solute concentration dissolved in the fluids. The equilibrium-dispersive model for the evolution of the solute is numerically investigated which is coupled to Darcy law. In this model the fluid viscosity depends upon the solute concentration with the solute undergoing a non-linear adsorption of Langmuir type. The non-linear adsorption results in the formation of shock layer, which is progressively developed at the interface of the two fluids. The simulation results reveal that as soon as there is viscosity contrast between the interplaying fluids, the steepened profile formed due to Langmuir adsorption speed up the instability phenomena. Thus for fluids having larger viscosity contrast the shock layer is never formed. However, for fluids having less viscosity gradient, the shock layer gets formed but it eventually vanishes at the onset of instability. Hence the viscous fingering instability and shock layer affect the occurrence of each other.

Monday, November 23, 2015 10:35AM - 12:45PM
Session H19 Vortex Dynamics: Dipoles, Pairs and Instabilities 207 - Gerardo Chavarria, Universidad Nacional Autonoma de Mexico

10:35AM H19.00001 The evolution of a dipole in a periodic forced flow

10:48AM H19.00002 On the stability of a solid-body-rotation flow in a finite-length pipe

11:01AM H19.00003 Evolution of Vortex Pairs Subject to the Crow Instability in Wall Effect

11:14AM H19.00004 Moment model for interacting dipoles in two-dimensional flows

11:27AM H19.00005 ABSTRACT WITHDRAWN

11:40AM H19.00006 Experimental investigation of the interaction of a vortex dipole with a deformable cantilevered plate
11:53AM H19.00007 A Quantitative Assessment of Asymmetric Vortex Interactions in Viscous Flow, PATRICK FOLZ, KEIKO NOMURA, University of California, San Diego — The interactions of two co-rotating vortices in viscous fluid are investigated using 2D numerical simulations, performed across a range of vortex strength ratios, \( \Lambda = \Gamma_1 / \Gamma_2 \), with differing initial shape and/or peak vorticity. In all cases, the interaction produces a single vortex which is quantitatively evaluated. In particular, the interaction produces a single vortex, which has been previously done for inviscid flow. The analysis monitors the vortex cores throughout the interaction and identifies the end of the interaction, at which time the existing vortex is assessed. Symmetric pairs produce a compound vortex with \( \varepsilon \) near the maximum value of 2. For asymmetric pairs, \( \varepsilon \) and the associated merging efficiency generally decrease with \( \Lambda \), although differing pairs with the same \( \Lambda \) may produce different outcomes. For significantly disparate vortices, one of the original vortices survives without enhancement, i.e., \( \varepsilon \sim 1 \). These observations are explained in terms of underlying physics. Comparisons are made with available experimental data.

12:06PM H19.00008 Transient wake and trajectory of free falling cones with various apex angles, YAQING JIN, ALI M. HAMED, LEONARDO P. CHAMORRO, Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — The early free-fall stages of cones with a density ratio 1.18 and apex angles of 30, 45, 60, and 90 were studied using a wireless 3-axis gyroscope and accelerometer to describe the cone 3D motions, while the induced flow in the near wake was captured using particle image velocimetry. The Reynolds number based on the cone diameter and the velocity at which the cone reaches the first local velocity maximum is found to set the limit between two distinctive states. Before this Re is reached the departure from the vertical path and cone rotations are insignificant, while relatively rapid growth is observed after this Re. Sequences of vertical velocity, swirling strength, LES-decomposed velocity, and pressure fields show the formation and growth of a large and initially symmetric recirculation bubble at the cone base and highlights the presence of a symmetric 3D vortex rollup dominating the near-wake in the early stages of the fall. Later, the shear layer at the edge of the wake manifests in the shedding of Kelvin-Helmholtz vortices that, due to the nature of the recirculation bubble, reorganize to constitute a part of the rollup. Later in the fall, the wake loses its symmetry and shows a large population of vortical structures leading to turbulence. The asymmetric wake leads to strong interactions between the flow field and the cone creating complex feedback loops.

12:19PM H19.00009 Late time vortex dynamics for a coherent structure interacting with fine-scale turbulence, ERIC STOUT, FAZLE HUSSAIN, Texas Tech University — The vortex dynamics of perturbations to a coherent vortex column with fine-scale turbulence induced axial flow are examined using direct numerical simulation. Turbulence forms into azimuthally oriented filaments, which naturally results in axial flow as the filaments self-advection. Axial flow (W) modifies vorticity generation in two ways: 1) the radial gradient of W causes radial perturbation vorticity to tilt into the axial direction; and 2) axial perturbation vorticity tilts mean azimuthal vorticity (the vortical equivalent of W) into the radial direction. Given the cyclic of radial and axial perturbation vorticity generation, with the concomitant generation of azimuthal vorticity by the column's mean strain, this provides a physical explanation for instability due to axial flow (i.e., instability of the Batchelor or \( q \)-vortex, where \( q \) is the ratio of peak azimuthal to peak axial velocities). Via this interpretation, the role of non-axisymmetric azimuthal modes in \( q \)-vortex instability is explained. Vorticity generation due to axial flow is explored using a simplified perturbation consisting of two, antiparallel helical vortex threads encircling a vortex column, which results in late time vorticity generation and energy production.

12:32PM H19.00010 Experimental investigation of boundary layer transition on rotating cones in axial flow in 0 and 35 degrees angle of attack, ALI KARGAR, KAMYAR MANSOUR, Amir Kabir University of Technology (Tehran Polytechnic) — In this paper, experimental results using hot wire anemometer and smoke visualization are presented. The results obtained from the hot wire anemometer for critical Reynolds number and transitional Reynolds number are compared with previous results. Excellent agreement is found for the transitional Reynolds number. The results for the transitional Reynolds number are also compared to previous linear stability results. The results from the smoke visualization clearly show the crossflow vortices which arise in the transition process from a laminar to a turbulent flow. A non-zero angle of attack is also considered. We compare our results by linear stability theory which was done by. We just emphasis. Also we compare visualization and hot wire anemometer results graphically. Our goal in this paper is to check reliability of using hot wire anemometer and smoke visualization in stability problem and check reliability of linear stability theory for this two cases and compare our results with some trusty experimental works.

Monday, November 23, 2015 10:35AM - 12:45PM –
Session H20 Turbulent Taylor-Couette and Mixing 208 - Bruno Eckhardt, Philipps-Universitaet Marburg

10:35AM H20.00001 Momentum transport in Taylor-Couette flow with vanishing curvature, BRUNO ECKHARDT, HANNES BRAUCKMANN, MATTHEW SALEWSKI, Philipps-Universitaet Marburg — We study the influence of system rotation on torque and on mean angular momentum profiles in turbulent Taylor-Couette flow for large cylinder radii in direct numerical simulations. In this limit, curvature effects that can cause a stabilization of the outer flow region become negligible. We find that the torque as a function of the system rotation shows two maxima at a shear Reynolds number of \( 2 \times 10^4 \). The broad torque maximum for a moderate system rotation is related to strong turbulent Taylor vortices. A model based on marginal stability of boundary layers reproduces this torque maximum. The comparison between our simulations and the model suggests that the second torque maximum at weak system rotation is caused by the transition to turbulent boundary layers.

10:48AM H20.00002 Exploring the phase space of multiple states in highly turbulent Taylor-Couette flow, ROELAND VAN DER VEEEN, SANDER HUISMAN, ON YU DUNG, HO LUN TANG, CHAO SUN, DETLEF LOHSE, University of Twente — It was recently found that multiple turbulent states exist for large Reynolds number (\( Re = 10^6 \)) Taylor-Couette flow in the regime of ultimate turbulence. Here we investigate how these turbulent states depend on the Reynolds number in the range of \( Re = 10^5 \) to \( 2 \times 10^6 \), by measuring global torque and local velocity while probing the phase space spanned by the rotation rate of the inner and outer cylinder. This sheds light on the question whether multiple states persist for Reynolds numbers beyond those currently reached. By mapping the flow structures for various rotation ratios in two Taylor-Couette setups with equal radius ratio but different aspect ratio, we furthermore investigate the influence of aspect ratio on the characteristics of the multiple states.
11:01AM H20.00003 Turbulence-driven mean flow generation within laboratory Taylor-Couette flow, M.J. BURIN, CSU San Marcos, H. JI, Princeton, G. TYNAN, UCSD, E. EDLUND, MIT, E. GILSON, K. CASPARY, PPPL, R. EZETA APARICIO, U. Twente, P. DANG, Princeton, M. MCNULTY, Rutgers — We report on a new experimental effort to study mean (or zonal) flow generation within a turbulent laboratory fluid with a wide-gap Taylor-Couette apparatus. Mean flows from externally-forced turbulence are observed, both with and without a linearly sloped end-cap, which when present enforces a radial variation of potential vorticity. We characterize the dependence of the mean flow on various experimental parameters, such as forcing strength and azimuthal mode number, and offer accounts of their dynamical origins.

11:14AM H20.00004 Enhanced transport by grooved walls in turbulent Taylor-Couette flow, XIAOJUE ZHU, RODOLFO OSTILLA-MONICO, ROBERTO VERZICCO, DETLEF LOHSE, Physics of Fluids Group, University of Twente — We present direct numerical simulations of Taylor-Couette flow with grooved walls at a fixed radius ratio $\eta = r_i/r_o = 0.714$ with inner cylinder Reynolds number up to $Re_i = 3.70 \times 10^4$, corresponding to Taylor number up to $Ta = 2.15 \times 10^9$. The grooves are axisymmetric V-shaped obstacles attached to the wall with a tip angle of 90°. Results are compared with the smooth wall case in order to investigate the effects of grooves on Taylor-Couette flow. We focus on the effective scaling laws for the torque, flow structures, and boundary layers. It is found, when the groove height is smaller than the boundary layer thickness, the torque is the same as that of the smooth wall cases. With increasing $Ta$, the boundary layer thickness becomes smaller than the groove height. Plumes are ejected from tips of the grooves and a secondary circulation between the latter is formed. This is associated to a sharp increase of the torque and thus the effective scaling law for the torque vs. $Ta$ becomes much steeper. Further increasing $Ta$ does not result in an additional slope increase. Instead, the effective scaling law saturates to the “ultimate” regime effective exponents seen for smooth walls.

11:27AM H20.00005 The near-wall region of highly turbulent Taylor-Couette flow, RODOLFO OSTILLA MONICO, Physics of Fluids, Twente University, Enschede, Netherlands, ROBERTO VERZICCO, DIll, UniversitäTor Vergata, Rome, Italy, DETLEF LOHSE, Physics of Fluids, Twente University, Enschede, Netherlands — Direct numerical simulations of the Taylor-Couette (TC) flow, the flow between two coaxial and independently rotating cylinders, have been performed. The study focuses on TC flow with mild curvature (small gap) with a radius ratio of $\eta = r_i/r_o = 0.909$, an aspect ratio of $\Gamma = l/d = 2\pi/3$, and a stationary outer cylinder. Three inner cylinder Reynolds of $1 \times 10^5, 2 \times 10^5$ and $3 \times 10^5$ were simulated, corresponding to frictional Reynolds numbers between $Re_f \approx 1400$ and $Re_f \approx 4000$. An additional case with a large gap, $\eta = 0.5$ and driving of $Re = 2 \times 10^4$ was also performed. Small-gap TC was found to be dominated by spatially-fixed large-scale structures, known as Taylor rolls (TRs). TRs are attached to the boundary layer, and are active, i.e. they transport angular velocity through Reynolds stresses. For small-gap TC, evidence for the existence of logarithmic velocity fluctuations, and of an overlap layer, in which the velocity fluctuations collapse in outer units, was found. Profiles consistent with a logarithmic dependence were also found for the angular velocity in large-gap TC, albeit in a very reduced range of scales.

11:40AM H20.00006 Angular statistics of fluid particle trajectories in confined two-dimensional turbulence, BENJAMIN KADDOCH, IUSTI, Aix-Marseille University, Marseille, France, WOUTER BOS, LMFA-CNRS, Ecole Centrale de Lyon, Ecully, France, KAI SCHNEIDER, MPI2-CNRS & CMI, Aix-Marseille University, Marseille, France — The directional change of fluid particles can be characterized by the angle between subsequent particle displacement increments evaluated as a function of the time lag [1]. At small values of the time-increment the so-defined angle is proportional to the curvature of the trajectory. At large values this coarse-grained curvature should be affected by the presence of solid no-slip walls around the flow domain. In [2] we applied these statistics to three-dimensional isotropic turbulence, here we compare homogeneous and confined two-dimensional turbulent flows. We show that at long times the probability density function of the angles carries the signature of the confining domain if finite size effects are present. At short times, the PDF of the cosine of the angle is given by a power law with a well defined exponent, reminiscent of the close to Batchelor’s theory. For high Schmidt number scalars, it predicts a $\kappa^{-1}$ scaling of energy spectrum in the viscous-convective region. When using the mean gradient forcing technique, the energy spectrum agrees well with Batchelor’s theory. When using the linear scalar forcing, on the other hand, the energy spectrum does not follow the $\kappa^{-1}$ scaling. This difference can be explained by considering Yaglom’s equation for the scalar structure functions and are due to the form of the forcing source term. These results give a hint to the disagreement between theoretical predictions and experimental data in turbulent mixing literature.

11:53AM H20.00007 Impact ofScalar Force Simulation Techniques on High Schmidt Number Turbulent Mixing, K. JEFF RAH, GUILLAUME BLANQUART, Caltech — Numerous Direct Numerical Simulations of turbulent scalar mixing have been performed with a forcing technique to prevent the turbulent mixing from decaying. In this work, two scalar forcing techniques, namely the mean gradient forcing and the linear scalar forcing, are compared to assess the validity of Batchelor’s theory. For high Schmidt number scalars, it predicts a $\kappa^{-1}$ scaling of energy spectrum in the viscous-convective region. When using the mean gradient forcing technique, the energy spectrum agrees well with Batchelor’s theory. When using the linear scalar forcing, on the other hand, the energy spectrum does not follow the $\kappa^{-1}$ scaling. This difference can be explained by considering Yaglom’s equation for the scalar structure functions and are due to the form of the forcing source term. These results give a hint to the disagreement between theoretical predictions and experimental data in turbulent mixing literature.

12:06PM H20.00008 Turbulent velocity and concentration measurements in a multi-inlet vortex nanoprecipitation reactor, MICHAEL G. OLSEN, ZHENPING LIU, EMMANUEL HITIMANNA, Iowa State University, Mechanical Engineering, JAMES C. HILL, RODNEY O. FOX, Iowa State University, Chemical and Biological Engineering — Turbulent flow characteristics in a multi-inlet vortex reactor (MIVR) are of interest due to this reactor’s importance in nanoprecipitation applications. In the presented work, velocity and passive scalar concentration fields in a macroscale MIVR have been investigated by using stereoscopic particle image velocimetry (SPIV) and planar laser induced fluorescence (PLIF). The measurements are focused near the reactor center where the turbulent mixing occurs. The investigated Reynolds numbers based on the bulk velocity and diameter at the reactor outlet range from 16800 to 42000, resulting in a complex turbulent swirling flow within the reactor. The mean velocity field can be divided into free vortex region and forced vortex region from the wall to the reactor center. Back flow appears due to the low pressure in the forced vortex region. Most of turbulent fluctuation and mixing also occur in the forced vortex region while two point spatial correlations show turbulent eddies undergo shear stretching in the free vortex region. The flow is found to be unsteady with a wandering vortex center. As expected, a mixing performance is found to improve with increasing Reynolds number.
12:19PM H20.00009 Statistics of High Atwood Number Turbulent Mixing Layers
JON BALITZER, DANIEL LIVESCU, Los Alamos National Laboratory — The statistical properties of incompressible shear-driven planar mixing layers between two miscible streams of fluids with different densities are investigated by means of Direct Numerical Simulations. The simulations begin from a thin interface perturbed by a thin broadband random disturbance, and the mixing layers are allowed to develop to self-similar states. The temporal simulations are performed in unprecedented domain sizes, with grid sizes up to $6144 \times 2048 \times 1536$, which allows turbulent structures to grow and merge naturally. This allows the flow to reach states far-removed from the initial disturbances, thereby enabling high-quality statistics to be obtained for higher moments, pdfs, and other quantities critical to developing closure models. A wide range of Atwood numbers are explored, ranging from nearly constant density to $At=0.87$. The consequences of increasing the density contrast are investigated for global quantities, such as growth rates, and asymmetries that form in statistical profiles. Additional simulations in smaller domains are performed to study the effects of domain size.

ADAM J. WACHTOR, JOZSEF BAKOSI, RAYMOND RISTORCELLI, Los Alamos National Laboratory — Among the parameters characterizing mixing by variable density turbulence of fluids involving density variations of a factor of 5 to 10 are the Atwood, Froude, Schmidt, and Reynolds numbers. There is evidence that the amount of each fluid present when the two pure fluids mix, as described by the probability density function of the mass or molar (volume) fraction, also strongly affects the mixing process. To investigate this phenomena, implicit large-eddy simulations (ILES) are performed for binary fluid mixtures in statistically homogeneous environments under constant acceleration. These coarse grained simulations are used as data for theory validation and mix model development. ILES has been demonstrated to accurately capture the mixing behavior of a passive scalar field through stirring and advection by a turbulent velocity field. The present work advances that research and studies the extent to which an under-resolved active scalar drives the subsequent fluid motion and determines the nature of the mixing process. Effects of initial distributions of the mass and molar (volume) fraction probability density function on the resulting variable density turbulence and mixing are investigated and compared to direct numerical simulations of the Johns Hopkins Turbulence Database.

---

**Session H21 Turbulence: Modeling I**

10:35AM H21.00001 Synthesizing non-Gaussian inhomogeneous turbulence using optimization techniques
YI LI, University of Sheffield — Synthetic turbulence is an important component of large eddy simulations, where it is used as the initial or inlet condition. Traditional synthetic models have not attempted to reproduce small scale dynamics even when it is important to do so. This problem was attacked recently by a Multi-Turnover Lagrangian Map (MTLM) model, which successfully reproduces a range of small-scale statistics in isotropic turbulence. In this talk, we introduce the constrained MTLM method (CMTLM), where optimization technique is used to generate inhomogeneous non-Gaussian MTLM synthetic fields. In CMTLM, the inhomogeneous statistics are set as the target, to be matched by the MTLM field. The MTLM field is found as the solution of an optimization problem with the random Gaussian input to MTLM as the control. We use several cases to show that the optimal MTLM field reproduces the inhomogeneous statistics while maintaining the realistic small scale statistics in many different flow conditions. The method thus proves to be a useful tool for large eddy simulations.

10:48AM H21.00002 Wall-resolved adaptive simulation with spatially-anisotropic wavelet-based refinement
GIULIANO DE STEFANO, University of Naples (ITALY), ERIC BROWN-DYMKOSKI, OLEG V. VASYLYEV, University of Colorado Boulder — In the wavelet-based adaptive multi-resolution approach to turbulence simulation, the separation between resolved energetic structures and unresolved flow is achieved through wavelet threshold filtering. Depending on the thresholding level, the effect of residual motions can be either neglected or modeled, leading to wavelet-based adaptive DNS or LES. Due to the ability to identify and efficiently represent energetic dynamically important flow structures, these methods have been proven reliable and effective for the computational modeling of wall-bounded turbulence. The wall-resolved adaptive approach however necessitates the use of high spatial resolution in the wall region, which practically limits the application to moderate Reynolds numbers. In order to address this issue, a new method that makes use of a spatially-anisotropic adaptive wavelet transform on curvilinear grids is introduced. In contrast to all known adaptive wavelet-based approaches that suffer from the “curse of anisotropy,” i.e., isotropic wavelet refinement and inability to have spatially varying aspect ratio of the mesh elements, this approach utilizes spatially-anisotropic wavelet-based refinement. The method is tested for the turbulent flow past a rectangular cylinder at moderately high Reynolds number.

---

**Monday, November 23, 2015 10:35AM - 12:45PM**

10:35AM H21.00001 Synthesizing non-Gaussian inhomogeneous turbulence using optimization techniques
YI LI, University of Sheffield — Synthetic turbulence is an important component of large eddy simulations, where it is used as the initial or inlet condition. Traditional synthetic models have not attempted to reproduce small scale dynamics even when it is important to do so. This problem was attacked recently by a Multi-Turnover Lagrangian Map (MTLM) model, which successfully reproduces a range of small-scale statistics in isotropic turbulence. In this talk, we introduce the constrained MTLM method (CMTLM), where optimization technique is used to generate inhomogeneous non-Gaussian MTLM synthetic fields. In CMTLM, the inhomogeneous statistics are set as the target, to be matched by the MTLM field. The MTLM field is found as the solution of an optimization problem with the random Gaussian input to MTLM as the control. We use several cases to show that the optimal MTLM field reproduces the inhomogeneous statistics while maintaining the realistic small scale statistics in many different flow conditions. The method thus proves to be a useful tool for large eddy simulations.

10:48AM H21.00002 Wall-resolved adaptive simulation with spatially-anisotropic wavelet-based refinement
GIULIANO DE STEFANO, University of Naples (ITALY), ERIC BROWN-DYMKOSKI, OLEG V. VASYLYEV, University of Colorado Boulder — In the wavelet-based adaptive multi-resolution approach to turbulence simulation, the separation between resolved energetic structures and unresolved flow is achieved through wavelet threshold filtering. Depending on the thresholding level, the effect of residual motions can be either neglected or modeled, leading to wavelet-based adaptive DNS or LES. Due to the ability to identify and efficiently represent energetic dynamically important flow structures, these methods have been proven reliable and effective for the computational modeling of wall-bounded turbulence. The wall-resolved adaptive approach however necessitates the use of high spatial resolution in the wall region, which practically limits the application to moderate Reynolds numbers. In order to address this issue, a new method that makes use of a spatially-anisotropic adaptive wavelet transform on curvilinear grids is introduced. In contrast to all known adaptive wavelet-based approaches that suffer from the “curse of anisotropy,” i.e., isotropic wavelet refinement and inability to have spatially varying aspect ratio of the mesh elements, this approach utilizes spatially-anisotropic wavelet-based refinement. The method is tested for the turbulent flow past a rectangular cylinder at moderately high Reynolds number.

---

This work was supported by NSF under grant No. CBET-1236505
11:14AM H21.00004 Towards Sparse-Direct Interaction Perturbation (SDIP) for Variable-Density Flow1, DAVID PETTY, CARLOS PANTANO, University of Illinois Urbana Champaign — A numerical method has been developed to solve the set of integro-differential equations which result from applying the Sparse Direct-Interaction Perturbation (SDIP) technique to the low-speed, variable-density Navier-Stokes equations. This type of turbulence is at the heart of mixing and combustion applications. SDIP is a second-order moment closure theory that has particular relevance to the modeling of fluid turbulence. The strongly nonlinear numerical problem has been formulated as a system of equations using finite differences in time decorrelation, interpolation, variable-order quadratures, and mesh adaptation. The solution to this system has been made practicable by the construction of the full Jacobian of the numerical model using the Automatic Differentiation by OverLoading in C++ (ADOL-C) library. Special coordinate transformations were found to be essential for robust calculations of integrals that are not absolutely convergent; cancellations of singularities must be treated accurately. Progress toward the determination of the turbulence kinetic energy spectrum and velocity-scaler cospctra of the low-speed, variable-density Navier-Stokes equations derived from the SDIP solver will be discussed.

1The authors would like to thank the grant support for this research provided by the Air Force Office of Scientific Research, and the Department of Energy

11:27AM H21.00005 Spectral models of strongly inhomogeneous turbulence, ANDREW BRAGG, SUSAN KURIEN, Los Alamos Natl Lab, TIMOTHY CLARK, University of New Mexico — We compare results from a spectral model for inhomogeneous turbulence with DNS, FVM, LBM and LES. The model shows that the SFML (Tordella 2008) is not well-suited for strongly inhomogeneous fields.

11:40AM H21.00006 Asymptotic stability of spectral-based PDF modeling for homogeneous turbulent flows, ALEJANDRO CAMPOS, Stanford University, KARTHIK DURAI-SAMY, University of Michigan Ann Arbor, GIANLUCA IACCARINO, Stanford University — Engineering models of turbulence, based on one-point statistics, neglect spectral information inherent in a turbulence field. It is well known, however, that the evolution of turbulence is dictated by a complex interplay between the spectral model of velocity, for instance, and the transport equations. This study explores the development of a high-order PDF model for non-equilibrium inhomogeneous turbulent flows.

11:53AM H21.00007 Adaptive variable-fidelity wavelet-based eddy-capturing approaches for compressible turbulence1, ERIC BROWN-DYMKSOSKI, OLEG V. VASILYEV, University of Colorado Boulder — Multiresolution wavelet methods have been developed for efficient simulation of compressible turbulence. They rely upon a filter to identify dynamically important coherent flow structures and adapt the mesh to resolve them. The filter threshold parameter, which can be specified globally or locally, allows for a continuous tradeoff between computational cost and fidelity, ranging seamlessly between DNS and adaptive LES. There are two main approaches to specifying the adaptive threshold parameter. It can be imposed as a numerical error bound, or alternatively, derived from real-time flow phenomena to ensure correct simulation of desired turbulent physics. As LES relies on often imprecise model formulations that require a high-quality mesh, this variable-fidelity approach offers a further tool for improving simulation by targeting deficiencies and locally increasing the resolution. Simultaneous physical and numerical criteria, derived from compressible flow physics and the governing equations, are used to identify turbulent regions and evaluate the fidelity. Several benchmark cases are considered to demonstrate the ability to capture variable density and thermodynamic effects in compressible turbulence.

1This work was supported by NSF under grant No. CBET-1236505

12:06PM H21.00008 Full field inversion: A tool to diagnose and improve closure models1, ANAND PRATAP SINGH, KARTHIK DURAI-SAMY, Univ of Michigan - Ann Arbor — Existing single-point closure models of turbulence are inaccurate in complex flows. The errors inherent in these models cannot be rectified by modifying parameters in the model – it is rather the functional form of the model that is in question. In this work, full-field inversion is used to infer the functional form of modeling discrepancies. The inference process is driven by Bayesian inversion applied to data from Direct Numerical and Large Eddy simulations and experimental measurements. A physically-constrained approach is used to regularize the heavily ill-posed problem. It is to be noted that the full-field inversion involves extreme-scale optimization and Hessian computations. Efficient surrogate-enhanced adjoint techniques are employed to obtain the maximum aposteriori estimate and covariance of the inferred functions. The procedure is applied in a number of problems involving adverse and favorable pressure gradients and separation. The extracted information is used as part of a data-driven inversion/machine learning framework to improve closure models.

1This work was supported by the NASA Aeronautics Research Institute under the Leading Edge Aeronautics Research for NASA (LEARN) fund.

12:19PM H21.00009 Closure modeling using field inversion and machine learning, KARTHIK DURAI-SAMY, University of Michigan, Ann Arbor — The recent acceleration in computational power and measurement resolution has made possible the availability of extreme scale simulations and data sets. In this work, a modeling paradigm that seeks to comprehensively harness large scale data is introduced, with the aim of improving closure models. Full-field inversion (in contrast to parameter estimation) is used to obtain corrective, spatially distributed functional terms, offering a route to directly address model-form errors. Once the inference has been performed over a number of problems that are representative of the deficient physics in the closure model, machine learning techniques are used to reconstruct the model corrections in terms of variables that appear in the closure model. These machine-learned functional forms are then used to augment the closure model in predictive computations. The approach is demonstrated to be able to successfully reconstruct functional corrections and yield predictions with quantified uncertainties in a range of turbulent flows.
12:32PM H21.00010 Subgrid-scale modeling for flows with strong density variations
SIDHARTH GS, GRAHAM CANDLER, University of Minnesota — High-speed reacting flows exhibit strong spatio-temporal density variations that arise from heat release, compressibility and differences in composition. Strong density gradients involve baroclinic and dilatational sources of vorticity, thereby influencing the flow dynamics. The present work develops subgrid-scale models for this class of flows. We employ the formulation based on filtered velocity ($\tilde{u}$) as the resolved-scale velocity variable. This is because the conventional Favre-filtered velocity ($\tilde{u}$) is deficient in capturing the resolved-scale velocity dynamics and ignores the subgrid-scale interactions of pressure gradient with density. Furthermore, we investigate the contribution of subgrid-scale density fluctuations to the local subgrid-scale stress (and subgrid-scale scalar fluxes) via generation of small-scale velocity gradients. This effect is studied in the framework of the stretched-vortex subgrid-scale model. A posteriori performance of the proposed modifications is analysed on large eddy simulations of inert and reacting mixing layers.

Monday, November 23, 2015 10:35AM - 12:45PM  
Session H22 Turbulent Boundary Layers III  
210 - Michele Gualà, University of Minnesota

10:35AM H22.00001 Scale contributions to inertial layer momentum transport in turbulent boundary layers
JUAN CARLOS CUEVAS-BAUTISTA, University of New Hampshire, CALEB MRRIL-WINTER, University of Melbourne, JOSEPH KLEWICKI, CHRISTOPHER WHITE, GREGORY CHINI, University of New Hampshire — A weight of evidence indicates that the inertial region of the turbulent boundary layer is physically composed of large scale uniform momentum zones segregated by narrow fissures of highly vortical flow. Relative to momentum transport, this physical structure predominantly stems from the correlation between the wall-normal velocity fluctuations, $v$, and the spanwise vorticity fluctuations, $\omega_z$. The present research seeks to better understand how the relative scales of the $v$ and $\omega_z$ motions contribute to this advective transport mechanism under increasing scale separation associated with increasing Reynolds number. The broader aim is to advance an asymptotically reduced partial differential equation model of the turbulent boundary layer. Here we report on spectral analyses of high resolution, high Reynolds number measurements acquired using a four wire hotwire sensor. The focus is on quantifying the scales associated with the individual $v$ and $\omega_z$ signals, as well as the predominant scales associated with their correlation.

1 This research is supported by the National Science Foundation and the Australian Research Council.

10:48AM H22.00002 A measure of scale-dependent asymmetry in turbulent boundary layer flows
MICHELE GUALÀ, St. Anthony Falls Laboratory, Dep. Civil, Env. and Geo Eng., UMN, ARVIND SINGH, Department of Civil, Env. and Construction Eng., UCF — The distribution of scale-dependent, streamwise velocity increments is investigated in turbulent boundary layer flows at laboratory and atmospheric Reynolds number, using the SAFL wind tunnel (Singh et al. Phys. of Fluids 2014) and the SLTEST data (Metzger et al. Phil. Trans Royal Soc. A 2007). The third order moments of velocity increments, or asymmetry index $A_s(a, z)$, is computed for varying wall distance $z$ and scale separation $a$, where it was observed to leave a robust, distinct signature in the form of a hump, independent of Reynolds number and located across the inertial subrange. The hump is observed for $z/c < 5000$, with a tendency to increase in amplitude, and shift towards smaller timescales, as the wall is approached. Comparing the two datasets, the hump vertical location, obeying to inner wall scaling, is regarded to as a genuine feature of the canonical turbulent boundary layer. The magnitude cumulant analysis of the scale-dependent velocity increments indicates that intermittency is also enhanced in the same flow region. The combination of asymmetry and intermittency is inferred to point at non-local energy transfer across a range of scales and may thus be used to quantify interactions between structural types in boundary layer flows.

11:01AM H22.00003 Non-linear scale interactions in a forced turbulent boundary layer
SUBRAHMANYAM DUVVURI, BEVERLEY MCKEON, California Institute of Technology — A strong phase-organizing influence exerted by a single synthetic large-scale spatio-temporal mode on directly-coupled (through triadic interactions) small scales in a turbulent boundary layer forced by a spatially-impulsive dynamic wall-roughness patch was previously demonstrated by the authors (J. Fluid Mech. 2015, vol. 767, R4). The experimental set-up was later enhanced to allow for simultaneous forcing of multiple scales in the flow. Results and analysis are presented from a new set of novel experiments where two distinct large scales are forced in the flow by a dynamic wall-roughness patch. The internal non-linear forcing of two other scales with triadic consistency to the artificially forced large scales, corresponding to sum and difference in wavenumbers, is dominated by the latter. This allows for a forcing-response (input-output) type analysis of the two triadic scales, and naturally lends itself to a resolvent operator based model (e.g. McKeon & Sharma, J. Fluid Mech. 2010, vol. 658, pp. 336-382) of the governing Navier-Stokes equations.

1 The support of AFOSR (grant #FA 9550-12-1-0469, program manager D. Smith) is gratefully acknowledged.

11:14AM H22.00004 Spatio-temporal frequency responses of turbulent shear flows
ARMIN ZARE, MIHAILO JOVANOVIĆ, TRYPHON GEORGIOU, University of Minnesota - Minneapolis — Low-dimensional approximations of the Navier-Stokes equations are commonly used for the purpose of analysis and control of turbulent flows. In particular, stochastically-forced linearized models can capture statistical signatures observed in experiments and numerical simulations. In such models the dynamics of forcing play a critical role. It has been recently recognized that white-in-time forcing cannot explain the observed second-order statistics. In contrast, such statistics can be exactly matched with colored-in-time forcing. In order to account for partially-available second-order statistics of turbulent flows, we identify the dynamics of forcing using a convex-optimization procedure. We also provide a constructive method for designing linear filters that generate the colored-in-time forcing and show that our forcing models can be interpreted as perturbations to the original linearized flows, thereby influencing the flow dynamics. The present work develops subgrid-scale models for this class of flows. We employ the formulation based on filtered velocity ($\tilde{u}$) as the resolved-scale velocity variable. This is because the conventional Favre-filtered velocity ($\tilde{u}$) is deficient in capturing the resolved-scale velocity dynamics and ignores the subgrid-scale interactions of pressure gradient with density. Furthermore, we investigate the contribution of subgrid-scale density fluctuations to the local subgrid-scale stress (and subgrid-scale scalar fluxes) via generation of small-scale velocity gradients. This effect is studied in the framework of the stretched-vortex subgrid-scale model. A posteriori performance of the proposed modifications is analysed on large eddy simulations of inert and reacting mixing layers.

11:27AM H22.00005 Investigation of the temperature field in a turbulent boundary layer
CLAYTON BYERS, MARCUS HULTMARK, Princeton University — The scaling and evolution of a developing turbulent thermal boundary layer is investigated. By allowing the temperature differences in the fluid to remain small enough to treat temperature as a passive scalar, the analysis can be extended to any turbulent convection/diffusion problem. Mean temperature scaling is developed and analyzed by utilizing the Asymptotic Invariance Principle developed by George and Castillo (1997). Possible effects of the Reynolds and Prandtl number are discussed. The derived power law solution for the inner and outer scaling is then used to develop a heat transfer law for the wall heat flux, $q_w$. Data collection is performed with a newly developed MEMS sensor, allowing improved performance and reduced spatial and temporal filtering of the signal. Integration with a PIV system will allow direct measurements of the turbulent heat flux to investigate the extent of the overlap layer and validity of the proposed scaling laws. Temperature variance $\frac{1}{2}q_w^2$ will also be investigated, with a possible scaling proposed.
11:40AM H22.00006 Simultaneous wall-shear-stress and wide-field PIV measurements in a turbulent boundary layer, GUILLAUME GOMIT, University of Southampton, GREGOIRE FOURRIE, Universite internationale de Rabat, ROELAND DE KAT, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton — Simultaneous particle image velocimetry (PIV) and hot-film shear stress sensor measurements were performed to study the large-scale structures associated with shear stress events in a flat plate turbulent boundary layer at a high Reynolds number (Re = 4000). The PIV measurement was performed in a streamwise-wall normal plane using an array of six high resolution cameras (4 x 16MP and 2 x 29MP). The resulting field of view covers 88 (where d is the boundary layer thickness) in the streamwise direction and captures the entire boundary layer in the wall-normal direction. The spatial resolution of the measurement is approximately 70 wall units (1.8 mm) and sampled each 35 wall units (0.9 mm). In association with the PIV setup, a spanwise array of 10 skin-friction sensors (spanning one d) was used to capture the footprint of the large-scale structures. This combination of measurements allowed the analysis of the three-dimensional conditional structures in the boundary layer. Particularly, from conditional averages, the 3D organisation of the wall normal and streamwise velocity components (u, v, and w) and the Reynolds shear stress (−u′v′) related to a low and high shear stress events can be extracted.

11:53AM H22.00007 Characteristics of turbulent spots in transitional boundary layers, OLAF MARXEN, Univ of Surrey, TAMER ZAKI, Johns Hopkins University — The laminar-turbulent transition process in a flat-plate boundary layer beneath free-stream turbulence takes place through the inception and spreading of confined patches of turbulence in an otherwise laminar flow. These patches, also referred to as turbulent spots, result from a secondary instability of the Klebanoff streaks in the transitional region. The dynamics of turbulence in the spots are investigated by analyzing data sets obtained from direct numerical simulations. Conditionally-averaged and spot-ensemble-averaged statistics are evaluated and describe the flow in the intermittent transition zone. Both mean-flow and disturbance root mean square levels obtained from conditional averaging agree very well with results for fully turbulent flows, in particular near the wall and at high intermittency levels. At relatively low intermittency, the spatial inhomogeneity of turbulence within the spots is important, and is examined using ensemble averaging of turbulent patches that have comparable volume and a similar streamwise location.

12:06PM H22.00008 Characterization of coherent motions in cross flow via DNS, CAN LIU, Texas Tech University, GUILLERMO ARAYA, University of Puerto Rico - Mayaguez, STEFANO LEONARDI, University of Texas at Dallas, MURAT TUTKUN, Institute for Energy Technology, LUCIANO CASTILLO, Texas Tech University — Direct numerical simulations are performed at a friction Reynolds number of 394, based on the bulk velocity and half channel height. It is shown that small local blowing perturbations near the leading edge of the channel produced a secondary peak in turbulent production. This peak is attributed to the presence of a strong adverse pressure gradient that occurs in the outer part of the boundary layer. Furthermore, this secondary peak is produced by the energy enhancement of the presence of large-scale motions, which is a result of a shear layer located at about y+ ≈ 60. It has been found that the pressure fluctuation is important in the energy distribution of small scale motions in the inner region and large scale motions.

12:19PM H22.00009 Preponderance of hairpin vortices and their life cycles in the outer region of the canonical flat-plate boundary layer, JAMES WALLACE, University of Maryland, XIAOHUA WU, Royal Military College of Canada — While the dominance of hairpin vortices in the transitional and early turbulent regions of the zero-pressure-gradient, flat-plate boundary layer has been widely accepted, recent literature is divided on this issue at higher Reynolds numbers. Here we investigate the representative vortex structures in the outer region of the canonical boundary layer over the momentum thickness Reynolds number range of 1000 to 3000, using the DNS database of Wu, Moin and Hickey (Phys. Fluids 26, 091104). In the outer region of the boundary layer we observe that hairpin vortices comprise nearly fifty percent of all the vortical structures visualized with the swirling strength λ3. Furthermore, these hairpins remain identifiable while they are advected downstream over distances corresponding to increases of about 300 - 400 in momentum thickness Reynolds numbers. Therefore, over the Reynolds number range studied, approximately three generations of hairpins go through their life cycles. This demonstrates that many of these outer region hairpin vortices are generated in the fully-turbulent region, and thus are not the debris of the upstream transition to turbulence. Coherent structures in the near-wall region will also be discussed.

12:32PM H22.00010 Identifying Orr-like behaviour in full-scale turbulent wall-bounded flows, MIGUEL P. ENCINAR, JAVIER JIMENEZ, Technical University of Madrid — The presence of linear, transient phenomena in fully developed non-linear turbulence is studied in the sense of relating the growth of the intensity of wall-normal velocity perturbations with their inclination angle. This phenomenon is predicted by the Orr-Sommerfeld equation, and can explain the formation of the energy-containing scales when paired with the lift-up mechanism. This process has been previously identified in minimal channels in which structures are represented by individual Fourier modes, and is generalised here to extended ones that contain many individual structures at different scales and stages of development. We present a method based on wavelet projection that addresses both spatial and spectral locality. After filtering the flow with a given wavelet, a local optimum wavelength and wavefront inclination is computed at each point of the filtered flow, and used to trace the Orr-like behaviour. The relation of the measured perturbations with the rest of the flow properties is explored, showing good agreement with the predictions of the Orr mechanism.

Mon, Nov 23, 2015 10:35AM - 12:45PM — Session H23 Biofluids: Squirmers, Cilia and Pumping

10:35AM H23.00001 Locomotion of spherical squirmers in a viscoelastic fluid near a planar interface, SHAHRZAD YAZDI, ALI BORHAN, Pennsylvania State University — In an attempt to better understand the confined swimming of a microorganism in a viscoelastic fluid, we have analytically studied the time-averaged locomotion of a spherical squirm with a reciprocal surface motion near a plane interface in a polymeric solution. The results are presented through a phase-portrait in the swimming orientation and distance from the interface. The ratio of viscosities of the two phases adjacent to the plane interface is varied to examine motion near different types of boundaries. Our analysis shows that the near-wall attraction layer previously reported for a 2D squimer no longer exists for spherical pullers and pushers. However, the presence of a stable node attracts the swimmer to the vicinity of the wall.
10:48AM H23.00002 Deformable micro torque swimmer. TAKUJI ISHIKAWA, TOMOYUKI TANAKA, TOSHIHIRO OMORI, YOHSHUKE IMAI, Tohoku University — We investigated the deformation of a ciliate swimming freely in a fluid otherwise at rest. The cell body was modeled as a capsule with a hyper elastic membrane enclosing Newtonian fluid. Thrust forces due to the ciliary beat were modeled as torques distributed above the cell body. Effects of the membrane elasticity, the aspect ratio of cell’s reference shape and the density difference between the cell and the surrounding fluid were investigated. The results showed that the cell deformed like heart shape when the parameter (Ca) was sufficiently large, and the swimming velocity decreased as Ca was increased. The gravity effect on the membrane tension suggested that the upwards and downwards swimming velocities of Paramecium might be regulated by the calcium ion channels distributed locally around the anterior end. Moreover, the gravity induced deformation made a cell directed vertically downwards, which resulted in a positive geotaxis like behavior with physical origin. These results are important to understand physiology of ciliate’s biological responses to mechanical stimuli.

11:01AM H23.00003 Optimal computational methods for swimming and pumping with helical filaments at low Reynolds number. JAMES MARTINDALE, MEHDI JABBARZADEH, HENRY FU, Univ of Nevada - Reno — The flows induced by biological and artificial helical filaments are important to many possible applications including microscale swimming and pumping. Microscale helices can span a wide range of geometries, from thin bacterial flagella to thick helical bacteria cell bodies. While the proper choice of numerical method is critical for obtaining accurate results, there is little guidance about which method is optimal for a specified filament geometry. Using two physical scenarios - a swimmer with a head, and a pump - I establish guidelines for the choice of numerical method based on helical geometry. For a range of helical geometries that encompass most natural and artificial helices, I create benchmark results using a surface distribution of regularized Stokeslets, then evaluate the accuracy of resistive force theory, slender body theory, and a centerline distribution of regularized Stokeslets. Taking the computational cost of each method into account, I present the optimal choice of numerical method for each filament geometry as a guideline for future investigations involving filament-induced flows.

11:14AM H23.00004 Geometric pumping in autophoretic channels. SEBASTIEN MICHELIN, LadHyX - Ecole Polytechnique, THOMAS MONTENEGRO JOHNSON, GABRIELE DE CANIO, DAMTP - University of Cambridge, NICOLAS LOBATTO-DAUZIER, LadHyX - Ecole Polytechnique, ERIC LAUGA, DAMTP - University of Cambridge — Pumping at the microscale has important applications from biological fluid handling to lab-on-a-chip systems. It can be achieved either from a global (e.g. imposed pressure gradient) or local forcing (e.g. ciliary pumping). Phoretic slip flows generated from concentration or temperature gradients are examples of such local flow forcing. Autophoresis is of particular importance for the design of self-propelled particles achieving force- and torque-free locomotion by combining two essential surface properties: (i) an activity that modifies the solute content of the particle’s environment (e.g. catalytic reaction or solute release), and (ii) a mobility that generates a slip flow from the resulting local concentration gradients. Recent work showed that geometric asymmetry is sufficient for a chemically-homogeneous particle to self-propel. Here we extend this idea to micro-pumping in active channels whose walls possess both chemical activity and phoretic mobility. Using a combination of theoretical analysis and numerical simulations, we show that geometrically-asymmetric but chemically-homogeneous channels can generate pumping and analyze the resulting flow patterns.

11:27AM H23.00005 Flow Induced by Bacterial Carpets and Transport of Microscale Loads. AMY BUCHMANN, LISA FAUCI, Tulane University, KARIN LEIDERMANN, University of California Merced, EPA STRAWBRIDGE, James Madison University, LONGHUA ZHAO, Case Western Reserve University — Microfluidic devices carry very small volumes of liquid though channels and may be used to gain insight into many biological applications including drug delivery and development. In many microfluidic experiments, it would be useful to mix the fluid within the chamber. However, the traditional methods of mixing and pumping at large length scales do not work at small length scales. Experimental work has suggested that the flagella of bacteria may be used as motors in microfluidic devices by creating a bacterial carpet. Mathematical modeling can be used to investigate this idea and to quantify flow induced by bacterial carpets. We simulate flow induced by bacterial carpets using the method of regularized Stokeslets, and also examine the transport of vesicles of finite size by arrays of rotating flagella.

11:40AM H23.00006 Squirming through shear thinning fluids. CHARU DATT, The University of British Columbia, Vancouver, LAILAI ZHU, Ecole Polytechnique Federale de Lausanne, GWYNN J. ELFRING, The University of British Columbia, Vancouver, ON SHUN PAK, Santa Clara University, Santa Clara, California — Many microorganisms find themselves surrounded by fluids which are non-Newtonian in nature; human spermatozoa in female reproductive tract and motile bacteria in mucosa of animals are common examples. These biological fluids can display shear-thinning rheology whose effects on the locomotion of microorganisms remain largely unexplored. Here we study the self-propulsion of a squirmer in shear-thinning fluids described by the Carreau-Yasuda model. The squirmer undergoes surface distortions and utilizes apparent slip-velocities around its surface to swim through a fluid medium. In this talk, we will discuss how the nonlinear rheological properties of a shear-thinning fluid affect the propulsion of a swimmer compared with swimming in Newtonian fluids.

11:53AM H23.00007 Squirming propulsion in viscoelastic fluids. MARCO DE CORATO, Università degli studi di Napoli Federico II, Dipartimento di Ingegneria chimica dei Materiali e della Produzione industriale, FRANCESCO GRECO, Istituto di Ricerche sulla Combustione, IRC-CNR, PIER LUCA MAFFETTONE, Università degli studi di Napoli Federico II, Dipartimento di Ingegneria chimica dei Materiali e della Produzione industriale — The locomotion of organisms in Newtonian fluids at low-Reynolds numbers displays very different features from that at large Reynolds numbers; indeed, in this regime the viscous forces are dominant over the inertial ones and propulsion is possible only with non-time-reversible swimming strokes. In many situations of biological interest, however, small organisms are propelled themselves through non-Newtonian fluids such as mucus or biofilms, which display highly viscoelastic properties. Fluid viscoelasticity affects in a complex way both the micro-organisms’ swimming velocity and dissipated power, possibly affecting their collective behavior. In our work, we so-called “squirmers” to study the motion of spherical, chemically stimulated organisms in a viscoelastic fluid. We derive analytical formulas for the squirmer swimming velocity and dissipated power that show a complex interplay between the fluid constitutive behavior and the propulsion mechanism.

12:06PM H23.00008 Self-propulsive motion and deformation of a chemically-driven drop. NATSUHIKO YOSHINAGA, Tohoku Univ — Spontaneous motion has attracted lots of attention in the last decades in fluid dynamics for its potential application to biological problems such as cell motility. Recently, several model experiments showing spontaneous motion have been proposed and revealed the underlying mechanism of the motion. The systems in these works consist of relatively simple ingredients, but nevertheless their motion and deformation give us an impression as if they are alive. Importantly, the system breaks symmetry and chooses one direction of motion. We theoretically derive a set of nonlinear equations exhibiting a transition between stationary and motile states starting from advection-reaction-diffusion equation driven away from an equilibrium state due to chemical reactions. A particular focus is on how hydrodynamic flow destabilizes an isotropic distribution of a concentration field. We also discuss a shape of the droplet. Due to self-propulsive motion and flow around the droplet, a spherical shape becomes unstable and it elongates perpendicular to the direction of motion. This fact would imply that the self-propulsion driven by chemical reaction is characterized as a pusher in terms of a flow field. We shall also show numerical results using the phase field model.
manipulating fluids at the microscale. In biological and clinical contexts, and (b), engineered ciliated systems exploiting a variety of design parameters could provide novel ways of transport phenomena that cannot be reduced to a single parameter. Here we present two case studies. In one system, the ciliated surface creates two distinct flow regimes for first trapping and then sheltering potential symbiont bacteria for further biochemical screening. In the other system, chronic disease induces a misalignment of ciliary beat, leading to a pathological transition from uniform mucus transport to a pattern of stagnation and circulation. These studies suggest that (a), we need to develop a wider range of metrics for describing ciliary transport in biological and clinical contexts, and (b), engineered ciliated systems exploiting a variety of design parameters could provide novel ways of manipulating fluids at the microscale.

Monday, November 23, 2015 10:35AM - 12:45PM — Session H24 Biofluids: Cardiovascular Fluid Dynamics I

10:35AM H24.00001 Secondary flow structure in a model curved artery: 3D morphology and circulation budget analysis 1 — KARTIK V BULUSU, MICHAEL W PLESNIAK, George Washington University — In this study, we examined the rate of change of circulation within control regions encompassing the large-scale vortical structures associated with secondary flows, i.e., deformed Dean-, Lyne- and Wall-type (D-L-W) vortices at planar cross-sections in a 180° curved artery model (curvature ratio, 1/7). Magnetic resonance velocimetry (MRV) and particle image velocimetry (PIV) experiments were performed independently, under the same physiological inflow conditions (Womersley number, 4.2) and using Newtonian blood-analog fluids. The MRV-technique performed at Stanford University produced phase-averaged, three-dimensional velocity fields. Secondary flow field comparisons of MRV-based data were made at various phase angles and inflow conditions. We hypothesize that the persistence and decay of arterial secondary flow vortices is intrinsically related to the influence of the out-of-plane flow, tilting, in-plane convection and diffusion-related factors within the control regions. Evaluation of these factors will elucidate secondary flow structures in arterial hemodynamics.

1Supported by the National Science Foundation under Grant Number CBET-0828903, and GW Center for Biomimetics and Bioinspired Engineering (COBRE). The MRV data were acquired at Stanford University in collaboration with Christopher Elkins and John Eaton.

10:48AM H24.00002 A Mixed Approach for Modeling Blood Flow in Brain Microcirculation 1 — SYLVIE LORTHOIS, MYRIAM PEYROUNETTE, YOHAN DAVIT, MICHEL QUINTARD, Institut de Mécanique des Fluides de Toulouse : UMR CNRS INP UPS 5502, GROUPE D'ETUDE SUR LES MILIEUX POREUX TEAM — Consistent with its distribution and exchange functions, the vascular system of the human brain cortex is a superposition of two components. At small-scale, a homogeneous and space-filling mesh-like capillary network. At large scale, quasi-fractal branched veins and arteries. From a modeling perspective, this is the superposition of: (a) a continuum model resulting from the homogenization of slow transport in the small-scale capillary network; and (b) a discrete network approach describing fast transport in the arteries and veins, which cannot be homogenized because of their fractal nature. This problematic is analogous to fast conducting wells embedded in a reservoir rock in petroleum engineering. An efficient method to reduce the computational cost is to use relatively large grid blocks for the continuum model. This makes it difficult to accurately couple both components. We solve this issue by adapting the well model concept used in petroleum engineering to brain specific 3D situations. We obtain a unique linear system describing the discrete network, the continuum and the well model. Results are presented for realistic arterial and venous geometries. The mixed approach is compared with full network models including various idealized capillary networks of known permeability.

1ERC BrainMicroFlow GA615102

11:01AM H24.00003 Automated Tuning for Parameter Identification in Multi-Scale Coronary Simulations — JUSTIN TRAN, DANIELE SCHIAVazzi, ABHAY RAMACHANDRA, Stanford University, ANDREW KAHN, University of California, San Diego, ALISON MARSден, Stanford University — Computational simulations of coronary flow can provide non-invasive information on hemodynamics that can aid in disease research. In this study, patient-specific geometries are constructed and combined with finite element flow simulations using the open source software SimVascular. Lumped parameter networks (LPN), consisting of circuit representations of hemodynamic behavior, can be used as coupled boundary conditions for the flow solver. The parameters of the LPN are tuned so the outputs match a patient’s clinical data. However, the parameters are usually manually tuned, which is time consuming and does not account for uncertainty in the measurements. We thus propose a Bayesian approach to parameter tuning that provides optimal parameter statistics through sampling from their posterior distribution and is particularly well suited for models characterized by a large number of parameters and scarce data. We also show that analysis of the local and global identifiability play an important role for dimensionality reduction in the estimation. We present the results of applying the proposed approach to a cohort of patients, and demonstrate the ability to match high priority targets. After identifying the LPN parameters for each patient, we demonstrate their use in 3D simulations.
11:14AM H24.00004 Validity of computational hemodynamics in human arteries based on 3D time-of-flight MR angiography and 2D electrocardiogram gated phase contrast images\textsuperscript{1}, HUIDAN WHITNEY YU, XI CHEN, ROU CHEN, Indiana University-Purdue University Indianapolis, ZHIQIANG WANG, Kent State University, YUHAN LIU, STEPHEN KRALIK, School of Medicine, Indiana University, YE ZHAO, Kent State University — In this work, we demonstrate the validity of 4-D patient-specific computational hemodynamics (PSCH) based on 3-D time-of-flight (TOF) MR angiography (MRA) and 2-D electrocardiogram (ECG) gated phase contrast (PC) images. The mesoscale lattice Boltzmann method (LBM) is employed to segment morphological arterial geometry from TOF MRA, to extract velocity profiles from ECG PC images, and to simulate fluid dynamics on a unified GPU accelerated computational platform. Two healthy volunteers are recruited to participate in the study. For each volunteer, a 3-D high resolution TOF MRA image and 10-2-D ECG gated PC images are acquired to provide the morphological geometry and the time-varying flow velocity profiles for necessary inputs of the PSCH. Validation results will be presented through comparisons of LBM vs. 4D Flow Software for flow rates and LBM simulation vs. MRA measurement for blood flow velocity maps.

\textsuperscript{1}Indiana University Health (IUH) Values Fund

11:27AM H24.00005 Sharp Interface Methods for Cardiac Fluid-Solid Interaction . EBRAHIM M. KOLAHDOUZ\textsuperscript{2}, BENJAMIN L. VADALA-ROTH\textsuperscript{2}, AMNEET P. S BHALLA\textsuperscript{3}, BOYCE E. GRIFFITH \textsuperscript{4}, University of North Carolina at Chapel Hill — Fluid-solid systems are common in scientific and engineering applications. The immersed boundary (IB) method is a general approach to simulating fluid-structure interaction (FSI) in such systems, but a difficulty of the IB formulation of these problems is that the pressure and viscous stress are generally discontinuous at fluid-solid interfaces. The immersed interface (II) method is an IB-like approach to FSI that exactly imposes stress jump conditions, but this method has largely been limited to FSI problems involving thin elastic boundaries. We present extensions of the IB method that sharply resolve stress discontinuities at fluid-solid interfaces that can be viewed as extensions of the immersed interface method to non-interfacial (codimension-0) solid bodies, and the application of these methods to cardiovascular FSI, including the dynamics of the cardiac valves.

\textsuperscript{1}Department of Mathematics
\textsuperscript{2}Department of Mathematics
\textsuperscript{3}Department of Mathematics
\textsuperscript{4}Departments of Mathematics and Biomedical Engineering

11:40AM H24.00006 ABSTRACT WITHDRAWN —

11:53AM H24.00007 Accuracy and Robustness Improvements of Echocardiographic Particle Image Velocimetry for Routine Clinical Cardiac Evaluation . BRETT MEYERS, PAVLOS VLACHOS, Purdue Univ, JOHN CHARONKO, Los Almos National Labs, MATTHEW GIARRA, Virginia Tech, CRAIG GOERGEN, Purdue Univ — Echo Particle Image Velocimetry (echoPIV) is a recent development in flow visualization that provides improved spatial resolution with high temporal resolution in cardiac flow measurement. Despite increased interest a limited number of published echoPIV studies are clinical, demonstrating that the method is not broadly accepted within the medical community. This is due to the fact that use of contrast agents are typically reserved for subjects whose initial evaluation produced very low quality recordings. Thus high background noise and low contrast levels characterize most scans, which hinders echoPIV from producing accurate measurements. To achieve clinical acceptance it is necessary to develop processing strategies that improve accuracy and robustness. We hypothesize that using a short-time moving window ensemble (MWE) correlation can improve echoPIV flow measurements on low image quality clinical scans. To explore the potential of the short-time MWE correlation, evaluation of artificial ultrasound images was performed. Subsequently, echoPIV of patients with diastolic dysfunction was evaluated. Qualitative and quantitative comparisons between echoPIV measurements and Color M-mode scans were carried out to assess the improvements delivered by the proposed methodology.

12:06PM H24.00008 Strategies for Pile-up and Over-refinement to improve performance of the Surrogate Management Framework in cardiovascular flow optimization\textsuperscript{1}. AEKAANSH VERMA, ALISON MARSDEN, Stanford University — Engineering optimization problems are often limited by the cost of function evaluations. Furthermore, calculation of gradients in such problems can be expensive or even infeasible. Derivative free optimization methods such as variants of the Surrogate Management Framework (SMF) are suitable for such problems, and offer a well-established convergence theory. The SMF is comprised of a search step that is accelerated by a surrogate-based global search, typically using Kriging. Traditionally, Kriging-based SMF develop problems during the course of the optimization which affect both global and local search performance. We propose strategies to alleviate two such issues - pile-up of evaluations in a certain region in parameter space and improper refinement of the local search grid. We quantify the performance of these strategies on analytical test cases and discuss the mechanisms of improvement. Finally, we apply these strategies to some illustrative problems in cardiovascular blood flow simulations and growth and remodeling.

\textsuperscript{1}This work was supported by an NSF CAREER award (OCI1150184) and a Burroughs Wellcome Fund Career Award at Scientific Interface.

12:19PM H24.00009 4D-Flow validation, numerical and experimental framework . KURT SANSON, HAINING LIU, GADOR CANTON, ALBERTO ALISEDA, CHUN YUAN, University of Washington — This work presents a group of assessment metrics of new 4D MRI flow sequences, an imaging modality that allows for visualization of three-dimensional pulsatile flow in the cardiovascular anatomy through time-resolved three-dimensional blood velocity measurements from cardiac-cycle synchronized MRI acquisition. This is a promising tool for clinical assessment but lacks a robust validation framework. First, 4D-MRI flow in a subject’s stenotic carotid bifurcation is compared with a patient-specific CFD model using two different boundary condition methods. Second, Particle Image Velocimetry in a patient-specific phantom is used as a benchmark to compare the 4D-MRI in vivo measurements and CFD simulations under the same conditions. Comparison of estimated and measurable flow parameters such as wall shear stress, fluctuating velocity rms, Lagrangian particle residence time, will be discussed, with justification for their biomechanics relevance and the insights they can provide on the pathophysiology of arterial disease: atherosclerosis and intimal hyperplasia. Lastly, the framework is applied to a new sequence to provide a quantitative assessment. A parametric analysis on the carotid bifurcation pulsatile flow conditions will be presented and an accuracy assessment provided.
varying particle sizes to gauge the importance of time-dependence on overall fluid transport and mixing. We then analyze individual particle paths and their displacements in a region around the swimmer. Of particular interest are particles near the dependence of their swimming. We model the organism by a time-dependent dumbbell consisting of a solid body and a regularized Stokeslet, which swims by rapidly beating two frontal flagella. Previous studies of transport by microswimmers have neglected the ubiquitous time-dependence. Jean-Luc Thiffeault, University of Wisconsin - Madison — We study the drift caused by the microscopic algae Chlamydomonas reinhardtii, a motile biflagellate that can swim towards light for its photosynthetic requirements, a behavior referred to as phototaxis. The cell responds upon light stimulation through its rudimentary eye – the eyespot – by changing the beating amplitude of its two flagella synchronously – a process called the photoreversal. All this occurs in a coordinated fashion as Chlamydomonas spins about its body axis while swimming, thus experiencing oscillating intensities of light. We use high-speed video microscopy to measure the flagellar dynamics of the photoresponse on immobilized cells and interpret the results with a mathematical model of adaptation similar to that used previously for Volvox. These results are incorporated into a model of phototactic steering to yield trajectories that are compared to those obtained by three-dimensional tracking. Implications of these results for the evolution of multicellularity in the Volvocales are discussed.

10:48AM H25.00002 Synchronization of Eukaryotic Flagella with an Imposed Periodic Flow. Greta Quaranta, Marie-Eve Aubin-Tam, Daniel Tam, TU Delft — The eukaryotic cilia and flagella are subcellular structures able to beat in synchrony for long periods of time. Recent studies have characterized the dynamics of flagellar locomotion and have focused on the physical mechanisms driving synchronous beating and especially on the importance of hydrodynamic interactions. We explored the possibility to control the beating of the two flagella of a single C. reinhardtii cell by imposing an external periodic hydrodynamic force. We do so by generating an oscillatory background flow around a single cell. Our study shows that flagellar beating can be phase locked to an external hydrodynamic forcing of non-biological origin and the synchronization transition is well represented by a low-order stochastic model. Remarkably, the hydrodynamic forces needed to synchronize the flagella and the background flow are considerably larger than the forces typically experienced in physiological conditions. Our results suggest that the importance of hydrodynamics in flagellar synchronization may be limited.

11:01AM H25.00003 Quiet swimming at low Reynolds number. Anders Andersen, Navish Wadhwa, Department of Physics and Centre for Ocean Life, Technical University of Denmark, Thomas Kiorboe, National Institute for Aquatic Resources and Centre for Ocean Life, Technical University of Denmark — Plankonic organisms that inhabit the water masses of the oceans are faced with a dilemma: They need to swim to find food and mates, but by swimming they inevitably create flow disturbances that attract predators. We discuss that planktonic swimmers can reduce the flow disturbances due to their swimming, simply by appropriately arranging their propulsion apparatus. Motivated by recent experiments, we demonstrate that a three-Stokeslet model of a breast stroke swimmer is an example of a quiet swimmer. We show that the flow disturbances around the organism in both the near field and the far field are small in comparison with simple pullers and pushers, and we find that the far field power laws are valid surprisingly close to the organism. Breast stroke swimming may thus be advantageous, and this might explain why it is very common in the world of the plankton.

11:14AM H25.00004 Feeding and swimming of flagellates. Julia Doelger, Department of Physics and Centre for Ocean Life, Technical University of Denmark, Lasse Tor Nielsen, Thomas Kiorboe, National Institute for Aquatic Resources and Centre for Ocean Life, Technical University of Denmark, Tommas Bohr, Anders Andersen, Department of Physics and Centre for Ocean Life, Technical University of Denmark — Hydrodynamics plays a dominant role for small planktonic flagellates and shapes their survival strategies. The high diversity of beat patterns and arrangements of appendages indicates different strategies balancing the trade-offs between the general goals, i.e., energy-efficient swimming, feeding, and predator avoidance. One type of flagellated algae that we observe, are haptophytes, which possess two flagella for flow creation and one so-called haptonema, a long, rigid structure fixed on the cell body, which is used for prey capture. We present videos and flow fields obtained using velocimetry methods around freely swimming haptophytes and other flagellates, which we compare to analytical results obtained from point force models. The observed and modelled flows are used to analyse how different morphologies and beat patterns relate to different feeding or swimming strategies, such as the capture mechanism in haptophytes.

1The Centre for Ocean Life is a VKR center of excellence supported by the Villum foundation.

11:27AM H25.00005 Fluid transport by an unsteady microswimmer. Peter Mueller, Jean-Luc Thiffeault, University of Wisconsin - Madison — We study the drift caused by the microscopic algae Chlamydomonas reinhardtii, which swims by rapidly beating two frontal flagella. Previous studies of transport by microswimmers have neglected the ubiquitous time-dependence of their swimming. We model the organism by a time-dependent dumbbell consisting of a solid body and a regularized Stokeslet. We then analyze individual particle paths and their displacements in a region around the swimmer. Of particular interest are particles near the swimmer, which have complex trajectories due to the unsteady model. Particles directly in front of the swimmer contribute large, but rare, displacements. We use this to determine the tails of the distribution of particle displacements. Finally we compare the effective diffusivity of varying particle sizes to gauge the importance of time-dependence on overall fluid transport and mixing.

1Supported by NSF grant DMS-1109315
11:40AM H25.00006 Propulsion of micro-structures in Oscillatory Stokes Flow. J Khoo, YANGyang Huang, Univ of Southern California, Walter ZIMMERMAN, University of Bayreuth, Eva KANSO, Univ of Southern California — Drug delivery often necessitates specific site-targeting within the human body. The use of micro and/or nano devices swimming through the bloodstream provides an attractive mechanism for targeted drug targeting, however the design and practical implementation of such devices remain very challenging. Inspired by flapping wings, we construct a two-dimensional wedge-like device, consisting of two links connected by a linear torsional spring and released in an oscillatory Stokes flow. We vary the stiffness and rest angle of the linear spring and the oscillation amplitude and frequency of the background flow to explore the behavior of the device. We find that the device achieves a net displacement, or propulsion, in oscillatory flows even when no elastic energy is stored initially, thus breaking Purcell's scallop's theorem. More importantly, the vehicle tends to align with the background flow under perturbations. We conclude by commenting on how to control the parameters of the device, the fluid and the fluid to achieve desired behavior of the device. These findings may have significant implications on the design of micro devices in viscous fluids.

11:53AM H25.00007 Narrower bottlenecks could be more efficient for concentrating choanoflagellates. G. Miño, Massachusetts Institute of Technology, Cambridge, USA, J. Sparacino, FaMAF-UNC and IFEG-CONICET, Córdoba, Argentina, M.A.R. Koehl, N. King, University of California, Berkeley, USA, R. Stocker, Massachusetts Institute of Technology, Cambridge, USA, A.J. Banchio, V.I. Marconi, FaMAF-UNC and IFEG-CONICET, Córdoba, Argentina — In evolutionary biology choanoflagellates are broadly investigated as the closest living relatives of the animal ancestors. Under diverse environmental cues, choanoflagellate Salpingoeca rosetta can differentiate in two types of solitary swimming cells: slow and fast microswimmers. Here we present a first phenomenological 2D-model for the choanoflagellates dynamics confined into a flat device divided by a wall of asymmetric microconstrictions. The model allows us to optimize the geometry of the microchutes for directing and concentrating cell populations under strict control. We solve our set of dynamical equations using Langevin dynamics. Experimental parameters for the motility of the slow and fast cells were measured and used for our numerical estimations of the direct transport efficiency, otherwise we have no adjustable parameters. We find remarkable differences in the rectification results: choanoflagellates behave us a strategyflagellate-like devices, with a suitable microchannel selecting a specific cell type. For a given population velocity, narrower bottlenecks, of similar size to the cell dimension, show to be more efficient as concentrator of populations. Experiments and simulations are in good agreement.

12:06PM H25.00008 Data-driven, low-order modeling of interflagella synchronization. JONATHAN H. Tu, MURAT ARCAK, MICHEL M. MAHARBIZ, Univ of California - Berkeley — Synchronization is a common feature in the locomotive strategies employed by microswimmers. At the level of individual organisms, it can manifest as flagellar bundling or metachronal coordination of cilia. In large populations of microswimmers, interorganism coordination can result in collective behavior. This study focuses on the hydrodynamic interactions between two nearby flagella, looking to develop low-order models that accurately capture the dynamics of flagellar synchronization. Rather than build up a model based on simplified geometries and asymptotic expansions, we take a data-driven, top-down approach. For a single, isolated flagellum, our low-order model exactly reproduces the dynamics of a high-fidelity simulation of the full equations of motion. To extend the model to two flagella, we use insight gleaned from high-fidelity simulations, along with symmetry arguments, to eliminate terms in the equations of motion that are unrelated to synchronization effects. The resulting model accurately predicts synchronization rates at a greatly reduced computational cost. In future work, we hope to extend this approach to model larger numbers of interacting flagella, for which high-fidelity simulations become impractical.

12:19PM H25.00009 Hydrodynamic interactions of cilia on a spherical body. Babak Nasouri, Gwenny J. Elfring, The University of British Columbia, Vancouver — The emergence of metachronal waves in ciliated microorganisms can arise solely from the hydrodynamic interactions between the cilia. For a chain of cilia attached to a flat ciliate, it was observed that fluid forces can lead the system to form a metachronal wave. However, several microorganisms such as paramaecium and volvox possess a curved shaped ciliate body. To understand the effect of this geometry on the formation of metachronal waves, we evaluate the hydrodynamic interactions of cilia near a large spherical body. Using a minimal model, we show that for a chain of cilia around the sphere, the embedded periodicity in the geometry leads the system to synchronize. We also report an emergent wave-like behavior when an asymmetry is introduced to the system.

12:32PM H25.00010 Integration of hydrodynamic interactions between filaments. Yi man, Eric Lauga, University of Cambridge — In many biological situations, slender filaments interact through a viscous fluid, and these hydrodynamic interactions play a crucial cellular role. Examples include the ability of peritrichous bacteria to bundle their flagella or the generation of metachronal waves in cilia arrays. In most cases of interest, three distinct length scales characterize the filaments, their typical thickness $a$, relative distance $h$, and length $L$, which are asymptotically separated as $a \ll h \ll L$. In this talk, we demonstrate how to analytically develop a long-wavelength integration of hydrodynamic singularities in this biologically-relevant limit.

Monday, November 23, 2015 10:35AM - 12:45PM — Session H26 Biofluids: Phonation, Speech and Airway Mechanics — University of Maine

10:35AM H26.00001 Flow-Structure-Acoustic Interaction Computational Modeling of Voice Production inside an Entire Airway. Weili JIANG, Xudong ZHENG, Qian XUE, University of Maine — Human voice quality is directly determined by the interplay of dynamic behavior of glottal flow, vibratory characteristics of VF and acoustic characteristics of upper airway. These multiphysics constituents are tightly coupled together and precisely coordinate to produce understandable sound. Despite many years’ research effort, the direct relationships among the detailed flow features, VF vibration and aeroacoustics still remains elusive. This study utilizes a first-principle based, flow-structure-acoustics interaction computational modeling approach to study the process of voice production inside an entire human airway. In the current approach, a sharp interface immersed boundary method based incompressible flow solver is utilized to model the glottal flow. A finite element based solid mechanics solver is utilized to model the vocal vibration; A high-order immersed boundary method based acoustics solver is utilized to directly compute sound. These three solvers are fully coupled to mimic the complex flow-structure-acoustic interaction during voice production. The geometry of airway is reconstructed based on the in-vivo MRI measurement reported by Story et al. (1995) and a three-layer-tissue resonant based vocal fold model is taken from Titze and Talkin(1979). Results from these simulations will be presented and further analyzed to get new insight into the complex flow-structure-acoustic interaction during voice production. This study is expected to improve the understanding of fundamental physical mechanism of voice production and to help to build direct cause-effect relationship between biomechanics and voice sound.
10:48AM H26.00002 The effect of vocal fold vertical stiffness gradient on sound production. BIAO GEN, QIAN XUE, XUDONG ZHENG, University of Maine — It is observed in some experimental studies on canine vocal folds (VFs) that the inferior aspect of the vocal fold (VF) is much stiffer than the superior aspect under relatively large strain. Such vertical difference is supposed to promote the convergent-divergent shape during VF vibration and consequently facilitate the production of sound. In this study, we investigate the effect of vertical variation of VF stiffness on sound production using a numerical model. The vertical variation of stiffness is produced by linearly increasing the Young’s modulus and shear modulus from the superior to inferior aspects in the cover layer, and its effect on phonation is examined in terms of aerodynamic and acoustic quantities such as flow rate, open quotient, skewness of flow wave form, sound intensity and vocal efficiency. The flow-induced vibration of the VF is solved with a finite element solver coupled with 1D Bernoulli equation, which is further coupled with a digital waveguide model. This study is designed to find out whether it’s beneficial to artificially induce the vertical stiffness gradient by certain implanting material in VF restoring surgery, and if it is beneficial, what gradient is the most favorable.

11:01AM H26.00003 Dynamic and energetic relevance of glottal jet asymmetry¹. JUBIAO YANG, Rensselaer Polytechnic Institute, MICHAEL KRANE, Pennsylvania State University, LUCY ZHANG, Rensselaer Polytechnic Institute — Numerical simulation of phonation is performed using the fully-coupled Immersed Finite Element Method (IFEM), for both half-space and full-space domains. The full and half-space domains are identical, except that symmetric flow and structure motion is enforced in the half-space domain simulations. We evaluate and examine various terms in the momentum and energy equations to assess the dynamic relevance of glottal jet symmetry, as well as energy utilization in phonation. Specifically, control volume analyses based on simulation results are used to estimate glottal resistance and aeroacoustic source strengths, the level and character of radiated sound, and the various types of work done by laryngeal flow.

¹The support of grant R01 DC005642 from NIH is gratefully acknowledged.

11:14AM H26.00004 Phonatory sound sources in terms of Lagrangian Coherent Structures¹. MICHAEL MCPHAIL, MICHAEL KRANE, ARL Penn State — Lagrangian Coherent Structures (LCS) are used to identify sound sources in phonation. Currently, it is difficult to causally relate changes in airflow topology from voice disorders to changes in voiced sound production. LCS reveals a flow’s topology by decomposing the flow into regions of distinct dynamics. The aeroacoustic sources can be written in terms of the motion of these regions in terms of the motion of the boundaries of the distinct regions. Breaking down the flow into constituent parts shows how each distinct region contributes to sound production. This approach provides a framework to connect changes in anatomy from a voice disorder to measurable changes in the resulting sound. This approach is presented for simulations of some canonical cases of vortex sound generation, and a two-dimensional simulation of phonation.

¹Acknowledge NIH grant 2R01 2R01DC005642.

11:27AM H26.00005 Measurements of the three-dimensional oscillatory flow in a double bifurcation. ANDRAS NEMES, SAHAR JALAL, TRISTAN VAN DE MOORTELE, FILIPPO COLETTI, University of Minnesota — Above a certain ventilation frequency, the unsteady nature of the respiratory flow becomes apparent, and inhalation and exhalation cannot be approximated as quasi-stationary processes. This is especially important in the upper and central airways, where length and velocity scales are the largest, making inertia and acceleration effects dominant over viscous dissipation. We experimentally investigate the primary features of the oscillatory flow through a symmetric double bifurcation which models the self-similar branching of the human bronchial tree. We consider a range of Reynolds and Womersley numbers relevant to physiological conditions between the trachea and the lobar bronchi. Three-component, three-dimensional velocity fields are acquired at multiple phases within the ventilation cycle using magnetic resonance imaging (MRI), and are complemented with instantaneous two-dimensional fields obtained by particle image velocimetry (PIV). The phase-averaged volumetric data provide a description of the rich flow topology, characterizing the main secondary flow structures and their spatio-temporal evolution. The instantaneous measurements reveal some of the dynamics of the laminar-to-turbulent transition in the bifurcations, and its aperiodicity throughout the respiratory cycle.

11:40AM H26.00006 CFD simulations of a deforming human lung using dynamic and static CT images¹. SHINJIRO MIYAWAKI, ERIC A. HOFFMAN, CHING-LONG LIN, The University of Iowa — The authors have developed a CFD model to simulate airflow in deforming lungs using dynamic (4D) CT images. After obtaining the surface mesh for one CT image, we deformed the surface mesh to match other CT images using an image registration technique. During the CFD simulations, we deformed the surface mesh by cubic interpolation as a function of lung volume, and deformed the volume mesh using a computational solid mechanics-based algorithm. To investigate the effect of CT scanning method and relative hysteresis with respect to lung volume on pressure drop along the central airways, we performed CFD simulations using different numbers of 4D and static CT images of one healthy subject. Based on the simulation with 13 4DCT images, we found that air flow fractions in airways remain nearly constant over time. By comparing the simulations with 13, 2, and 1 4DCT images, we found that the overall effect of relative hysteresis of lung structure on pressure drop along each branch at peak inspiration was 12%, and the effect of deformation was 16%. As a result of the comparison between simulations with 2 and 1 of 4D and static CT images, the effect of CT scanning method was 16-39%, depending on the deformation of the lung.

¹NIH grants R01-HL094315, U01-HL114494, R01-HL112986, and S10-RR022421. Computer time provided by XSEDE.

11:53AM H26.00007 Energy utilization in phonation¹. MICHAEL KRANE, ARL Penn State — A control volume analysis of energy utilization in phonation is presented. Conversion of subglottal airstream potential energy into work done vibrating the vocal folds, air flowing through the glottis, and radiating sound are described. An approximate numerical model is used to compute the contributions of each of these mechanisms, as a function of subglottal pressure, for normal phonation. An efficiency measure for each energy conversion mechanism is proposed.

¹Acknowledge NIH grant 2R01 2R01DC005642.
12:06PM H26.00008 The evolution of viscous flow structures in the esophagus during tracheoesophageal speech. BYRON ERATH, FRANK HEMSING, Clarkson University — A laryngectomy is an invasive surgical procedure whereby the entire larynx is removed, usually as a result of cancer. Removal of the larynx renders conventional voiced speech impossible, with the most common remediation following surgery being tracheoesophageal (TE) speech. TE speech is produced by inserting a one-way valve to connect the posterior wall of the trachea with the anterior wall of the esophagus. As air is forced up from the lungs it passes through the prosthesis and into the esophagus. The resulting esophageal pressure field incites self-sustained oscillations of the pharyngoesophageal segment (PES), which ultimately produces sound. Unfortunately, the physics of TE speech are not well understood, with up to 50% of individuals unable to produce intelligible sound. This failure can be related to a lack of understanding regarding the esophageal flow field, where all previous scientific investigations have assumed the flow is one-dimensional and steady. An experimental TE speech flow facility was constructed and particle image velocimetry measurements were acquired at the exit of the model prosthesis (entrance of the esophagus). The flow is observed to be highly unsteady, and the formation and propagation of vortical flow structures through the esophageal tract are identified. Observations regarding the influence of the flow dynamics on the esophageal pressure field and its relation to the successful production of TE speech are discussed.

12:19PM H26.00009 Altered vocal fold kinematics in synthetic self-oscillating models that employ adipose tissue as a lateral boundary condition. HIBA SAIDII, BYRON D. ERATH, Clarkson University — The vocal folds play a major role in human communication by initiating voiced sound production. During voiced speech, the vocal folds are set into sustained vibrations. Synthetic self-oscillating vocal fold models are regularly employed to gain insight into flow-structure interactions governing the phonation process. Commonly, a fixed boundary condition is applied to the lateral, anterior, and posterior sides of the synthetic vocal fold models. However, physiological observations reveal the presence of adipose tissue on the lateral surface between the thyroid cartilage and the vocal folds. The goal of this study is to investigate the influence of including this substrate layer of adipose tissue on the dynamics of phonation. For a more realistic representation of the human vocal folds, synthetic multi-layer vocal fold models have been fabricated and tested while including a soft lateral layer representative of adipose tissue. Phonation parameters have been collected and are compared to those of the standard vocal fold models. Results show that vocal fold kinematics are affected by adding the adipose tissue layer as a new boundary condition.

12:32PM H26.00010 Study of human phonation in a full-body domain. SHAKTI SAURABH, DANIEL BODONY. Univ of Illinois - Urbana — The generation and propagation of the human voice is studied in two-dimensions using a full-body domain, using direct numerical simulation. The fluid/air in the vocal tract is modeled as a compressible and viscous fluid interacting with the non-linear, viscoelastic vocal folds (VF). The VF tissue material properties are multi-layered, with varying stiffness, and a finite-strain model is utilized and implemented in a quadrilateral finite element code. The fluid-solid domains are coupled through a boundary-fitted interface and utilize a Poisson equation-based mesh deformation method. The full-body domain includes the near VF region, the vocal tract, a simplified model of the soft palate and mouth, and extends out into the acoustic far-field. A new kind of inflow boundary condition based upon a quasi-one-dimensional formulation with constant sub-glottal volume velocity, which is linked to the VF movement, has been adopted. The sound pressure levels (SPL) measured are realistic and we analyze their connection to the VF dynamics and glottal and vocal tract geometries.

Supported by the National Science Foundation (CAREER award number 1150439)


10:35AM H27.00001 The roles of aerodynamic and inertial forces on maneuverability in flapping flight. HAMID VEJDAHNI, DAVID BOERMA, SHARON SWARTZ, KENNETH BREUER, Brown University — We investigate the relative contributions of aerodynamic and the whole-body dynamics in generating extreme maneuvers. We developed a 3D dynamical model of a body (trunk) and two rectangular wings using a Lagrangian formulation. The trunk has 6 degrees of freedom and each wing has 4 degrees of actuation (flapping, sweeping, wing pronation/supination and wing extension/flexion) and can be massless (like insect wings) or relatively massive (like bats). To estimate aerodynamic forces, we use a blade element method; drag and lift are calculated using a quasi-steady model. We validated our model using several benchmark tests, including gliding and hovering motion. To understand the roles of aerodynamic and inertial forces, we start by constraining the wing motion to flapping and wing length extension/flexion motion. This decouples the trunk degrees of freedom and affects only roll motion. For bats’ dynamics (massive wings), the model is much more maneuverable than the insect dynamics case, and the effect of inertial forces dominates the behavior of the system. The role of the aerodynamic forces increases when the wings have sweeping and flapping motion, which affects the pitching motion of the body. We also analyzed the effect of all wing motions together on the behavior of the model in the presence and in the absence of aerodynamic forces.

10:48AM H27.00002 What’s its wave? A 3D analysis of flying snake locomotion. ISAAC J. YEATON, GRANT A. BAUMGARDNER, TALIA M. WEISS, GARY NAVE, SHANE D. ROSS, JOHN J. SOCHA, Virginia Tech — Arboreal snakes of the genus Chrysopelea are the only known snakes to glide. To execute aerial locomotion, a snake jumps from a tree into the air while simultaneously flattening its body into an aerodynamically favorable shape. Snake gliding is distinguished by complex, three-dimensional body undulations resulting in a stable glide. However, these undulations have not been sufficiently characterized for a proper dynamical analysis. Here we ask, what is the body waveform employed during a glide, and how does this waveform enhance rotational stability? We report on recent glide experiments in which we recorded the three-dimensional body position during 8.5 m glides using a multi-camera motion-capture system. We quantify the body posture using complex modal analysis, which then serves as input in a variable-geometry rigid-body simulation of the snake while gliding. By separating the inertial and aerodynamic contributions in the equations of motion, we can now quantify the stability of the snake’s ‘gait’.

Supported by NSF 1361322

11:01AM H27.00003 Modification of the wake behind a bat ear with and without tubercles. CHRISTOPHER PETRIN, BRIAN ELBLING, Oklahoma State University — The Mexican Free-Tailed Bat (Tadarida brasiliensis) is a highly aerobatic bat, known to dive from altitudes of several thousand feet into their home caves, reaching estimated speeds of 27 m/s (Davis et al., Ecological Monographs, 32, 1962). A series of small tubercles have been observed on the leading edge of the bat’s ear, which mimic the pattern of tubercles found on the fins of the humpback whale (Megaptera novaeangliae). The tubercles on the whale fins have been proven to delay the onset of turbulence and allow the whale to maintain better control during dives. The goal of the current study is to assess whether the bat ear tubercles fulfill a similar purpose of improving flow control, particularly at high angles of attack. This was accomplished by acquiring PIV measurements of the bat ear wake with and without the tubercles. The velocity profiles were used to assess the drag and lift as a function of angle of attack. These results will be presented and the impact of the tubercles assessed.
11:14 AM H27.00004 Aerodynamics of a freely flying owl from PIV measurements in the wake. HADAR BEN-GIDA, Technion, ROI GURKA, Coastal Carolina University, DANIEL WEIHS, Technion — The mechanisms of the silent flight of owls have been the subject of scientific interest for many decades and a source of inspiration in the context of reducing flight noise. Over millions of years of evolution, owls have produced many specialized configurations to reduce the aerodynamic noise, which is found to be essential for successful hunting of potential prey. Here, we study how the three-dimensional flow field formed over the wing affect the vortical structures develop in the wake of a freely flying owl. We study the unique flight patterns of the Boobook owl; a mid-sized owl, which has the feature of stealth flight during both gliding and flapping flight. The owl was flown in a hypobaric avian wind tunnel at its comfort speed and deformation of hummingbird wings from high-quality high-speed videos. The observed wing surface morphing is highly complex and a pattern of the wingbeat cycle. The stealthy flight mode, which is a result of noise reduction mechanisms, formed over the wings (presumably by the leading-edge serrations) results in a unique signature in the wake flow field, which is characterized using the present data.

11:27 AM H27.00005 Minimum Wind Dynamic Soaring Trajectories under Boundary Layer Thickness Limits, GABRIEL BOUSQUET, MICHAEL TRIANTAFYLLOU, JEAN-JACQUES SLOTINE, Massachusetts Inst of Tech-MIT — Dynamic soaring is the flight technique where a glider, either avian or manmade, extracts its propulsive energy from the non-uniformity of horizontal winds. Albatrosses have been recorded to fly an impressive 5000 km/week at no energy cost of their own. In the sharp boundary layer limit, we show that the popular image, where the glider travels in a succession of half turns, is suboptimal for travel speed, airspeed, and soaring ability. Instead, we show that the strategy that maximizes the three criteria simultaneously is a succession of infinitely small arc-circles connecting transitions between the calm and windy layers. The model is consistent with the recordings of albatross flight patterns. This lowers the required wind speed for dynamic soaring by over 50% compared to previous beliefs. In the thick boundary layer limit, energetic considerations allow us to predict a minimum wind gradient necessary for sustained soaring consistent with numerical models.

11:40 AM H27.00006 Enhanced flight characteristics by heterogeneous autorotating wings, LIONEL VINCENT, MIN ZHENG, EVA KANSO, University of Southern California — We investigate experimentally the effect of mass distribution and flexibility on the descent motion of thin rectangular autorotating wings. We vary the wing thickness and the blade-element model based on quasi-steady flow assumption were adopted to analyze the aerodynamics. The result shows that while the weight support is generated during downstroke, little negative weight support is produced during upstroke. On the other hand, thrust is generated during both downstroke and upstroke, which allows the bird to overcome drag induced at fast flight. The lift and thrust characteristics are closely related to the instantaneous wing position and motion. In addition, the flow visualization shows that the leading-edge vortex is stable during most of the wing-beat, which may have contributed to the lift and thrust enhancement.

11:53 AM H27.00007 Computational modeling of aerodynamics in the fast forward flight of hummingbirds, JIALEI SONG, HAOXIANG LUO, Vanderbilt Univ, BRENDAN KINSELLA, University of Montana, TYSON HEDRICK, University of North Carolina at Chapel Hill — Computational models of the hummingbird at flight speed 8.3 m/s is built based on high-speed imaging of the real bird flight in the wind tunnel. The goal is to understand the lift and thrust production of the wings at the high advance ratio (flight speed to the average wingtip speed) around 1. Both the full 3D CFD model based on an immersed-boundary method and the blade-element model based on quasi-steady flow assumption were adopted to analyze the aerodynamics. The result shows that while the weight support is generated during downstroke, little negative weight support is produced during upstroke. On the other hand, thrust is generated during both downstroke and upstroke, which allows the bird to overcome drag induced at fast flight. The lift and thrust characteristics are closely related to the instantaneous wing position and motion. In addition, the flow visualization shows that the leading-edge vortex is stable during most of the wing-beat, which may have contributed to the lift and thrust enhancement.

12:06 PM H27.00008 Low Dimensional Analysis of Wing Surface Morphology in Hummingbird Free Flight, GREGORY SHALLCROSS, YAN REN, GENG LIU, HAIBO DONG, University of Virginia, BRENDAN KINSELLA, University of Montana — Surface morphology in flapping wings is a hallmark of bird flight. In current work, the role of dynamic wing morphing of a free flying hummingbird is studied in detail. A 3D image-based surface reconstruction method is used to obtain the kinematics and deformation of hummingbird wings from high-quality high-speed videos. The observed wing surface morphing is highly complex and a number of modeling methods including singular value decomposition (SVD) are used to obtain the fundamental kinematical modes with distinct motion features. Their aerodynamic roles are investigated by conducting immersed-boundary-method based flow simulations. The results show that the chord-wise deformation modes play key roles in the attachment of leading-edge vortex, thus improve the performance of the flapping wings.

12:19 PM H27.00009 A numerical study of a freely-falling maple seed with autorotation, INJAE LEE, HAECHEON CHOI, Seoul National University — Many single winged seeds such as those of maples exploit autorotation to decrease the descending velocity and increase the dispersal distance for the conservation of species. In this study, a numerical simulation is conducted for flow around a freely-falling maple seed (Acer palmatum) at the Reynolds number of 1186 (based on the mean chord length and characteristic terminal velocity). We use an immersed boundary method in a non-inertial reference frame (Kim & Choi, JCP, 2006) for the simulation. After a transient period, the seed reaches the steady autorotation with a stable leading edge vortex attached on the surface of the wing at which the descending velocity significantly decreases. At steady autorotation, the descending velocity is proportional to the square root of disc loading. We also study the effect of the initial position of the seeds on the timing of autorotation, and show that the autorotation occurs earlier when the wing leading edge or nut is initially positioned upward.

1Supported by NSF CBET-0954381

1This work is supported by NSF CBET-1313217 and AFOSR FA9550-12-1-0071
Bumblebees meet fully developed turbulence: high resolution numerical simulations, THOMAS ENGELS, M2P2-CNRS, Aix-Marseille University, Marseille, France & Institut für Strömungsmechanik und Technische Akustik (ISTA), TU Berlin, Germany, DMITRY KOLOMENSKY, Biomechanical Engineering Laboratory, Chiba University, Chiba, Japan, KAI SCHNEIDER, M2P2-CNRS & CMI Aix-Marseille University, Marseille, France, JOERN SESTERHENN, Institut für Strömungsmechanik und Technische Akustik (ISTA), TU Berlin, Germany, FRITZ-OLAF LEHMANN, Department of Animal Physiology, Institute of Biological Sciences, University of Rostock, Rostock, Germany — Numerical experiments of a tethered bumblebee in a wind tunnel with turbulent inflow of different intensity are performed at realistic Reynolds numbers on massively parallel computers. Ensemble averaging of different flow realizations shows that the mean forces (lift and drag, or horizontal and vertical), the moments (roll, pitch and yaw), and power, are robust and are not modified significantly by the turbulent inflow. Phase averaging of the vorticity field illustrates that in all cases the leading edge vortex is indeed persistent (in the average sense) as it is the case for laminar inflow, which explains the above findings. However, as expected, the corresponding standard deviations do increase with the turbulence intensity. In particular the roll moment shows the strongest increase of standard deviation. Considering that the moment of inertia of the bumblebee is the smallest around this axis yields a possible explanation for the experimentally observed flight instability around the roll axis under turbulent flow conditions.


10:35AM H28.00001 Reducing Sliding Friction with Liquid-Impregnated Surfaces, MOHAMMAD HABIBI, Virginia Tech, C. PATRICK COLLIER, Oak Ridge National Laboratory, JONATHAN BOREYKO, Virginia Tech, NATURE INSPIRED FLUIDS AND INTERFACES TEAM, CENTER FOR NANOPHASE MATERIALS SCIENCES TEAM — Liquid-impregnated surfaces are fabricated by using a lubricating liquid into the micro/nano roughness of a textured substrate, such that the surface is slippery for any deposited liquid immiscible with the lubricant. To date, liquid-impregnated surfaces have almost exclusively focused on repelling liquids by minimizing the contact angle hysteresis. Here, we demonstrate that liquid-impregnated surfaces are also capable of reducing sliding friction for solid objects. Ordered arrays of silicon micropillars were infused with lubricating liquids varying in viscosity by two orders of magnitude. Five test surfaces were used: two different micropillared surfaces with and without liquid infusion and a smooth, dry control surface. The static and kinetic coefficients of friction were measured using a polished aluminum cube as the sliding object. Compared to the smooth control surface, the sliding friction was reduced by at least a factor of two on the liquid-impregnated surfaces.

10:48AM H28.00002 Differential approach to Capillary Breakup Rheometry: role of filament asymmetry induced by sample volume and strain, LOUISE MCCARROLL, WILLIAM SCHULTZ, MICHAEL SOLOMON, University of Michigan — We investigate the operating range of the 1-D, Newtonian, differential analysis for capillary breakup rheometry. Capillary breakup rheometry (CBR) derives specimen physical properties (e.g. viscosity) from measurements of the filament evolution after a sudden deformation. In our differential analysis, derivatives of the filament radius as a function of the axial coordinate and time are measured to determine the ratio of surface tension to viscosity. We evaluate the accuracy of the differential method by applying it to Newtonian fluids with a range of viscosities and for experiments with different sample volumes and strains. We investigate the impact of filament asymmetry on the performance of the differential method for the range of conditions studied and with a 1-D numerical model. This evaluation yields recommendations for using the differential CBR technique. We discuss the scope for extending the differential analysis to more complex cases, such as for insoluble surfactant at the fluid-air interface.

11:01AM H28.00003 Stretching liquid bridges with moving contact lines: Comparison of model predictions and experiments, CHUNG-HSUAN HUANG, University of Minnesota, MARCIO CARVALHO, PUC-Rio, SATISH KUMAR, University of Minnesota — Transfer of liquid from one surface to another plays a key role in printing processes. During liquid transfer, a liquid bridge is formed and then undergoes significant extensional motion while its contact lines are free to move on the bounding solid surfaces. In this work, we develop slender-jet and finite-element models of this phenomenon and compare the resulting predictions with experimental data. For very low capillary numbers (quasi-static stretching), predictions from both models agree well with the experimental data. For O(1) capillary numbers, the models predict that each surface receives half the liquid, in agreement with experiments. For intermediate capillary numbers, each model can deviate substantially from each other and from the experimental data due to deviations between the predicted and the observed contact-line motion. The models are also used to understand the influence of initial bridge shape on liquid transfer and to rationalize experimental observations. The results from these fundamental studies will aid the optimization of gravure and other printing processes for manufacturing of printed electronic devices.

11:14AM H28.00004 Wetting and phase separation in soft adhesion, KATHARINE JENSEN, ERIC DUFRESNE, Yale University — In the classic theories of solid adhesion, surface energies drive deformation to increase contact area while bulk elasticity opposes it. However, recently solid surface tension has also been shown to play an important role in resisting deformation in soft materials. We explore the consequences for the physics of adhesive contact by performing experiments bringing small, rigid spheres into contact with compliant silicone gel substrates. We measure the quasi-static deformation of the substrate, particularly focusing on its structure near the contact line. In order to satisfy the wetting condition prescribed by surface tension balance while avoiding an elastic singularity at the contact line, we find that the gels undergo an adhesion-induced phase separation. This creates a four-phase contact zone with two additional, hidden contact legs. Our results indicate that accurate theories of adhesion of soft gels need to account both for the compressibility of the gel elastic network and for a non-zero surface stress between the gel and its solvent.

11:27AM H28.00005 Cusps and cuspidal edges at fluid interfaces: existence and application, ROUSLAN KRECHETNIKOV, University of Alberta — One of the intriguing questions in fluid dynamics is on the interrelation between dynamic singularities in the solutions of fluid dynamic equations – unboundedness of the velocity field in an appropriate norm – and the geometric ones – divergence of curvature at fluid interfaces. The present talk focuses on two generic interfacial singularities – genuine cusps and cuspidal edges – found here in both two and three dimensions thus establishing a relation between real fluid interfaces and geometric singularity theory. The key new finding is the necessary condition for the existence of geometric singularities, which is a variation of surface tension. It is also established here that the dynamic and geometric singularities entail each other only in the case of three-dimensional cusps. Explicit asymptotic solutions for the flow field and interface shape near steady-state singularities at fluid interfaces are developed as well. The practical motivation for the present study comes from the fundamental role interfacial singularities play in sustaining self-driven conversion of chemical into mechanical energy.
Institute of Technology — This talk discusses an experimental study of convection in a layer of volatile liquid subject to a horizontal temperature gradient driven by a horizontal temperature difference across the surface element allowing the study of the surface tension from a mechanical standpoint. Analysis of its Fourier components gives both structural and dynamical routes to compute the surface based on hydrodynamic theory. The vapour interface that binary-fluid coolants could reduce film dryout in two-phase thermal management devices. At lower C for a range of pressures (and therefore different concentrations of air in the vapor space above the liquid), C is found to be strong enough to drive the liquid near the free surface towards the heated end over the entire horizontal 4.9 cm extent of the liquid layer when c < 60% and the MeOH molar fraction C > 20%, suggesting that binary-fluid coolants could reduce film dryout in two-phase thermal management devices. At lower C, the flow reverses near the heated end, however, suggesting that thermocapillarity is dominant at low (local) MeOH concentration. The maximum flow speed is found to be in reasonable agreement with that predicted by lubrication theory at low C. At C > 80%, thermocapillarity drives the flow near the free surface away from the heated end. Finally, the flow becomes unsteady at intermediate values of C.

We acknowledge financial support from European Research Council via Advanced Grant No. 247031.

11:53AM H28.00007 A flow map of Buoyancy-Marangoni convection in binary fluids driven by a horizontal temperature gradient1, YAOFA LI, University of Notre Dame, MINAMI YODA, Georgia Institute of Technology — This talk discusses an experimental study of convection in a layer of volatile liquid subject to a horizontal temperature gradient, driven by thermocapillarity, solutocapillarity and buoyancy. We investigated a ~ 0.3 cm-deep layer of a methanol(MeOH)-water mixture in a sealed rectangular cuvette driven by a temperature difference ΔT ≈ 6 °C for a range of pressures (and therefore different concentrations of air in the vapor space above the liquid c). Solutocapillarity was found to be strong enough to drive the liquid near the free surface towards the heated end over the entire horizontal 4.9 cm extent of the liquid layer when c < 60% and the MeOH molar fraction C > 20%, suggesting that binary-fluid coolants could reduce film dryout in two-phase thermal management devices. At lower C, the flow reverses near the heated end, however, suggesting that thermocapillarity is dominant at low (local) MeOH concentration. The maximum flow speed is found to be in reasonable agreement with that predicted by lubrication theory at low C. At C > 80%, thermocapillarity drives the flow near the free surface away from the heated end. Finally, the flow becomes unsteady at intermediate values of C.

Supported by ONR and NSF

12:06PM H28.00008 Thermo/Soluto-capillary instabilities in 3D bi-component liquid pools using DNS1, ADAM WILLIAMS, The University of Edinburgh, PEDRO SAENZ, MIT, PRASHANT VALLURI, KHELLIL SEFIANE, The University of Edinburgh — The behaviour of surface tension dominated flows in the presence of a temperature gradient and phase change is of great importance in designing micro-cooling devices. While evaporating pools and droplets have been investigated numerically and experimentally, these studies have dealt only with pure fluids. For bicomponent liquid mixtures, limited experimental studies have been conducted but a rigorous numerical model is absent. We present a two-phase multicomponent DNS model to simulate thermo/soluto-capillary instabilities in bicomponent liquid layers subject to a horizontal temperature gradient. The strategy fully accounts for a deformable interface using a variant of volume-of-fluid method. The presence of a second component introduces thermophoresis in the liquid phase which then gives rise to solutal Marangoni effects. By combining mixture thermodynamics with multiphase DNS, we investigate thermo/soluto-capillary and interfacial instabilities of a 3D bicomponent liquid pool. An important aspect we quantify is the strength of solutal over thermal Marangoni convection and its effect on stability of resultant interfacial waves and phase-separation in the liquid. The model is robust enough to include phase-change and the advection-diffusion of volatile species in the gas phase.

1Funded by EPSRC, Grant No. EP/K00963X/1

12:19PM H28.00009 Tuning the Dynamics of Particles and Drops at Engineered Nanostructured Interfaces, CARLOS COLOSQUI, State Univ of NY- Stony Brook, ANTONIO CHECCO, Brookhaven National Laboratory — Harnessing the full potential of current nanofabrication capabilities requires significant progress in understanding non-equilibrium phenomena produced by nanoscale interfacial structure and thermal motion. In diverse colloidal systems relevant to complex fluids and soft materials, the nanoscale interfacial structure can induce transitions from fast dynamics dominated by (deterministic) hydrodynamic and surface forces to arrested dynamics dominated by (random) thermally-activated processes. Recent work provides guidelines for engineering geometries and surface structures to tune the dynamic behavior of nano/microscale particles and droplets. For example, small reductions of the radius of a microparticle can lead to dramatic increases in the time for adsorption at liquid interfaces or membranes. Similarly, reducing the radius of a millimeter-sized droplet can lead to arrested spreading dynamics with logarithmic-in-time relaxation. Furthermore, nanostructured surfaces with directional asymmetry can convert thermal motion into directed transport processes at controllable rates. This talk will discuss theoretical and computational predictions that have been confirmed in recent experimental work by our and other groups and new predictions that can guide future experimental studies.

12:32PM H28.00010 Antifreeze Polysaccharide Coating Study for De-icing Aircraft, KATSUJAI MORITA, JAXA, HIROTAKA SAKAUE, University of Notre Dame, AZUMA ANDO, Hitotsubashi University, YOSHIIKU MATSUDA, HIDEHISA KAWAHA, Kansai University — Anti-icing or deicing of an aircraft is necessary for a safe flight operation. Mechanical processes, such as heating and deicer boot, are widely used. Deicing fluids, such as propylene glycol and ethylene glycol, are used to coat the aircraft. However, these should be coated every time before the take-off, since the fluids come off from the aircraft while cruising. We study an antifreeze polysaccharide (AFPS) coating as a deicer for an aircraft. It is designed to coat on the aircraft without removal. Since an AFPS coating removes ice by reducing the interfacial energy, it would be an alternative way to prevent ice on the aircraft. We provide a temperature-controlled room, which can control its temperature under icing conditions (-8 and -4 °C). Ice adhesion tests are performed for AFPS coating and compared with a fundamental specimen without the coating.

Monday, November 23, 2015 10:35AM - 12:45PM –
Session H29 Nonlinear Dynamics: Model Reduction 310 - Themistoklis Sapsis, MIT
A variational principle for the extraction of time-dependent modes associated with transient instabilities

THEMISTOKLIS SAPSIS, HESSAM BABAEEMIT — We introduce a variational formulation for the determination of a finite-dimensional, time-dependent, orthonormal basis that captures directions of phase space associated with finite-time instabilities. While these instabilities have finite lifetime they can play a crucial role either by altering the system dynamics through the activation of other instabilities, or by creating sudden nonlinear energy transfer that lead extreme responses. However, their essentially transient character makes their description a particularly challenging task. Here we develop a variational framework that focuses on the optimal approximation of the system dynamics over finite-time intervals under the orthonormal basis constraint. This variational formulation results in differential equations that evolve a time-dependent basis so that it optimally approximates the most unstable directions over finite times.

1Supported by ARO Award # 66710-EG-YIP.

Time-dependent modes associated with finite time instabilities in unstable fluid flows

HESSAM BABAEEMIT — We apply a recently developed variational formulation for the determination of a finite-dimensional, time-dependent, orthonormal basis that captures the directions associated with finite-time instabilities. We demonstrate the capability of the method for two problems: the Orr-Sommerfeld/Squire operator and the vertical jet in crossflow. In the first problem we demonstrate that the time-dependent subspace captures the strongly transient non-normal energy growth (in the short time regime), while for longer times the modes capture the expected asymptotic behavior of the dynamics. We also consider the vertical jet in crossflow at the jet Reynolds number of Rej = 900. We demonstrate that the subspace instantaneously captures the most unstable directions of the time-dependent flow. We explore the connection between the shear flow, non-normal growth and persistent instabilities.

1Supported by ARO Award # 66710-EG-YIP.

Nonlinear reduced order models for fluids systems using extended dynamic mode decomposition

SCOTT DAWSON, CLARENCE ROWLEY, Princeton University — The development of techniques that can extract simple, accurate, and computationally tractable models from fluids data is of importance for enhanced prediction, control, and fundamental understanding of such systems. Modeling approaches can take the form of identifying modes upon which to project the governing equations (e.g., Galerkin projection onto a set of POD modes), or in determining (or calibrating) the temporal dynamics from data, such as in dynamic mode decomposition (DMD), or various modifications to Galerkin projection. Here, we demonstrate that choosing appropriate observables (such as linear and quadratic monomials of POD coefficients) can allow for nonlinear behavior to be accurately captured using the recently proposed extended DMD algorithm. For cylinder wake data spanning the transient and vortex shedding limit cycle regimes, the identified nonlinear models show significant improvement in accuracy and robustness over standard DMD and Galerkin projection. Compared to traditional DMD, this approach should allow for a better global approximation of the Koopman operator for the dynamical system. We make connections with other related model identification algorithms, and additionally investigate the performance of the method upon spatially sparse and noisy data.

1This work was supported by the Air Force Office of Scientific Research, under award No. FA9550-12-1-0075

Network Structure of Two-Dimensional Homogeneous Turbulence

KUNIHIKO TAIRA, ADITYA NAIR, Florida State University, STEVEN BRUNTON, University of Washington — The network structure of two-dimensional incompressible homogeneous turbulence is characterized by highlighting the vortical interactions in the flow field. By analyzing the degree distribution of the turbulence network, it is observed that turbulence has an underlying scale-free network that describes how vortical structures are interconnected. In the network-theoretic framework, we can identify strong vortices that serve as hubs that are strongly connected to other vortical hubs. Smaller and weaker eddies are found to be predominantly influenced by the neighboring hubs. These observations complement previous knowledge of turbulence based on vortex dynamics. The time evolution of the fluid flow network shows that the scale-free property is achieved when turbulence is sustained but is not observed when the flow reaches a laminar regime through dissipation. The finding that turbulence has a scale-free interaction network enables us to identify the type of perturbations that turbulence is resilient against. These insights from network analysis enable us to examine how the behavior of turbulent flows can be modified.

1This work was supported by the US Army Research Office (Grant W911NF-14-1-0386) and the US Air Force Office of Scientific Research (Grant FA9550-13-1-0183).

Introducing Spectral Proper Orthogonal Decomposition: Superior identification of coherent structures in turbulent flows

MORITZ SIEBER, KILIAN OBERLEITHNER, C. OLIVER PASCHEREIT, Chair of Fluid Dynamics, Hermann-Foettinger-Institut, TU Berlin — The identification of coherent structures from experimental or numerical data is an essential task in fluid dynamics. Today’s commonly used approaches employ the construction of a modal base that captures the dominant flow structures. Typically, these modes are either energy (POD) or frequency (Fourier decomposition) ranked. However, there are numerous examples where the relevant coherent structures occur at low energies or at multiple frequencies. To overcome the shortcoming of the current “rigid” approaches, we propose a new method - Spectral Proper Orthogonal Decomposition (SPOD). It is based on classical POD and it applies to spatially and temporally resolved data. The new method involves an additional temporal constraint that enables a clear separation of phenomena that occur at multiple frequencies and energies. It allows for a continuous shifting from the energy ranked POD to the frequency ranked Fourier decomposition by changing a single parameter. In this presentation we demonstrate the SPOD on experimental data of some flow cases, where the commonly used methods fail to assign the relevant coherent structures to single modes. The SPOD, however, achieves a proper separation of spatially and temporally coherent structures that might be hidden in noise or spread over a wide frequency range. In spite of all these benefits, the algorithmic complexity and computational cost of the SPOD is still comparable to the snapshot POD.
11:40AM H29.00006 Dynamic reconstruction of sub-sampled data using Optimal Mode Decomposition. JAKUB KROL, ANDREW WYNN, Imperial College London — The Nyquist-Shannon criterion indicates the sample rate necessary to identify information with particular frequency content from a dynamical system. However, in experimental applications such as the interrogation of a flow field using Particle Image Velocimetry (PIV), it may be expensive to obtain data at the desired temporal resolution. To address this problem, we propose a new approach to identify temporal information from undersampled data, using ideas from modal decomposition algorithms such as Dynamic Mode Decomposition (DMD) and Optimal Mode Decomposition (OMD). The novel method takes a vector-valued signal sampled at random time instances (but at Sub-Nyquist rate) and projects onto a low-order subspace. Subsequently, dynamical characteristics are approximated by iteratively approximating the flow evolution by a low order model and solving a certain convex optimization problem. Furthermore, it is shown that constraints may be added to the optimization problem to improve spatial resolution of missing data points. The methodology is demonstrated on two dynamical systems, a cylinder flow at Re = 60 and Kuramoto-Sivashinsky equation. In both cases the algorithm correctly identifies the characteristic frequencies and oscillatory structures present in the flow.

11:53AM H29.00007 Data-driven reduced order model for prediction of wind turbine wake dynamics. MITHU DEBNATH, CHRISTIAN SANTONI, MARIO A. ROTEA, STEFANO LEONARDI, GIACOMO VALERIO IUNGO, UT Dallas — Wind turbine wakes are highly turbulent flows for which coherent vorticity structures lead to complex dynamics and instabilities. In this study, high-fidelity large eddy simulations (LES) data of a utility-scale wind turbine is analyzed through proper orthogonal decomposition (POD) and dynamic mode decomposition (DMD) in order to detect the main dynamic contributions to the temporal and spatial evolution of a wind turbine wake. Eigenmodes obtained from modal decomposition are clustered as a function of their physical origin, energy, spectral contribution and growth rate. A subset of the eigenmodes is then selected according to a customized objective function in order to represent an optimal blend of the different dynamic contributions. The selected eigenmodes are embedded in a time-marching algorithm enabling the prediction of the wake velocity field and loads on downstream turbines. This reduced order model is characterized by a relatively low rank compared to the dimension of the physical space of the original LES data, thus by a low computational cost. The reduced order model is then embedded within a Kalman filter in order to perform data assimilation of new available observations in order to maximize agreement between the forecast and observations.

12:06PM H29.00008 Network-based representation of energy transfer in unsteady separated flow1. ADITYA NAIR, KUNIHICO Taira, Florida State University — We construct a network-based representation of energy pathways in unsteady separated flows using a POD-Galerkin projection model. In this formulation, we regard the POD modes as the network nodes and the energy transfer between the modes as the network edges. Based on the energy transfer analysis performed by Noack et al. (2008), edge weights are characterized on the interaction graph. As an example, we examine the energy transfer within the two-dimensional incompressible flow over a circular cylinder. In particular, we analyze the energy pathways involved in flow transition from the unstable symmetric steady state to periodic shedding cycle. The growth of perturbation energy over the network is examined to highlight key features of flow physics in order to represent an optimal blend of the different dynamic contributions. The selected eigenmodes are embedded in a time-marching algorithm enabling the prediction of the wake velocity field and loads on downstream turbines. This reduced order model is characterized by a relatively low rank compared to the dimension of the physical space of the original LES data, thus by a low computational cost. The reduced order model is then embedded within a Kalman filter in order to perform data assimilation of new available observations in order to maximize agreement between the forecast and observations.

12:19PM H29.00009 Principal interval decomposition framework for POD-based model reduction of convective flows. OMAR SAN, Oklahoma State University, JEFF BORGGAAARD, Virginia Tech — A principal interval decomposition (PID) framework is proposed to build more reliable reduced-order models for unsteady flow problems. The PID method optimizes the lengths of the time windows over which proper orthogonal decomposition (POD) is performed and can be highly effective in building reduced-order models for convective problems. The performance of these POD models with and without using the PID approach is investigated by applying these methods to the unsteady lock-exchange flow problem modeled by solving the Boussinesq equations in vorticity-streamfunction formulation. This benchmark problem exhibits a strong shear flow induced by a temperature jump and results in the Kelvin-Helmholtz instability. This is considered a challenging benchmark problem for the development of reduced order models. The predictive performance of our model is then analyzed over a wide range of computational modeling and physical parameters. It is shown that the PID approach provides a significant improvement in accuracy over the standard Galerkin POD reduced-order model. Our numerical assessment of the PID shows that it may represent a reliable model reduction tool for convection-dominated, unsteady-flow problems.

12:32PM H29.00010 Reduced-order modeling of the flow around a high-lift configuration with unsteady Coanda blowing1. RICHARD SEMAAN, Tech Univ Braunschweig, LAURENT CORDIER, PPRI ME, Universite de Poitiers, BERND NOACK, PPRI ME, Universite de Poitiers, Tech Univ Braunschweig, PRAD EE KUMAR, MARCO BURNAZZI, Tech Univ Braunschweig, GILLES TISSOT, PPRI ME, Universite de Poitiers — We propose a low-dimensional POD model for the transient and post-transient flow around a high-lift airfoil with unsteady Coanda blowing over the trailing edge. This model comprises the effect of high-frequency modulated blowing which mitigates vortex shedding and increases lift. The structure of the dynamical system is derived from the Navier-Stokes equations with a Galerkin projection and from subsequent dynamic simplifications. The system parameters are determined with a data assimilation (4D-Var) method. The boundary actuation is incorporated into the model with actuation modes following Graham et al. (1999); Kasnako˘glu et al. (2008). As novel enabler, we show that the performance of the POD model significantly benefits from employing additional actuation modes for different frequency components associated with the same actuation input. In addition, linear, weakly nonlinear and fully nonlinear models are considered. The current study suggests that separate actuation modes for different actuation frequencies improve Galerkin model performance, in particular with respect to the important base-flow changes.


Monday, November 23, 2015 10:35AM - 12:45PM –
Session H30 DFD GPC: Geophysical Fluid Dynamics: Air-Sea and Wave Interaction
311 - Baylor Fox-Kemper, Brown University
10:35AM H30.00001 Interactions of Ocean Fronts with Waves and Turbulence1, BAYLOR FOX-KEMPER, NOBUHIRO SUZUKI, Brown Univ — High resolution simulations and observations of the ocean surface boundary layer have revealed 100m to 10km frontal and filamentary structures in temperature and other properties worldwide. The formation and evolution of these features, through frontogenesis, instability, and frontolysis, is an important and often poorly-simulated part of the climate system, yet fronts and filaments strongly affect surface layer dynamics and the transport of energy, momentum, and gases through this layer. These features also dominate the transport of oil spills and pollutants over a wide range of scales. Analysis of a multi-scale, non-hydrostatic, large eddy simulation spanning 20km fronts to 5m turbulence will be presented. The theory of the interactions of the fronts with turbulence and surface waves will be illustrated, and the consequences of these interactions on frontal strength and tracer transport will be quantified.

1Supported by NSF 1258907 and BP/The Gulf of Mexico Research Initiative (CARTHE)

10:48AM H30.00002 LES of Langmuir supercells under constant crosswind tidal forcing1, RACHEL WALKER, JIE ZHANG, MARIO JUHA, University of South Florida, CHESTER GOSCH, Old Dominion University, ANDRES TEJADA-MARTINEZ, University of South Florida — We report on the impact of a crosswind tidal current on Langmuir supercells (LSCs) in shallow water computed via LES. LSCs consist of parallel counter rotating vortices engulfing the water column in unstratified conditions. These cells have been observed in shallow continental shelf regions of ~15 meters depth during the passage of storms. The cells are aligned roughly in the crosswind direction and are powered by the shear current with the Stokes drift velocity induced by surface gravity waves. Without tides, LES reveals that the typical crosswind width of a LSC is ~4 times the water column depth (H). Under a relatively weak crosswind tidal current (weaker than the downwind current), the constant crosswind tidal forcing applied causes a merging of cells leading to cells of width ~H. The opposite occurs under a crosswind tidal current stronger than the downwind current as the constant crosswind tidal force is able to break up the LSCs giving rise to smaller scale cells with different turbulent structure than that associated with LSC. The statistics of the turbulence during strong and weak crosswind tides will be contrasted and implications of an oscillating crosswind tidal force will be discussed.

1Support from the US National Science Foundation and the Gulf of Mexico Research Initiative is gratefully acknowledged

11:01AM H30.00003 Characteristics and Evolution of Passive Tracers in the Oceanic Mixed Layer, KATHERINE SMITH, PETER HAMLINGTON, University of Colorado Boulder, BAYLOR FOX-KEMPER, Brown University — Ocean tracers such as CO2 and plankton reside primarily in the mixed layer where air-sea gas exchange occurs and light is plentiful for photosynthesis. There can be substantial heterogeneity in the distributions of these tracers due to turbulent mixing, particularly in the submesoscale regime where partly geostrophic eddies and small-scale 3D turbulence are both active. In this talk, LES spanning scales from 20km down to 5m are used to examine the role of turbulent mixing on nonreactive passive ocean tracers. The simulations include the effects of both wave-driven Langmuir turbulence and submesoscale eddies, and tracers with different initial and boundary conditions are examined. Tracer properties are characterized using spatial fields, statistics, multiscale fluxes, and spectra, and results show that passive tracer mixing depends on the air-sea flux rate, release depth, and flow regime. The results indicate that submesoscale eddies transport tracer energy upward to extract potential energy, the same is not true of passive tracers, whose entrainment is instead suppressed. Early in the evolution of some tracers, counter-gradient transport occurs co-located with regions of negative potential vorticity, suggesting that symmetric instabilities may act to oppose turbulent mixing.

11:14AM H30.00004 Propagation of acoustic pulses in random gravity wave fields, CHRISTOPHE MILLET, CEA, DAM, DIF, ALVÁRIO DE LA CAMARA, CMLA, ENS Cachan, FRANÇOIS LÖTT, LMD CNRS, ENS Paris — A linear solution modeling the interaction between an incoming acoustic wave and a randomly perturbed atmosphere is developed, using the normal mode method. The wave mode structure is determined by a sound speed profile that is confining. The environmental uncertainty is described by a stochastic field obtained with a multiwave stochastic parameterization of gravity waves (GW). Using the propagating modes of the unperturbed atmosphere, the wave propagation problem is reduced to solving a system of ordinary differential equations. We focus on the asymptotic behavior of the transmitted waves in the weakly heterogeneous regime. In this regime, the coupling between the acoustic pulse and the randomly perturbed waveguides is weak and the propagation distance must be large enough for the wave to experience significant scattering. A general expression for the pressure far-field is derived in terms of saddle-point contributions. The saddle-points are obtained from a WKB approximation of the vertical eigenvalue problem. We present preliminary results that show how statistics of the transmitted signal are related to some eigenvalues and how an “optimal” GW field can trigger large deviations in the acoustic signals. The present model is used to explain the variability of infrasound signals.

11:27AM H30.00005 On the relative contribution of inertia-gravity wave radiation to asymmetric instabilities in tropical cyclone-like vortices1, KONSTANTINOS MENELAOU, McGill University, DAVID A. SCHECHETER, NorthWest Research Associates, PETER M. K. YAU, McGill University — Intense geophysical vortices may experience various asymmetric instabilities during their life cycles. This study presents a method for evaluating the relative importance of different mechanisms that can simultaneously influence the growth of an asymmetric perturbation. The method is illustrated for vortices whose basic states are barotropic and have mononotonic radial distributions of potential vorticity (PV). A diagnostic formula for the growth rate of the perturbation is derived from an equation expressing conservation of angular pseudomomentum. In this formula, the growth rate is decomposed into several components relevant to the most unstable modes. One component accounts for the destabilizing interaction of phase-locked counter-propagating vortex Rossby (VR) waves. Other components account for inertia-gravity (IG) wave radiation and PV stirring in one or more critical layers. The dominant instabilities are examined in a parameter regime deemed relevant to tropical cyclone perturbations. As the Froude number increases from its lower bound, the main cause of instability typically transitions from VR-VR wave interaction (or critical layer stirring) to IG wave radiation. The transition can occur gradually or abruptly at a critical point for reasons that will be explained.

1NSERC and NSF Grants AGS-1101713 and AGS-1250533

11:40AM H30.00006 Wave modulation: the geometry, kinematics, and dynamics of surface-wave packets, NICHOLAS PIZZO, W. KENDALL MELVILLE, Scripps Institution of Oceanography, UCSD — We derive moment evolution equations of the modified nonlinear Schrodinger equation (MNLS) with application to interpreting the geometry, kinematics and dynamics of focusing deep-water wave packets. Our theory predicts modifications to the group velocity and associates wave packet convergence with the breakdown of equipartition between kinetic and potential energy. The evolution of the first moment of the energy density yields a natural way to interpret the concept of group velocity for these compact wave groups, predicting a velocity increase as the packet focuses, and is found to be up to 10% larger than that predicted by linear theory, consistent with laboratory observations. The second moment yields a virial theorem, associating energy convergence with deviations from equipartition. The derivation of these moment equations relies crucially on the variational structure of the spatial version of the MNLS, and the subsequent derivation of three conservation laws. These predictions are then examined numerically for focusing wave packets governed by both the MNLS as well as the full potential flow equations, and the results are discussed in the context of existing theoretical, numerical and laboratory studies.
Case where both the frequency $\omega$ distribution is found to follow a power law of the radius, $r^{-3}$, and to scale linearly with the time dependent turbulent dissipation rate during the active breaking stages. The time-averaged bubble size distribution is found to follow the same power law of the radius and to scale linearly with the wave dissipation rate per unit length of breaking crest. We propose a phenomenological turbulent bubble break-up model that describes the numerical results and existing experimental results. [1] Popinet, S. 2003. Journal of Computational Physics 190, 572-. [2] Deike, L., Popinet, S., and Melville, W.K. 2015. Journal of Fluid Mechanics. vol 769, p541-569.

Air entrainment and bubble statistics in three-dimensional breaking waves, Luc Deike, W.K. Melville, Scripps Institution of Oceanography, University of California San Diego, Stephane Popinet, Institut d’Alembert, UPMC, Paris — Wave breaking in the ocean is of fundamental importance in order to quantify wave dissipation and air-sea interaction, including gas and momentum exchange, and to improve parameterizations for weather and climate models. Here, we investigate air entrainment and bubble statistics in three-dimensional breaking waves through direct numerical simulations of the two-phase air-water flow using the Open Source solver Gerris [1]. As in previous 2D simulations [2], the dissipation due to breaking is found to be in good agreement with previous experimental observations and inertial-scaling arguments. For radii larger than the Hinze scale, the bubble size distribution is found to follow a power law of the radius, $r^{-3}$, and to scale linearly with the time dependent turbulent dissipation rate during the active breaking stages. The time-averaged bubble size distribution is found to follow the same power law of the radius and to scale linearly with the wave dissipation rate per unit length of breaking crest. We propose a phenomenological turbulent bubble break-up model that describes the numerical results and existing experimental results. [1] Popinet, S. 2003. Journal of Computational Physics 190, 572-. [2] Deike, L., Popinet, S., and Melville, W.K. 2015. Journal of Fluid Mechanics. vol 769, p541-569.

Simultaneous measurements of shape characteristics and radar backscattering of a water surface in a rain field, Ren Liu, Xinan Liu, James H. Duncan, University of Maryland, College Park — The characteristics of radar backscattering from a water surface that is stimulated by a rain field are studied at laboratory scale. The experiment is carried out in a 1.22-m by 1.22-m water pool with a water depth of 0.3 m. Simulated raindrops are generated by an array of 22-gauge needles that are attached to the bottom of a water reservoir located above the pool. A two-dimensional horizontal translational motion is added to the water reservoir in order to vary the drop impact location for each needle during each experimental run. A cinematic Laser-Induced-Florescence (LIF) technique is used to measure the water surface shape while radar backscattering from the water surface is continuously recorded by a dual-polarized, ultra-wide band radar. Both the radar return intensity and the water surface shape are measured for a range of rain rates and a range of radar incidence angles. The relationship between the geometric features of the water surface shape and the radar return are explored.

Simulation-based study of air-sea momentum fluxes nearshore, Xuanting Hao, Lian Shen, University of Minnesota — Momentum fluxes at sea surface are crucial to air-sea interactions. In nearshore regions, the bathymetry variation has a significant impact on the wave surface field and complicates the momentum fluxes at water surface. In this study, we extend a high order spectral method to address wave-bottom interactions and wave modeling. From the wave simulation data, we use the Hilbert-Huang transform to quantify the properties of the wave spectrum, based on which the wave field is reconstructed for the detailed mechanistic study of wind-wave interactions using large-eddy simulation for the wind field. The roughness of the water surface is quantified using a dynamic model for the effects of subgrid-scale waves. The results show that the waves are sensitive to the water depth variation. Associated with the changes in the wave field, the momentum fluxes at the air-sea interface increase in shallow regions.

Extending dispersive waves theory to use in semi-open systems, Lyubov Chumakova, University of Edinburgh — We study the performance of large ($O(100)$) wave energy converter (WEC) arrays that are used for ocean energy harvesting. We developed a fast computational algorithm based on the multiple scattering framework that is capable of handling large arrays of different configurations (general finite-size arrays, periodic arrays, periodic arrays of subarrays); for axisymmetric bodies the algorithm imposes no constraints on the body-size-to-wavelength ratio or on the inter-body spacings. Using this fast algorithm, we optimize the spatial configurations of arrays of different types and with increasing number of bodies (up to 400), with the goal of maximizing energy extraction. The results show that employing non-uniform spacings between the bodies in ordered and non-ordered arrays can increase the array gain several times. This holds for body resonant and near-resonant frequencies, as well as for the full spectrum cases. The optimal configurations are analyzed from a physical standpoint and compared to other structured arrays in physics. These results give a guideline on the possible future design of WEC arrays.
11:01 AM H31.00003 On the response of a water surface to a surface pressure source moving at trans-critical gravity-capillary wave speeds\(^1\), NAEEM MASNADI, University of Maryland, YEUN-WOO CHō, Korea Advanced Institute of Science and Technology (KAIST), JAMES H. DUNCAN, University of Maryland, TRIANTAPHYLLOS AKYLAS, Massachusetts Institute of Technology — The non-linear response of a water free surface to a pressure source moving at speeds near the minimum speed of linear gravity-capillary waves (\(C_{min} \approx 23 \text{ cm/s}\)) is investigated with experiments and theory. In the experiments, waves are generated by a vertically oriented air-jet that moves at a constant speed over the water surface in a long tank. The 3-D surface shape behind the air-jet is measured using a cinematic refraction-based technique combined with an LIF technique. At towing speeds just below \(C_{min}\), an unsteady pattern is formed where localized depressions periodically appear in pairs and move away from the source along the arms of a downstream V-shaped pattern. This behavior is analogous to the periodic shedding of solitary waves upstream of a source moving at the maximum wave speed in shallow water. The gravity-capillary depressions are rapidly damped by viscosity and their speed-amplitude characteristics closely match those from inviscid calculations of gravity-capillary lumps. The shedding frequency of the lumps in the present experiments increases with both increasing towing speed and air-flow rate. Predictions of this behavior using a model equation that incorporates damping and a quadratic nonlinearity are in good agreement with the experiments.

\(^1\)The partial support of the National Science Foundation under grant OCE0751853 is gratefully acknowledged.

11:14 AM H31.00004 Spatiotemporal measurement of surfactant distribution on gravity-capillary waves\(^1\), STEPHEN STRICKLAND, Department of physics, North Carolina State University, MICHAEL SHEARER, Department of mathematics, North Carolina State University, KAREN DANIELS, Department of physics, North Carolina State University — Materials adsorbed to the surface of a fluid - for instance, crude oil, biogenic slicks, or industrial/medical surfactants - will move in response to surface waves. Due to the difficulty of non-invasive measurement of the spatial distribution of a molecular monolayer, little is known about the dynamics that couple the surface waves and the evolving density field. We report measurements of the spatiotemporal dynamics of the density field of an insoluble surfactant driven by gravity-capillary waves in a shallow cylindrical container. Standing Faraday waves and traveling waves generated by the meniscus are superimposed to create a non-trivial surfactant density field. We measure both the height field of the surface using moire-imaging and the density field of the surfactant via the fluorescence of NBD-tagged phosphatidylcholine. Through phase-averaging stroboscopically-acquired images of the density field, we determine that the surfactant accumulates on the leading edge of the traveling waves generated by the meniscus and in the troughs of the standing Faraday waves. We fit the spatiotemporal variations in the two fields and report measurements of the wavenumbers as well as a temporal phase shift between the two fields. These measurements suggest that longitudinal waves contribute to the dynamics.

\(^1\)Funded by NSF grant DMS-0968258

11:27 AM H31.00005 Observation of Wood’s anomalies on surface gravity waves propagating on a channel\(^1\), CLAUDIO FALCÓN, ANDREA SCHMESSANE, Universidad de Chile — We report experiments demonstrating the appearance of Wood’s anomalies in surface gravity waves in a shallow water limit propagating along a channel with a submerged obstacles. Space-time measurements of surface gravity waves allows us to compute the stationary complex field of the wave and the amplitude growth of localized and propagative modes over all the channel including the scattering region. This allows us to access the near and far field dynamics, which constitute a new and complementary way of observation of mode resonances of the incoming wave displaying Wood’s anomalies. Transmission coefficient, dispersion relations and normalized wave energy of the incoming wave and the excited mode are measured and found to be in good agreement with theoretical predictions.

\(^1\)FONDECYT 1130354

11:40 AM H31.00006 ABSTRACT WITHDRAWN —

11:53 AM H31.00007 Efficient method for the computation of wave propagation in the atmosphere: horizontal rays and vertical normal modes, NOE LAHAYE, STEFAN LLEWELLYN SMITH, Department of Mechanical and Aerospace Engineering, University of California, San Diego — The development of efficient methods for computing the propagation of waves throughout the atmosphere is a longstanding issue. The widely-used WKBJ approximation is inaccurate when the typical scale of the fluid properties is of the order of the wave scale, or in particular regions such as turning points or critical levels. Homogeneity in the horizontal allows one to reduce the problem to an ODE (generally in the vertical) and solve this numerically with no further approximation. However, this may not be a valid approximation in applications; for example tsunami-generated acoustic-gravity waves have a large length scale and propagate over long distances up to the ionosphere. We propose a resolution method for 3D wave propagation that combines normal-modes and ray tracing, relying on scale separation between vertical and horizontal directions. This method has been widely used in the oceanic acoustic context and in waveguide theory, yet few applications in the atmospheric context seem to have been reported. First, we present some results in a simple framework (quiescent fluid, rigid boundary conditions), then show how the method may be adapted in the atmospheric context (including compressibility) to the propagation of waves emitted by a moving source and/or in a moving fluid.

12:06 PM H31.00008 Growth of gravity-capillary waves in countercurrent air/water turbulence\(^1\), ALFREDO SOLDATI, FRANCESCO ZONTA, Dept. of Elec. Manag. and Mechanical Engineering, University of Udine, Italy, MIGUEL ONORATO, Dept. of Physics, University of Torino, Italy — We use Direct Numerical Simulation (DNS) of the Navier Stokes equations to analyze the dynamics of the interface between air and water when both phases are driven by opposite pressure gradients (countercurrent configuration). The Reynolds number \(Re\), the Weber number \(We\) and the Froude number \(Fr\) fully describe the physical problem. We examine the problem of the transient growth of interface waves for different combinations of physical parameters. Keeping \(Re\), \(We\) and \(Fr\) constant and varying \(We\) and \(Fr\), we show that, in the initial stages of the wave generation process, the amplitude of the interface elevation \(\eta\) grows in time as \(\eta \propto t^{1/5}\). Wavenumber spectra, \(E(k_z)\), of the surface elevation in the capillary range are in good agreement with the prediction of the Wave Turbulence Theory. Finally, the wave-induced modification of the average wind and current velocity profiles will be addressed.

\(^1\)Support from Regione Autonoma Friuli Venezia Giulia under grant PAR FSC 2007/2013 is gratefully acknowledged.
Flexible Boundary

12:00PM H31.00003 Analysis of Nonlinear Internal Wave Systems Driven From a Flexible Boundary.

TOM DOBRA, ANDREW LAWRIE, University of Bristol, STUART DALZIEL, University of Cambridge — We present experiments and analysis of internal wave systems generated from motion of a flexible boundary driven by an array of electrical actuators. These actuators are independently controllable using arbitrary displacement profiles, and here we choose to coordinate their motion to create boundary displacements with a prescribed spectrum of temporal frequency and spatial wavenumber. We present first the simple case of two steady periodic signals of different frequency emanating from the same spatial location, and demonstrate a novel decomposition technique we have recently developed that isolates wave beams from their surroundings. This new methodology enables us to quantify weak nonlinearities in their interaction that are spatially local and aperiodic.

We then apply our methodology to the more complex case of convergent focussed wave-beams generated from a spatially distributed source.

12:30PM H31.00010 Solitary waves on a ferrofluid jet.

MARK BLYTH, EMILIAN PARAU, University of East Anglia — The propagation of axisymmetric solitary waves on the surface of an otherwise cylindrical ferrofluid jet subjected to a magnetic field is investigated. An azimuthal magnetic field is generated by an electric current flowing along a stationary metal rod which is mounted along the axis of the moving jet. A numerical method is used to compute fully-nonlinear travelling solitary waves and predictions of elevation waves and depression waves by Rannacher & Engel (2006) using a weakly-nonlinear theory are confirmed in the appropriate ranges of the magnetic Bond number. New nonlinear branches of solitary wave solutions are identified. As the Bond number is varied, the solitary wave profiles may approach a limiting configuration with a trapped toroidal-shaped bubble, or they may approach a static wave (i.e. one with zero phase speed). For a sufficiently large axial rod, the limiting profile may exhibit a cusp.

Monday, November 23, 2015 10:35AM - 12:45PM
Session H32 Drops: Elastic Surfaces and Fibers 313 - Jesse Belden, Naval Undersea Warfare Center

10:35AM H32.00001 Please comply: the water entry of soft spheres.

JEFF Belden, Naval Undersea Warfare Center, RANDY HURD, Utah State University, TATE FANNING, Brigham Young University, MICHAEL JANDRON, Naval Undersea Warfare Center, JOHN REKOS, University of South Florida, ALLAN BOWER, Brown University, TADD TRUSCOTT, Utah State University — The typical phenomena associated with sphere water impact are significantly altered when the sphere material is highly compliant rather than rigid. We describe the water entry of soft elastic spheres, which exhibit large oscillatory deformations throughout entry that carve nested disturbances into the normally smooth air cavity, altering cavity shape and pinch-off. Using an analytical model, we relate the maximum sphere deformation to the material properties and impact velocity. This characteristic deformation is used to reconcile the differences between cavities formed by compliant and rigid spheres. In addition to the nested disturbances seen with the homogeneous spheres, we observe azimuthal irregularities on the cavity during water entry of hollow elastic spheres. Based on experiments and finite-element modeling, we suggest that these disturbances are initiated by vibration mode shapes excited in the hollow spheres upon impact. For all sphere types, we compare the forces throughout water entry to the rigid sphere case.

10:45AM H32.00002 Capillary-driven folding of a thin floating annular film.

JOSEPH D. PAULSEN, Syracuse University, VINCENT DEMERY, PCT-ESPCI, France, K. BUGRA TOGA, Eastman Chemical Company, ZHANLONG QIU, BENNY DAVIDOVITCH, THOMAS P. RUSSELL, NARAYANAN MENON, Univ of Mass - Amherst — A thin elastic sheet that is compressed on a substrate will form wrinkles to gather excess material while conforming to its foundation. When compressed further, the sheet may form folds, which can be understood as minimizing the sum of the bending energy and the energy to deform the substrate.

Here we demonstrate in a simple planar geometry a folding transition that is independent of the mechanical properties of the sheet. We study the deformations of a thin polymer film that is cut into an annular shape and floated onto a flat air-water interface. By increasing the concentration of surfactant outside the film, we reduce the surface tension that pulls on the outer boundary of the annulus. The larger, inward tension causes the film to wrinkle and fold. Folding occurs at a threshold ratio of inner to outer tension that depends on the geometry of the sheet, but is independent of its bending rigidity. Our results are consistent with the simple geometric principle that the sheet assumes the unstretched shape that minimizes the interfacial energy of the exposed liquid surface. A similar principle was found to control how a thin elastic sheet wraps a liquid drop.

2. Paulsen et al., BAPS.2015.MAR.G34.4

11:01AM H32.00003 Drops spreading on flexible fibers.

KATARZYNA SOMSZOR, FRANCOIS BOULGNE, Princeton University, ALBAIN SAURET, SVI - CNRS, EMILIE DRESSAIRE, NYU Polytechnic School of Engineering, HOWARD STONE, Princeton University — Fibrous media are encountered in many engineered systems such as textile, paper and insulating materials. In most of these materials, fibers are randomly oriented and form a complex network in which drops of wetting liquid tend to accumulate at the nodes of the network. Here we investigate the role of the fiber flexiblity on the spreading of a small volume of liquid on a pair of crossed flexible fibers. A droplet of silicone oil is dispensed at the point of contact of the fibers and we characterize the liquid morphologies as we vary the volume of liquid, the angle between the fibers, and the length and bending modulus of the fibers. Drop morphologies previously reported for rigid fibers, i.e. a drop, a column and a mixed morphology, are also observed on flexible fibers with modified domains of existence. Moreover, at small inclination angles of the fibers, a new behavior is observed: the fibers bend and collapse. Depending on the volume, the liquid can adopt a column or a mixed morphology on the collapsed fibers. We rationalize our observations with a model based on energetic considerations. Our study suggests that the fiber flexibility adds a rich variety of behaviors that can be crucial for industrial applications.

11:14AM H32.00004 Self-Propelled Droplet Removal from Hydrophobic Fiber-Based Coalescers.

KUNGANG ZHANG, FANGJIE LIU, ADAM WILLIAMS, XIAOPENG QU, Duke University, JAMES FENG, University of British Columbia, CHUAN-HUA CHEN, Duke University — Fiber-based coalescers are widely used to accumulate droplets from aerosols and emulsions, where the accumulated droplets are typically removed by gravity or shear. We report self-propelled removal of drops from a hydrophobic fiber, where the surface energy released upon drop coalescence overcomes the drop-fiber adhesion, producing spontaneous departure that would not occur on a flat substrate of the same contact angle. The self-removal takes place above a threshold drop-to-fiber radius ratio, and the departure speed is close to the capillary-inertial velocity at large radius ratios. [K. Zhang et al., Phys. Rev. Lett., in press.]

11:27AM H32.00005 Liquid Droplets on a Highly Deformable Membrane.

RAFAEL SCHULMAN, KARI DALNOKI-VERESS, McMaster University — We present measurements of the deformation produced by micro-droplets atop thin elastomeric and glassy free-standing films. Due to the Laplace pressure, the droplets deform the elastic membrane thereby forming a bulge. Thus, there are two angles that define the droplet/membrane geometry: the angle the liquid surface makes with the film and the angle the deformed bulge makes with the film. The contact line geometry is well captured by a Neumann construction which includes contributions from interfacial and mechanical tensions. Finally, we show that a droplet atop a film with biaxial tension assumes an equilibrium shape which is elongated along the axis of high tension.

1. Paulsen et al., BAPS.2015.MAR.G34.4
Sophisticated compound droplets on fiber networks, FLORIANE WEYER, MARJORIE LISMONT, LAURENT DREESEEN, NICOLAS VANDEWALLE, University of Liege — Droplets on fibers are part of our everyday lives. Indeed, many phenomena involve drops and fibers such as the formation of dew droplets on a spiderweb, the trapping of water droplets on cactus spines or the dyeing of cotton or wool fibers. Therefore, this topic has been widely studied in the recent years and it appears that droplets on fibers can be the starting point for an open digital microfluidics. We study the behavior of soapy water droplets on a fiber array. We use high-speed videography to characterize the dynamics of drops impacting fibers. Our systematical experiments reveal that the outcome of droplet collision critically depends on the relative magnitude of inertial to capillary forces and the ratio of the thickness of fiber to the diameter of the drop. We identify three outcomes of the collision using a non-dimensional regime map. The selection among the modes of single capturing, single division and divided drop falling is explained through a scale analysis of forces. We also examine the droplet retention on the fiber after impact. For each mode, we suggest the mathematical models to predict the amount of residual water on the fiber. Our study can be extended to predicting the remaining droplet, the critical problem in air filtration, water collection, and fiber coating.

This work was supported by the National Research Foundation of Korea(NRF) grant funded by the Korea government(MSIP) (2009-0083510).

Elastocapillary mist collector, CAMILLE DUPRAT, ROMAIN LABB, ANA REWAKOWICZ, LaHyX, Ecole polytechnique — Fibrous media are commonly used to collect droplets from an aerosol. In particular, woven textiles are used to harvest fresh water from fog, and coalescing filters made of non-woven entangled fibers are used to extract oil drops from gas streams. We propose a novel mist collector made of a forest of vertical flexible threads. As the droplets accumulate on the fibers, capillary bridges are formed, leading to the collapse of adjacent fibers and thus forming liquid columns. This improve the liquid collection by preventing clogging, enabling high capture and precluding re-entrainment of drops in the gas stream due to the immediate coalescence of incoming droplets, and promoting fast drainage. We find that the collection flow rate is constant and can be adjusted by varying the fibers arrangement and flexibility. We show that there is an optimal situation for which this collection rate, i.e. the global efficiency, is maximal due to an elastocapillary clogging, enabling high capture and precluding re-entrainment of drops in the gas stream due to the immediate coalescence of incoming droplets, and promoting fast drainage. We find that the collection flow rate is constant and can be adjusted by varying the fibers arrangement and flexibility. We show that there is an optimal situation for which this collection rate, i.e. the global efficiency, is maximal due to a liquid column formation, which conveniently marks the original diameter of the air-disc. This ring of bubbles arises owing to multiple initial contacts just before the first contact of a drop hitting a solid surface, does not occur at a point but along a ring owing to viscous lubrication pressure in the intervening air layer. This always leads to the entrapment of a small bubble under the center of the drop. The nature of the actual first contact is affected by the roughness of the solid. We use ultra-high-speed imaging, with 200 ns time resolution, to observe the structure of this first contact between the liquid and a smooth solid surface. For a water drop impacting onto regular micro-scope glass slide we observe a ring of micro-bubbles as observed by Thoroddsen et al.1 which conveniently marks the original diameter of the air-disc. This ring of bubbles arises owing to multiple initial contacts just before the formation of the fully wetted outer section. These contacts are spaced by a few microns and quickly grow in size until they meet each other, entrappping the bubbles. We thereby conclude that the localized contacts are due to nanometric roughness of the glass surface and the presence of the micro-bubbles can therefore distinguish between glass with 10 nm roughness from perfectly smooth glass.

1Thoroddsen et al., The air-bubble entrapped under a drop impacting on a solid surface. J. Fluid Mech. 545, 203-212.
patches and finally entrap bubbles when the wetted patches meet the advancing contact line, through topological change. Isolate bubbles are
superhydrophobic surfaces to those above the Leidenfrost temperature. Finally, we present a physical model that rationalizes our results.

At sufficient velocity, a drop impacting a junction might be expected to breakup into smaller droplets; yet it is unclear how many droplets would
be produced and the time for these droplets to clear the surface. Here, we show that the number droplets and overall residence time depends on
Ohnesorge numbers higher than unity.

Viscous drops exhibit a "prompt tumbling-rebound" behavior. As such, viscous glycerol drops surprisingly rebound faster than three orders of magnitude
less viscous water drops. This is made possible by a small conversion of translational to rotational kinetic energy, at non-axisymmetric impact
conditions, as also confirmed by additional Lattice Boltzmann simulations: a rapid transition of the internal angular velocity prior to rebound to
a constant value, as in a tumbling solid body, promotes a rapid rebound of more viscous drops, which are capable to rebound without recoiling.

By studying drop impact dynamics, we explore the drop behavior in contactless and frictionless conditions, and identify the Ohnesorge number
as the primary parameter to predict the transition between different impact regimes on tilted sublimating slopes, with tumbling observed for
Ohnesorge numbers higher than unity.

Reducing the residence time of a bouncing drop with spiked macrotexture , COLIN PATTERSON, GE Aviation; Boston University, JAMES BIRD. Boston University — Liquid drops can bounce when they impact non-wetting surfaces. Recently, studies have demonstrated that the time that the bouncing drop resides at the surface can be adjusted with the presence of ridged macrotextures. When non-parallel macrotextures are present, they intersect to create spiked junctions.

Spontaneous removal of droplets from surfaces is of significant importance in nature and many technologies, e.g., self-cleaning surfaces. Despite progress, the understanding of phenomena leading to such behavior continues to be a challenge. We show how water droplets in contact with superhydrophobic surfaces in a low-pressure environment can self-remove through sudden spontaneous launching and subsequent repeated bouncing behavior. We demonstrate that this bouncing results from the combined effect of droplet vaporization, vapor flow in the surface texture, and substrate adhesion leading to a forced, underdamped, mass-spring-damper system behavior. This work is a step toward understanding inherent physical phenomena of droplet-surface interactions manifesting themselves at conditions promoting vaporization, e.g., low-pressure environments, and shows how surface texture design aware of such phenomena alone, can prohibit water retention on surfaces.

Bubble entrapment under the impact of a viscous drops onto a solid surface, K. LANGLEY, E.Q. LI, S.T. THORODDSEN. King Abdullah University of Science and Technology — When a high-speed viscous drop impacts onto a solid surface, the lubrication pressure leads to the entrapment of an air-disc which contracts into an isolated central bubble. The maximum disc-diameter is marked by microbubbles, followed by a fully wetted outer region. However, the outer edge of this wetted region forms a lamellar jet traveling along the surface, which entraps a myriad of micro-bubbles. The tip of the spreading lamella separates from the solid surface and levitates on a lubricating air layer. The local contacts between the levitated sheet and the solid surface form wetted patches and finally entrap bubbles when the wetted patches meet the advancing contact line, through topological change. Isolate bubbles are also entrained through the advancing contact line. These bubble entrapment mechanisms are investigated with high-speed video imaging and are found to be highly dependent on the drop viscosity and surface properties. We use high-speed interferometry to measure the thickness of the air-layer under the advancing lamella.

Water drop dynamics on a granular layer, CORALINE LLORENS, ANNE- LAURE BIANCE, CHRISTOPHE YBERT, CHRISTOPHE PIRAT. ILM UMR5306 Universite Lyon1, LIQUIDS AND INTERFACES TEAM — Liquid drop impacts, either on a solid surface or a liquid bath, have been studied for a while and are still subject of intense research. Less is known concerning impacts on granular layers that are shown to exhibit an intermediate situation between solid and liquid. In this study, we focus on water drop impacts on granular matter made of micrometer-sized spherical glass beads. In particular, we investigate the overall dynamics arising from the interplay between liquid and grains throughout the impact. Depending on the relevant parameters (impact velocity, drop and grain sizes, as well as their wetting properties), various behaviors are evidenced. In particular, the behavior of the beads at the liquid-gas interface (ball-bearing vs imbibition) is shown to greatly affect the spreading dynamics of the drop, as well as satellite droplets formation, beads ejection, and the final crater morphology.
An Experimental Investigation of Water Droplets onto Superhydrophobic Surfaces Pertinent to Aircraft Icing Phenomena¹. 

HAIXING LI, RYE WALDMAN, HUI HU, Iowa State University — Superhydrophobic surfaces have self-cleaning properties that make them promising candidates as anti-icing solutions for various engineering applications, including aircraft anti-/de-icing. However, under sufficient external pressure, the liquid water on the surface can transition to a wetted state, defeating the self-cleaning properties of superhydrophobic surfaces. In the present study, an experimental investigation was conducted to quantify the transient behavior of water droplets impinging onto test surfaces with different hydrophobicity properties under different environmental icing conditions. The experiments were performed in the Icing Research Tunnel of Iowa State University (IRT-ISU) with a NACA0012 airfoil. In addition to using a high-speed imaging system to reveal transient behavior of water droplets impinging onto test surfaces with different hydrophobicity properties, an IR thermometry was also used to quantify the unsteady heat transfer and dynamic phase changing process within the water droplets after impingement onto the test plates with different frozen cold temperatures. The high-speed imaging results were correlated with the quantitatively temperature measurements to elucidate underlying physics in order to gain further insight into the underlying physics pertinent to aircraft icing phenomena.

¹The research work is partially supported by NASA with grant number NNX12AC21A and National Science Foundation under award numbers CBET-1064196 and CBET-1435590

Ballroom B - Chihiro Inoue, University of Tokyo

10:35AM H35.00001 Visco-capillarity in Sparkling Fireworks. CHIHIRO INOUE, The University of Tokyo, EMMANUEL VILLERMAUX, IRPHE, UTOKYO TEAM, IRPHE TEAM — A unique toy firework called sparkling fireworks is popular in Japan for 400 years, but the physics behind the beauty remains a hidden mystery. Sparkling fireworks are made by a twisted paper simply wrapping 0.1g of black powder at the lower end. Ignited there, the powder melts in a fireball of molten salts, and streaks of light are ejected. The beautiful fragile streaks are visible from the black body radiation of the hot surface of the ejected droplets. The droplets suddenly fragment up to ten times successively and their light streaks traces are like pine needles. We have already clarified why the droplets are ejected through the bursting of a gas bubble on the mother fireball, leading to successive fragmentations by micro explosions. To quantify phenomenon, we measure the diameter and the ejection velocity of the droplets. It is found that not only inertia and capillarity of the liquid matter, but also its viscosity is important (the Ohnesorge number is about 0.1). The droplets ejection velocity is determined by the liquid surface tension and viscosity, and separate from the mother drop on a visco-capillarity time scale.

10:48AM H35.00002 Experimental study of combustion of decane, dodecane and hexadecane with polymeric and nano-particle additives. MOHSEN GHAMARI, ALBERT RATNER, University of Tokyo — Recent studies have shown that adding combustible nano-particles could have promising effects on increasing burning rate of liquid fuels. Combustible nano-particles could enhance the heat conduction and mixing within the droplet. Polymers have also higher burning rate than regular hydrocarbon fuels because of having the flame closer to the droplet surface. Therefore adding polymeric additive could have the potential to increase the burning rate. In this study, combustion of stationary fuel droplets of n-Decane, n-Dodecane and n-Hexadecane doped with different percentages of a long chain polymer and also a very fine nano carbon was examined and compared with the pure hydrocarbon behavior. In contrast with hydrocarbon droplets with no polymer addition, several zones of combustion including a slow and steady burning zone, a strong swelling zone and a final fast and fairly steady combustion zone were also detected. In addition, increasing polymer percentage resulted in a more extended swelling zone and shorter slow burning zone in addition to a shorter total burning time. Addition of nano-particles also resulted in an overall increased burning rate and shortened burning time which is due to enhanced heat conduction within the droplet.

11:01AM H35.00003 Fragment structure from vapor explosions during the impact of molten metal droplets into a liquid pool. NADIA KOURAYTEM, ER QIANG LI, IVAN URIEV VAKARELSKI, SIGURDUR THORODDSEN, King Abdullah University of Science and Technology — High-speed video imaging is used in order to look at the impact of a molten metal droplet falling into a liquid pool. The interaction regimes are three: film boiling, nucleate boiling or vapor explosion. Following the vapor explosion, the metal fragments and different textures are observed. It was seen that, using a tin alloy, a porous structure results whereas using a distinctive eutectic metal, Field’s metal, micro beads are formed. Different parameters such as the metal type, molten metal temperature, pool surface tension and pool boiling temperature have been altered in order to assess the role they play on the explosion dynamics and the molten metal’s byproduct.

11:14AM H35.00004 High Frequency Stable Oscillate boiling. FENFANG LI, SILVESTRE ROBERTO GONZALEZ-AVILA, CLAUS DIETER OHL, Nanyang Tech Univ — We present an unexpected regime of resonant bubble oscillations on a thin metal film submerged in water, which is continuously heated with a focused CW laser. The oscillatory bubble dynamics reveals a remarkably stable frequency of several 100 kHz and is resolved from the side using video recordings at 1 million frames per second. The emitted sound is measured simultaneously and shows higher harmonics. Once the laser is switched on the water in contact with the metal layer is superheated and an explosively expanding cavitation bubble is generated. However, after the collapse a microbubble is nucleated from the bubble remains which displays long lasting oscillations. Generally, pinch-off from of the upper part of the microbubble is observed generating a continuous stream of small gas bubbles rising upwards. The cavitation expansion, collapse, and the jetting of gas bubbles are detected by the hydrophone and are correlated to the high speed video. We find the bubble oscillation frequency is dependent on the bubble size and surface tension. A preliminary model based on Marangoni flow and heat transfer can explain the high flow velocities observed, yet the origin of bubble oscillation is currently not well understood.

11:27AM H35.00005 Pool Boiling Enhancement on Textured Surfaces using Acoustic Actuation. THOMAS BOZIUK, MARC SMITH, ARI GLEZER, Georgia Institute of Technology — Boiling heat transfer on submerged textured heated surfaces is enhanced using ultrasound actuation. The heated surface is textured using an array of open microchannels that advantageously separate the nucleation sites on the surface and inhibit the transition to film boiling, which significantly increases the critical heat flux compared to a smooth surface of the same planform dimensions. The present investigation shows that the formation and evolution of vapor bubbles on the heated surface can be substantially altered by a highly directional ultrasound (1.7 MHz) beam, and leads to significant enhancement in heat transfer, including reduced surface superheat and increased critical heat flux (exceeding 55%). The effects of the beam incidence and azimuthal angle on vapor formation, advection, and resulting effect on surface superheat are investigated experimentally in a liquid test cell. Heat transfer enhancement characterized by changes in the boiling curve (i.e., superheat and CHF) varies with surface texturing and is also dependent on acoustic beam orientation relative to the surface texture pattern.
11:40 AM H35.00006 Cooling of hot bubbles by surface texture during the boiling crisis\(^1\). NAVDEEP DHILLON, JACOPO BUONGIORNO, KRIPA VARANASI, Massachusetts Inst of Tech-MIT — We report the existence of maxima in critical heat flux (CHF) enhancement for pool boiling on textured hydrophilic surfaces and reveal the interaction mechanism between bubbles and surface texture that governs the boiling crisis phenomenon. Boiling is a process of fundamental importance in many engineering and industrial applications but the maximum heat flux that can be absorbed by the boiling liquid (or CHF) is limited by the boiling crisis. Enhancing the CHF of industrial boilers by surface texturing can lead to substantial energy savings and reduction in greenhouse gas emissions on a global scale. However, the fundamental mechanisms behind this enhancement are not well understood, with some previous studies indicating that CHF should increase monotonically with increasing texture density. However, using imbibition experiments on a parametrically designed set of plain and nano-textured micropillar surfaces, we show that there is an optimum intermediate texture density that maximizes CHF and further that the length scale of this texture is of fundamental significance. Using imbibition experiments and high-speed optical and infrared imaging, we reveal the fundamental mechanisms governing the CHF enhancement maxima in boiling crisis.

\(^1\)We acknowledge funding from the Chevron corporation

11:53 AM H35.00007 Verification and validation of a GPU-based multi-resolution direct numerical simulation of multiphase flow with phase change. CHRISTOPHER J. FORSTER, MARC K. SMITH, Georgia Institute of Technology — Nucleate boiling processes span a large range of length scales, ranging from $O(1 \mu m)$ to $O(1 m)$, that arise due to steep gradients in the density and temperature near the liquid-vapor interface and the contact line. The Wavelet Adaptive Multiresolution Representation (WAMR) method is a general and robust technique for providing grid adaptivity around sharp features in the solutions of partial differential equations and is capable of resolving the large disparity of length scales present in nucleate boiling. A new flow solver based on the WAMR method and specifically parallelized for a GPU computing architecture has been developed. The compressible formulation of the Navier-Stokes equations is used in the flow solver and a preconditioned dual time-stepping integration scheme provides accurate solutions for flows approaching the incompressible limit. The WAMR method provides a direct measure of the local error over the entire grid and coupling spatial error control with adaptive time integration allows for a priori control of the error in the solution. Verification and validation cases, including single vapor bubble growth, will be presented to demonstrate the efficiency and accuracy of the method for solving nucleate boiling problems.

12:06 PM H35.00008 Sliding bubbles on a hot horizontal wire in a subcooled bath\(^1\). ALEXIS DUCHESNE, CHARLES DUBOIS, HERVÉ CAPS, GRASP- Université de Liège — When a wire is heated up to the boiling point in a liquid bath some bubbles will nucleate on the wire surface. Traditional nucleate boiling theory predicts that bubbles generate from active nucleate site, grow up and depart from the heating surface due to buoyancy and inertia. However, an alternative scenario is presented in the literature for a subcooled bath: bubbles slide along the horizontal wire before departing. New experiments were performed by using a constantan wire and different liquids, varying the injected power. Silicone oil, water and even liquid nitrogen were tested in order to vary wetting conditions, liquid viscosities and surface tensions. We explored the influence of the wire diameter and of the subcooled bath temperature. We observed, of course, sliding motion, but also a wide range of behaviors from bubbles clustering to film boiling. We noticed that bubbles could change moving sense, especially when encountering with another bubble. The bubble speed is carefully measured and can reach more than 100 mm/s for a millimetric bubble. We investigated the dependence of the speed on the different parameters and found that this speed is, for a given configuration, quite independent of the injected power. We understand these phenomena in terms of Marangoni effects.

\(^1\)This project has been financially supported by ARC SuperCool contract of the University of Liège

Monday, November 23, 2015 10:35AM - 12:45PM — Session H36 Bubbles: Micro-bubbles and Nano-bubbles Ballroom C - Jose Gordillo, Universidad de Sevilla

10:35 AM H36.00001 Contact line pinning favors the mass production of monodisperse microbubbles. JOSE MANUEL GORDILLO, FRANCISCO CAMPO-CORTES, GUILLAUME RIBOUX, Escuela Superior de Ingenieros, Universidad de Sevilla, Spain — A robust method for the generation of phospholipid covered monodisperse microbubbles of diameters $\sim$10 microns at production rates exceeding 0.1 MHz, is presented here. We show that bubbles are periodically formed from the tip of a long and thin gas ligament stabilized thanks to both the strong favorable pressure gradient existing at the entrance region of a long rectangular PDMS-PDMS channel and to the pinning of the gas-liquid interface at a centered groove of several microns width placed on one of its walls. Moreover, the long exit channel incorporated in our design, favors the transport of phospholipid molecules towards the gas-liquid interface. Our experiments show that the resulting phospholipid shell inhibit both the diffusion of the gas in the surrounding liquid as well as the coalescence between contacting bubbles. These evidences indicate that the proposed method is suitable for the generation of monodisperse microbubbles for diagnosis or therapeutic applications.

10:48 AM H36.00002 Pressure gradient induced generation of microbubbles. ALVARO EVANGELIO, FRANCISCO CAMPO-CORTES, JOSE MANUEL GORDILLO, Escuela Superior de Ingenieros, Universidad de Sevilla, Spain — It is well known that the controlled production of monodisperse bubbles possesses untouchable applications in medicine, pharmacy and industry. Here we provide with a detailed physical description of the bubble formation processes taking place in a type of flow where the liquid pressure gradient can be straightforwardly controlled. In our experiments, a gas flow rate discharges through a cylindrical needle into a pressurized chamber. The pressure gradient created from the exit of the injection needle towards the entrance of a extraction duct promotes the stretching of the gas ligament downstream. In our analysis, which is supported by an exhaustive experimental study in which the liquid viscosity is varied by three orders of magnitude, different regimes can be distinguished depending mainly on the Reynolds number. Through our physical modeling, we provide closed expressions for both the bubbling frequencies and for the bubble diameters as well as the conditions under which a monodisperse generation is obtained in all regimes found. The excellent agreement between our expressions and the experimental data fully validates our physical modeling.
11:01AM H36.00003 Surface elastic waves on a viscoelastic boundary generated by an oscillating microbubble. MARC TINGUELY, MATTHEW HENNESSY, ANGELO POMMELLA, OMAR MATAR, VALERIA GARBIN, Imperial College London — Acoustically-driven microbubbles are used as contrast agents for ultrasound medical imaging, or to enhance the uptake of molecules by cells in drug delivery. For both applications, the microbubbles oscillate near soft interfaces, whose viscoelastic properties vary depending on neighbouring tissues. The effect of these properties on the deformation of the boundary, and on the stresses generated by the microbubbles, is still poorly understood. We use high-speed video microscopy to investigate the deformation of a viscoelastic boundary generated by an oscillating microbubble via tracking the displacement of embedded particles. We observe the propagation of surface elastic waves (Rayleigh waves) whose velocity of propagation, phase shift, and particle trajectories depend on the viscoelastic properties of the boundary. We develop a Kelvin-Voigt viscoelastic model to predict the deformation of the gels. The results of the model are in good agreement with the experimental observations, which permits the estimation of the magnitude of the stresses generated on the surface of the gel by the oscillating bubble.

1 National Swiss Foundation (MT), and EPSRC: Programme Grant EP/K003976/1, and EP/L022176/1

11:14AM H36.00004 Nano bubble migration in a tapered conduit in the asymptotic limits of zero capillary and Bond Numbers - Theory and Experiments. MICHAEL NORTON, University of Pennsylvania, FRANCES ROSS, IBM Watson Research Center, HAIM BAU, University of Pennsylvania — Using a hermetically sealed liquid cell, we observed the growth and migration of bubbles (tens of hundreds of nanometers in diameter) in a tapered conduit and supersaturated solution with a transmission electron microscope. To better understand bubble shape and migration dynamics, we developed simple 2D and 3D models valid in the limit of zero capillary and Bond numbers. The 3D model is restricted to small taper slope, weakly non-circular contact line geometries and large bubble aspect ratio (high confinement), and was solved using a pseudo-spectral decomposition. Both models utilize the Blake-Haynes mechanism to relate dynamic contact angle to local contact line velocity. The influence of pinning of a portion of the contact line on bubble geometry was also shown. Our 2D and 3D models predict growth rates in agreement with experimental observations, but several orders of magnitude lower than predicted by the classical Epstein – Plesset theory.

1 The work was supported, in part, by NSF CBET grant 1066573.

11:27AM H36.00005 Stability of surface nanobubbles, SHANTANU MAHESHWARI, MARTIN VAN DER HOEF, Physics of Fluids, University of Twente, P.O. Box 217, 7500 AE, Enschede, The Netherlands, XUEHUA ZHANG, School of Civil, Environmental and Chemical Engineering, RMIT University, Melbourne, VIC 3001, Australia, DETLEF LOHSE, Physics of Fluids, University of Twente, P.O. Box 217, 7500 AE, Enschede, The Netherlands — We have studied the stability and dissolution of surface nanobubbles on the chemical heterogenous surface by performing Molecular Dynamics (MD) simulations of binary mixture consists of Lennard-Jones (LJ) particles. Recently our group has derived the exact expression for equilibrium contact angle of surface nanobubbles as a function of oversaturation of the gas concentration in bulk liquid and the lateral length of bubble. It has been shown that the contact line pinning and the oversaturation of gas concentration in bulk liquid is crucial in the stability of surface nanobubbles. Our simulations showed that how pinning of the three-phase contact line on bubble heterogenous surface lead to the stability of the nanobubble. We have calculated the equilibrium contact angle by varying the gas concentration in bulk liquid and the lateral length of the bubble. Our results showed that the equilibrium contact angle follows the expression derived analytically by our group. We have also studied the bubble dissolution dynamics and showed the “stick-jump” mechanism which was also observed experimentally in case of dissolution of nanodrops.

11:40AM H36.00006 Distinguishing between microscale gaseous bubbles and liquid drops, BENG HAU TAN, HONGJIE AN, CHON U CHAN, CLAUS-DIETER OHL, Nanyang Technological University — In recent years, there has been strong research interest in decorating surfaces with tiny bubbles and drops due to their potential applications in reducing slippage in micro and nanofluidic devices. Both nanobubbles and nanodrops are typically nucleated by exchanging fluids over a suitable substrate. However, the nucleation experiments present many challenges, such as reproducibility and the possibility of contamination. The use of one-use plastic syringes and needle cannulas in nucleation experiments can introduce polymeric contamination. A contaminated experiment may nucleate bubbles, drops or both. Moreover, it is surprisingly difficult to distinguish between bubbles and drops under the usual atomic force microscopy or optical techniques. Here we present an experimental study comparing bubbles and oil (PDMS) drops on an atomically smooth surface (HOPG). Instead of nucleating the objects via solvent exchange, we directly introduced bubbles via electrolysis, and oil drops by injecting a dilute solution. Contrary to previous reports, we find that under careful AFM characterisation, liquid drops and gaseous bubbles respond differently to a change in imaging force, and moreover present different characteristic force curves.

11:53AM H36.00007 Analysis of cavitation effect for water purifier using electrolysis, DONG HO SHIN, HAN SEO KO, SEUNG HO LEE, Sungkyunkwan University — Water is a limited and vital resource, so it should not be wasted by pollution. A development of new water purification technology is urgent nowadays since the original and biological treatments are not sufficient. The microbubble-aided method was investigated for removal of algal in this study since it overcomes demerits of the existing purification technologies. Thus, the cavitation effect in a venturi-type tube using the electrolysis was analyzed. Ruthenium-coated titanium plates were used as electrodes. Optimum electrode interval and applied power were determined for the electrolysis. Then, the optimized electrodes were installed in the venturi-type tube for generating cavitation. The cavitation effect could be enhanced without any byproduct by the bubbly flow induced by the electrolysis. The optimum mass flow rate and current were determined for the cavitation with the electrolysis. Finally, the visualization techniques were used to count the cell number of algal and microbubbles for the confirmation of the performance. As a result, the energy saving and high efficient water purifier was fabricated in this study.

1 This work was supported by the Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Korean government (MEST) (No. 2013R1A2A2A01068653).

12:06PM H36.00008 Surface nanobubble nucleation dynamics during water-ethanol exchange, CHON U CHAN, CLAUS-DIETER OHL, Nanyang Technological University, Singapore — Water-ethanol exchange has been a promising nucleation method for surface attached nanobubbles since their discovery. In this process, water and ethanol displace each other sequentially on a substrate. As the gas solubility is 36 times higher in ethanol than water, it was suggested that the exchange process leads to transient supersaturation and is responsible for the nanobubble nucleation. In this work, we visualize the nucleation dynamics by controllably mixing water and ethanol. It depicts the temporal evolution of the conventional exchange in a single field of view, detailing the conditions for surface nanobubble nucleation and the flow field that influences their spatial organization. This technique can also pattern surface nanobubbles with variable size distribution.
dependence is found for the skin friction due to the weak development of near-wall turbulence. To appear after reattachment, and to be a challenging phenomenon for turbulence theories and models. In the latter region, a significant reduction slightly in the present DNSs. At all three Reynolds numbers. This extends the work of Abe et al. (CTR Annual Brief, 2012) but removes the stagnation point, present over the bubble.

layer separating from a flat plate and reattaching have been performed with inlet data generated by rescaling-recycling at injection in thin liquid film layers designed to separate NH$_3$ from NH$_3$-H$_2$O solutions. The transport phenomena exhibited by the microbubbles helps them separate volatile liquids effectively with negligible sensible heat transfer. The process is nearly isothermal simply because evaporation by microbubbles is controlled by internal mixing, which is fast relative to sensible heat transfer, when limited to short contact times in thin films. A 1000-3000-fold increase in mass transfer, over conventional striping, and a 100% separation efficiency achieved in a processing time of 30 minutes is observed, potentially, if persisting with industrial scale up, resulting in a 200-fold reduction in processing time.

There are some strong evidences that this mechanism may be generalized to axisymmetric bodies with base cavity. Elliptical toric recirculation. The stabilization can be modeled with a Langevin equation. The plausible mechanism for drag reduction with the sufficient deep cavity is able to stabilize the wake toward a symmetry preserved wake, thus suppressing the RSB modes and leading to a weaker stresses along the lateral wall create a novel vortical structure similar to a horseshoe vortex. A vortex along the top wall appears at Re = 400. As the Reynolds number increases, the top vortex increases both in spanwise and streamwise length and stunts the growth of the primary vortex. The downstream motion of the top vortex releases frictional stress thus destroying some critical points upstream while creating new vortices downstream in the channel with same and different sizes. By varying the size of the check valve, we study the effects of channel ratio between the check valve and channel on the dynamics of bubble-driven liquid circulation and seek for an optimal channel ratio to support experimental design.

This research is supported by NSF Collaborative Research (1264739).

Monday, November 23, 2015 10:35AM - 12:45PM — Session H37 Separated Flows Sheraton Back Bay A - Anastasios Liakos, United States Naval Academy

10:35AM H37.00001 Topological study of steady state, three dimensional flow over a backward facing step 1, ANASTASIOS LIAKOS, United States Naval Academy, NIKOLAOS MALAMATARIS, George Mason University — The topology and evolution of flow over a backward facing step in three dimensional channel flow is examined for low to moderate Reynolds numbers. Direct numerical simulations were performed via a home made parallel finite element code. The computational domain has been designed according to actual laboratory experimental conditions. Analysis of the results is performed using the three dimensional theory of separation. Results indicate that a (primary) vortex is present for all Reynolds numbers immediately downstream from the step. Frictional stresses along the lateral wall create a novel vortical structure similar to a horseshoe vortex. A vortex along the top wall appears at Re = 400. As the Reynolds number increases, the top vortex increases both in spanwise and streamwise length and stunts the growth of the primary vortex. The downstream motion of the top vortex releases frictional stress thus destroying some critical points upstream while creating new ones downstream. Finally, at Re = 900 and 950, the primary and top vortices are twisting severely, which may indicate the onset of instability.

1 Financial support from ONRG-VSP, Grant Number N62909-14-1-V068 is acknowledged

10:48AM H37.00002 Fluid force and static symmetry breaking modes of 3D bluff bodies. OLIVIER CADOT, ANTOINE EVRARD, ENSTA-ParisTech, DFA TEAM — A cavity at the base of the squareback Ahmed model at Re = 6.10$^5$ is able to reduce the base suction by 18% and the drag coefficient by 9%, while the flow at the separation remains unaffected. Instantaneous pressure measurements at the body base, fluid force measurements and wake velocity measurements are investigated varying the cavity depth from 0 to 35% of the base height. Due to the reflectional symmetry of the rectangular base, there are two Reflectional Symmetry Breaking (RSB) mirror modes present in the natural wake that switch from one to the other randomly in accordance with the recent findings of Grandemange et al. (2013). It is shown that these modes exhibit an energetic 3D static vortex system close to the base of the body. A sufficiently deep cavity is able to stabilize the wake toward a symmetry preserved wake, thus suppressing the RSB modes and leading to a weaker elliptical toric recirculation. The stabilization can be modeled with a Langevin equation. The plausible mechanism for drag reduction with the base cavity is based on the interaction of the static 3D vortex system of the RSB modes with the base and their suppression by stabilization. There are some strong evidences that this mechanism may be generalized to axisymmetric bodies with base cavity.

11:01AM H37.00003 Direct numerical simulations of a turbulent separation bubble over a wide Reynolds-number range. HIROYUKI ABE, YASUHIRO MIZOBUCHI, YUICHI MATSUO, Japan Aerospace Exploration Agency, PHILIPPE R. SPALART, Boeing Commercial Airlines — Direct numerical simulations (DNSs) of a turbulent boundary layer separating from a flat plate and reattachment have been performed with inlet data generated by rescaling-recycling at $Re_a$ = 300, 600 and 900. The focus is put on massive separation and the set-up close to those of Spalart & Coleman (1997) and Na & Moin (1998) at lower Reynolds number. This extends the work of Abe et al. (CTR Annual Brief, 2012) but removes the stagnation point, present over the bubble and due to strong blowing and suction $V_{top}$, at the upper boundary. The new simulations have a reduced $V_{top}$ compensated by a smaller ceiling height. The overall agreement with the earlier DNSs is satisfactory. A small difference appears in the recovery region, in which turbulence is reduced slightly in the present DNSs. At all three $Re_a$, separation and reattachment locations are nearly identical. Also, the mean spanwise vorticity is conserved to a large extent along the bubble. We associate this inviscid transport with the high peak in Reynolds shear stress known to appear after reattachment, and to be a challenging phenomenon for turbulence theories and models. In the latter region, a significant dependence is found for the skin friction due to the weak development of near-wall turbulence.
11:14AM H37.00004 Mitigation of Helical Vortex Separation of a 5:1 Prolate Spheroid at an Angle of Attack . JAMES SCHULWEISTER, MICHAEL TRIANTAFYLLOU, MIT — Ocean and air vehicle maneuvering performance is determined by the balance between a vehicle’s capacity to exert large lateral forces and its fluid dynamic resistance. The flow past a 5:1 prolate spheroid at an angle of attack is representative of many maneuvering hull forms where cross-flow separation creates helical vortices that dramatically increase drag during maneuvers. We investigate a shape modification to a 5:1 prolate spheroid that streamlines the spheroid in the cross-flow direction in order to mitigate vortex separation. We conduct a series of experiments with models towed at angles of attack between 5 and 30 degrees at a Reynolds number by length equal to 100,000. Reconstructions of the helical vortex separation from cross-flow particle image velocimetry measurements show that modest streamlining significantly reduces the size and circulation of separated vortices, indicating drag reduction.

11:27AM H37.00005 Numerical study of turbulent flow separation over a wall mounted circular cylinder1 , TAEJONG YU, DONGHYUN YOU, Department of Mechanical Engineering, Pohang Univ of Sci & Tech — Flow over a wall-mounted circular cylinder with a finite span and a free end is numerically studied at a range of Reynolds numbers. Separated flow behind a wall-mounted cylinder is characterized by dominant vortical structures developed around and behind the cylinder: i.e., Karman vortices and tip-shedding vortices. The formation and interaction among the vortices are found to be distinct depending on the aspect ratio of the span length to the diameter of the cylinder as well as the Reynolds number. It is also found that drag and lift forces on the cylinder show different dominance of Karman vortices and tip vortices for different span-to-diameter ratios. A detailed analysis of the spectral content. We describe the mean and turbulent flow topology and evolution of the structures in the wake flow and discuss the origin of characteristic frequencies observed in the pressure signal at the launcher base. The influence of a propulsive jet on the evolution and topology of the wake flow is discussed in detail.

1The German Research Foundation DFG is gratefully acknowledged for funding this research within the SFB-TR40 “Technological foundations for the design of thermally and mechanically highly loaded components of future space transportation systems.”

11:40AM H37.00006 Flow structure and unsteadiness in the supersonic wake of a generic space launcher1 , ANNE-MARIE SCHREYER, SÖREN STEPHAN, ROLF RADESPIEL, TU Braunschweig — At the junction between the rocket engine and the main body of a classical space launcher, a separation-dominated and highly unstable flow field develops and induces strong wall-pressure oscillations. These can excite structural vibrations detrimental to the launcher. It is desirable to minimize these effects, for which a better understanding of the flow field is required. We study the wake flow of a generic axisymmetric space-launcher model with and without propulsive jet (cold air). Experimental investigations are performed at Mach 2.9 and a Reynolds number \( Re_D = 1.3 \times 10^6 \) based on model diameter D. The jet exits the nozzle at Mach 2.5. Velocity measurements by means of Particle Image Velocimetry and mean and unsteady wall-pressure measurements on the main-body base are performed simultaneously. Additionally, we perform hot-wire measurements at selected points in the wake. We can thus observe the evolution of the wake flow along with its spectral content. We describe the mean and turbulent flow topology and evolution of the structures in the wake flow and discuss the origin of characteristic frequencies observed in the pressure signal at the launcher base. The influence of a propulsive jet on the evolution and topology of the wake flow is discussed in detail.

11:53AM H37.00007 Flow past a finite cylinder of constant curvature . JESSICA SHANG, Stanford University, HOWARD STONE, Princeton University, ALEXANDER SMITS, Princeton University, Monash University — Wake visualization experiments were conducted on a finite cylinder whose plane of curvature is aligned with the freestream, at \( 230 \leq Re_D \leq 916 \). The curvature of the cylinder is constant and ranges from a straight cylinder to a quarter-ring. The wake characteristics depend on the curvature, aspect ratio, and Reynolds number. The wake of a cylinder whose stagnation face is concave to the flow exhibited regions of non-shedding separated flow due the spanwise flow induced by the free end, in addition to regions of oblique and/or parallel shedding. In some cases, vortex shedding was entirely suppressed. In the laminar wake regime, increasing the curvature or decreasing the aspect ratio restricted vortex shedding to smaller regions along the span of the cylinder. The spanwise location where vortex shedding occurred was self-similar across cylinders of the same aspect ratio and varying curvature. After the wake transitioned to turbulence, the vortex shedding extended along most of the cylinder span. In contrast, a cylinder convex to the flow always exhibited vortex shedding in this \( Re_D \) regime. The shed vortex lines were oblique to the flow at lower \( Re_D \) and became more normal to the flow with increasing \( Re_D \). The Strouhal number also increased with \( Re_D \).

12:06PM H37.00008 Optimal control of an asymptotic model of flow separation . UBAID QADRI, PETER SCHMID, Imperial College London, LPC-UK TEAM — In the presence of surface imperfections, the boundary layer developing over an aircraft wing can separate and reattach, leading to a small separation bubble. We are interested in developing a low-order model that can be used to control the onset of separation at high Reynolds numbers typical of aircraft flight. In contrast to previous studies, we use a high Reynolds number asymptotic description of the Navier–Stokes equations to describe the motion of motion of the fluid. We obtain a steady solution to the nonlinear triple-deck equations for the separated flow over a small bump at high Reynolds numbers. We derive for the first time the adjoint of the nonlinear triple-deck equations and use it to study optimal control of the separated flow. We calculate the sensitivity of the properties of the separation bubble to local base flow modifications and steady forcing. We assess the validity of using this simplified asymptotic model by comparing our results with those obtained using the full Navier–Stokes equations.

12:19PM H37.00009 A numerical experiment that provides new results regarding the inception of separation in the flow around a circular cylinder . NIKOLAOS MALAMATARIS, George Mason University, ANASTASIOS LIAKOS, United States Naval Academy — The exact value of the Reynolds number regarding the inception of separation in the flow around a circular cylinder is still a matter of research. This work connects the inception of separation with the calculation of a positive pressure gradient around the circumference of the cylinder. The hypothesis is that inception of separation occurs when the pressure gradient becomes positive around the circumference. From the most cited laboratory experiments that have dealt with that subject of inception of separation only Thom has measured the pressure gradient there at very low Reynolds numbers (up to \( Re=3.5 \)). For this reason, the experimental conditions of his tunnel are simulated in a new numerical experiment. The full Navier Stokes equations in both two and three dimensions are solved with a home made code that utilizes Galerkin finite elements. In the two dimensional numerical experiment, inception of separation is observed at \( Re=4.3 \), which is the lowest Reynolds number where inception has been reported computationally. Currently, the three dimensional experiment is under way, in order to compare if there are effects of three dimensional theory of separation in the conditions of Thom’s experiments.
12:32PM H37.00010 3-D Stall Cell Inducement Using Static Trips on a NACA0015 Airfoil1, HALEY DELL’ORSO, MICHAEL AMITAY, RPI — Stall cells typically occur at high angles of attack and moderate to high Reynolds numbers \(10^5\) to \(10^6\), which are applicable to High Altitude Long Endurance (HALE) vehicles. Under certain conditions stall cells can form abruptly and have a severe and detrimental impact on flight. In order to better understand this phenomenon, stall cell formation is studied using oil flow visualization and SPIV on a NACA0015 airfoil with AR = 2.67. It was shown that there is a critical Reynolds number above which stall cells begin to form, and that \(Re_{crit}\) varies with angle of attack. Zig-zag tape and balsa wood trips were used to induce stall cells at lower Reynolds numbers than they would otherwise be present. This will aid in understanding the formation mechanism of these cells. It was also demonstrated that, in the case of full span trips, stall cells are induced by the 3-D nature of zig-zag trips and did not appear when balsa wood trips were used. This suggests that the formation of the stall cell might be due to 3-D disturbances that are naturally present in a flow field.

1 AFOSR Grant Number FA9550-13-1-0059


10:35AM H38.00001 Acoustic source analysis of supersonic jets from complex nozzles1, JOSEPH W. NICHOLS, JORDAN KREITZMAN, Aerospace Engineering and Mechanics, University of Minnesota — We compute acoustic source terms corresponding to Goldsteins generalized acoustic analogy from an unstructured high-fidelity large eddy simulation of a supersonic jet issuing from a rectangular nozzle with chevrons. The simulation data are validated against experimental measurements of mean and turbulence flow statistics as well as far-field noise. We evaluate fourth-order correlations from the simulation data to assess assumptions of quasi-normality and statistical axisymmetry that underpin reduced-order acoustic source models originally developed for round jets. A spatial analysis of these correlations in relation to the complex geometry of the nozzle reveals locations where the validity of these assumptions begins to break down. Using two point two-time correlations of the simulation data, we also directly evaluate and compare the accuracy of four different acoustic source models, including the Gaussian, moving-frame, fixed-frame, and modified distance models.

1 Computational resources were provided by the Argonne Leadership Computing Facility.

10:48AM H38.00002 Input-output analysis of high-speed turbulent jet noise, JINAH JEUN, JOSEPH W. NICHOLS, University of Minnesota — We apply input-output analysis to predict and understand the aeroacoustics of high-speed isotothermal turbulent jets. We consider axisymmetric linear perturbations about Reynolds-averaged Navier-Stokes solutions of ideally expanded turbulent jets with Mach numbers \(0.6 < M_j < 1.8\). For each base flow, we compute the optimal harmonic forcing function and its linear response by singular value decomposition of the resolvent operator. In addition to the optimal mode, input-output analysis also yields suboptimal modes associated with a spectrum of lesser singular values. For supersonic jets, the optimal response closely resembles a wavepacket in both the nearfield and the farfield such as those obtained by the parabolized stability equations (PSE), and this mode dominates the response. For subsonic jets, however, the singular values indicate that the contributions of suboptimal modes to noise generation are nearly equal to that of the optimal mode, explaining why PSE misses some of the farfield sound in this case. Finally, high-fidelity large eddy simulation (LES) is used to assess the prevalence of suboptimal modes in the unsteady data. By projecting LES data onto the corresponding input modes, the weighted gain of each mode is examined.

11:01AM H38.00003 Global mode decomposition of supersonic impinging jet noise1, NATHANIEL HILDEBRAND, JOSEPH W. NICHOLS, University of Minnesota - Twin Cities — We apply global stability analysis to an ideally expanded, Mach 1.5, turbulent jet that impinges on a flat surface. The analysis extracts axisymmetric and helical instability modes, involving coherent vortices, shocks, and acoustic feedback, which we use to help explain and predict the effectiveness of microjet control. High-fidelity large eddy simulations (LES) were performed at nozzle-to-wall distances of 4 and 4.5 throat diameters with and without sixteen microjets positioned uniformly around the nozzle lip. These flow configurations conform exactly to experiments performed at Florida State University. Stability analysis about LES mean fields predicted the least stable global mode with a frequency that matched the impingement tone observed in experiments at a nozzle-to-wall distance of 4 throat diameters. The Reynolds-averaged Navier-Stokes (RANS) equations were solved at five nozzle-to-wall distances to create base flows that were used to investigate the influence of this parameter. A comparison of the eigenvalue spectra computed from the stability analysis about LES and RANS base flows resulted in good agreement. We also investigate the effect of the boundary layer state as it emerges from the nozzle using a multi-block global mode solver.

1 Computational resources were provided by the Argonne Leadership Computing Facility.

11:14AM H38.00004 Global Mode-Based Control of Supersonic Jet Noise1, MAHESH NATARAJAN2, JONATHAN FREUND3, DANIEL BODONY4, University of Illinois at Urbana-Champaign — The loudest source of high-speed jet noise appears to be describable by unsteady wavepackets that resemble instabilities. We seek to reduce their acoustic impact by developing a novel control strategy that uses global modes to model their dynamics and structural sensitivity of the linearized compressible Navier-Stokes operator to determine effective linear feedback control. Using co-located actuators and sensors we demonstrate the method on an axisymmetric Mach 1.5 fitted with a nozzle. Direct numerical simulations using this control show significant noise reduction, with additional reduction with increase in control gain. Eigenanalysis of the uncontrolled and controlled mean flows reveal fundamental changes in the spectrum at frequencies lower than that used by the control. The non-normality of the global modes is shown to enable this control to affect a wide range of frequencies. The low-frequency wavepacket components are made less acoustically efficient, which is reflected in the far-field noise spectrum. Mean flow alterations are minor near the nozzle and only become apparent further downstream.

1 Office of Naval Research and National Science Foundation
2 PhD student, Department of Aerospace Engineering
3 Professor, Department of Mechanical Science and Engineering and Department of Aerospace Engineering
4 Associate Professor, Department of Aerospace Engineering
The efficiency of different disturbance types and shapes is examined in detail. A one-way coupling procedure is developed for passing fluctuations downstream through the flowpath. This method effectively isolates the core noise from other acoustic sources, enabling a straightforward study of the interaction between core noise and jet exhaust, and allows for a more comprehensive understanding of its impact on the far-field jet acoustics.

Coherent structures and the noise they produce are the focus of this study. The filtered signals including/excluding the events are compared, and the results are further tested using synthetic and randomized signals. The statistics of the event properties, including intermittency, frequency, and magnitude, are consistent with observations from other researchers. We investigate the potential theory applicable to flows with sharp gradients. It degenerates into the LST, global, and PSE analysis under suitable conditions.

This work has been funded by Spectral Energies LLC through an Air Force Research Lab SBIR, an AFOSR Grant and Syracuse University.
10:35AM H39.00001 Anisotropy of small scale turbulence in premixed flames

BOBBITT, BROCK, GUILLAUME BLANQUART, California Institute of Technology — The three Kolmogorov hypotheses are fundamental to the description and modeling of turbulence. It is currently unclear if all three hypotheses remain valid within premixed flames, where dramatic changes in density and viscosity occur. The objective of this study is to assess the validity of Kolmogorov’s hypothesis of local isotropy within turbulent premixed flames. Anisotropy is investigated by considering the vorticity vector, which is characteristic of the smallest turbulent scales. This study is performed on a series of direct numerical simulations of n-heptane/air flames which spans a wide range of Karlovitz numbers and density ratios. It is found that the vorticity becomes isotropic for sufficiently high Karlovitz numbers, supporting the validity of the hypothesis of local isotropy. For smaller Karlovitz numbers, the extent of small scale anisotropy can vary through the flame and depends on the Reynolds number as well as the local Karlovitz number. These results are explained through preferential orientation of the flame surface and its alignment with vorticity. A correlation is proposed for the magnitude of the anisotropy with statistics of the flame surface orientation.

10:48AM H39.00002 One Dimensional Modeling of Vorticity in high Karlovitz Number Turbulent Premixed Flames

CHANDRU DHANDAPANI, Graduate Aerospace Laboratories, California Institute of Technology, BROCK BOBBITT, GUILLAUME BLANQUART, Mechanical Engineering Department, California Institute of Technology — Turbulent combustion involves the interaction of two complex non-linear phenomena: turbulence and chemistry. The current study focuses on modeling the effects of the flame on the turbulence characteristics, more specifically on the vorticity $\omega$, which is characteristic of the smallest turbulent scales. This is performed through an a-priori analysis of high Karlovitz number turbulent premixed flames. The objective is to derive a one-dimensional model equation for the transport of enstrophy, $\omega^2 = \omega \cdot \omega$. The terms in the enstrophy transport equation are modeled and scaled by a combination of flow properties. Results from a series of previously performed direct numerical simulations (DNS), spanning a range of Karlovitz number (Ka), Reynolds number (Re) and flame density ratios, are analysed to obtain the coefficients in the one-dimensional differential equation for enstrophy and demonstrate their dependence, or lack thereof, on Ka and Re. Lastly, the model equation is solved and the results are compared with the DNS results.

11:01AM H39.00003 Dynamic Mode Decomposition (DMD) application to premixed Low Swirl Injector flames

PAUL PALIES, United Technologies Research Center, East Hartford, CT, ROBERT CHENG, Lawrence Berkeley National Laboratory, MS 70-108B, 1 Cyclotron Rd., Berkeley, CA 94720, USA, DUSTIN DAVIS, Pratt and Whitney, 400 Main Street, East Hartford, CT 06108, USA, MILOS ILAK, United Technologies Research Center, East Hartford, CT — DMD is implemented and applied to premixed flame image data from the Low Swirl Injector. The data consists of high speed video flame images at three different equivalence ratios, corresponding to low-amplitude oscillation, transient growth, and high-amplitude oscillation regimes. DMD reveals spectra of growth rates and frequencies with corresponding spatial modes, ranked by mode norm. For the low-amplitude oscillation regime, DMD does not capture any dominant mode shapes or frequencies. For the high-amplitude oscillation case, the frequency of the dominant mode and its harmonics match the frequency recorded by pressure measurement. The spatial mode from DMD is used to extract the propagation velocity of perturbations. In the transient regime, DMD captures the growth rate and frequency of the transient mode. The corresponding DMD spatial mode shows a similar shape to the high oscillation case indicating that the transition to a limit cycle is associated with a convective mode. The underlying mechanism of unsteady heat release is identified as induced by a convected wave along the flame front, whose velocity is confirmed by a separate analysis.

1Supported by Dept.of Energy Contract No. DE-AC02-05CH11231.

11:14AM H39.00004 Direct numerical simulations of flow-chemistry interactions in statistically turbulent premixed flames

PAUL ARIAS, University of Michigan, HARSHAVARDHANA URANAKAR, SWETAPROVO CHAUDHURI, India Institute of Science, Bangalore, HONG IM, King Abdullah University of Science and Technology — The effects of Damköhler number and Karlovitz number on the flame dynamics of three-dimensional statistically planar turbulent premixed flames are investigated by direct numerical simulation incorporating detailed chemistry and transport for a hydrogen-air mixture. The mean inlet velocity was dynamically adjusted to ensure a stable flame within the computational domain, allowing the investigation of time-averaged quantities of interest. A particular interest was on understanding the effects of turbulence on the displacement speed of the flame relative to the local fluid flow. Results show a linear dependence on the displacement speed as a function of total strain, consistent with earlier work on premixed-laminar flames. Additional analysis on the local flame thickness reveals that the effect of turbulence is twofold: (1) the increase in mixing results in flame thinning due to the enhancement of combustion at early onset of the flame, and (2) for large Reynolds number flows, the penetration of the turbulence fur into the preheat zone and into the reaction zone results in localized flame broadening.

11:27AM H39.00005 Quantifying real-gas effects on a laminar n-dodecane – air premixed flame

ABISHEK GOPAL, University of Maryland, College Park, SHASHANK YELLAPANTULA, GE Global Research Center, Niskayuna, NY, JOHAN LARSSON, University of Maryland, College Park — With the increasing demand for higher efficiencies in aircraft gas-turbine engines, there has been a progressive march towards high pressure-ratio cycles. Under these conditions, the aviation fuel, Jet A, is injected into the combustor at supercritical pressures. In this work, we study and quantify the effects of transcriticality on a 1D freely propagating laminar n-dodecane – air premixed flame. The impact of the constitutive state relations arising from the Ideal Gas equation of state (EOS) and Peng-Robinson EOS on flame structure and propagation is presented. The effects of real-gas models of transport properties, such as viscosity on laminar flame speed, are also presented.
11:40AM H39.00006 The flame anchoring mechanism and associated flow structure in bluff-body stabilized lean premixed flames, DAN MICHAELS, SANTOSH SHANBHOGUE, AHMED GHONIEM, Department of Mechanical Engineering, Massachusetts Institute of Technology, Cambridge, MA, USA — We present numerical analysis of a lean premixed flame anchoring on a heat conducting bluff-body. Different mixtures of CH$_4$/H$_2$/air are analyzed in order to systematically vary the burning velocity, adiabatic flame temperature and extinction strain rate. The study was motivated by our experimental measurements in a step combustor which showed that both the recirculation zone length and stability map under acoustically coupled conditions for different fuels and thermodynamic conditions collapse using the extinction strain rate. The model fully resolves unsteady two-dimensional flow with detailed chemistry and species transport, and without artificial flame anchoring boundary conditions. The model includes a low Mach number operator-split projection algorithm, coupled with a block-structured adaptive mesh refinement and an immersed boundary method for the solid body. Calculations reveal that the recirculation zone length correlates with the flame extinction strain rate, consistent with the experimental evidence. It is found that in the vicinity of the bluff body the flame is highly stretched and its leading edge location is controlled by the reagents combustion characteristics under high strain. Moreover, the flame surface location relative to the shear layer influences the vorticity thus impacting the velocity field and the recirculation zone. The study sheds light on the experimentally observed collapse of the combustor dynamics using the reactants extinction strain rate.

11:53AM H39.00007 Study on Turbulent Premixed Flame Regimes with Ignition Using a Reactor Assisted Turbulent Slot Burner, SANG HEE WON, CHRISTOPHER REUTER, Princeton University, BRET WINDOM, University of Colorado Colorado Springs, YIGUANG JU, Princeton University — Turbulent premixed flames of n-heptane/air and toluene/air mixtures affected by ignition have been experimentally investigated by using a reactor-assisted turbulent slot (RATS) burner at two burner temperatures, 450 K and 700 K. Turbulent burning velocities (ST) and flame structures have been measured by the simultaneous OH and CH$_2$O planar laser-induced fluorescence (PLIF) imaging at various equivalence ratios and turbulent Reynolds numbers. Three distinct turbulent premixed flame regimes are identified for n-heptane/air mixture: chemical frozen (CF) regime at low temperature, low temperature ignition (LTI) regime, and high temperature ignition (HTI) regime for respectively lean and rich conditions at 700 K. For CF regime, the measured turbulent burning velocities of n-heptane and toluene at 450 K follow a conventional correlation of turbulent intensity (defined as u’/SL). In LTI regime, substantial changes in chemical composition alter the laminar flame speed and transport property, leading to rapid increase of turbulent burning velocity. In HTI regime, it is found that the turbulent premixed flame structure is significantly modified by the appearance of volumetric ignition kernel structures associated with the transition from LTI to HTI. The turbulent premixed flame regime in HTI is no longer represented by the thin reaction zone regime. The measured turbulent burning velocities in HTI regime increase substantially as increasing ignition Damkohler number over those in LTI regime.

12:06PM H39.00008 Analysis of the electric currents in 1D premixed flames under applied voltages, JIE HAN, MEMDOUH BELHI, FABRIZIO BISETTI, Clean Combustion Research Center, KAUST, TIERNAN CASEY, Combustion Modeling Laboratory, Univ. of California, Berkeley, HONG G. IM, Clean Combustion Research Center, KAUST, JHYUAN CHEN, Combustion Modeling Laboratory, Univ. of California, Berkeley — Studying electric currents in flames has practical aspects such as the determination of the ionization state of a flame, the analysis of the flame behavior under an electric field and the use of flame electric properties for combustion diagnostics. This study proposes a simplified model to compute the electric currents in lean-to-stoichiometric 1D premixed flames under applied voltages. The Navier-Stokes equations coupled with transport equations for neutral and charged species along with a Poisson equation for the electric potential are solved. The model reproduces qualitatively the voltage-current characteristic found experimentally. The sensitivity of the electric currents to the applied voltage, equivalence ratio, and pressure is studied and the key parameters affecting the saturation current are determined. Results show that the saturation current is controlled by the amount of charged species created by the chemi-ionization reaction. We found that the recombination rate of electrons with cations and transport coefficients of charged species are the most important parameters affecting the voltage at which saturation occurs. Analytical formulas for the voltage-current characteristic and the potential of saturation are developed and used to explain the obtained results.

12:19PM H39.00009 Time-resolved stereoscopic PIV study of flashback in swirl flames at elevated pressures, RAKESH RANJAN, DOMINIK EBI, NOEL CLEMENS, University of Texas at Austin — Boundary layer flashback of turbulent premixed swirl flames can pose a major challenge to the operation of stationary gas turbines, especially with hydrogen-rich fuels. To improve our understanding of the physics behind this phenomenon at gas turbine relevant conditions, it is essential to investigate flashback at elevated pressures. With this purpose in mind, flashback experiments with hydrogen/methane-air premixtures are conducted in a swirl combustor installed in an optically accessible high-pressure combustion facility. We have employed stereoscopic PIV in conjunction with high speed chemiluminescence imaging to study the upstream propagation of the flame in the premix tube during flashback. Experiments are run at pressures ranging from 1 atm to 5 atm. These time-resolved measurements provide valuable insight into the flame-flow interaction during flashback at elevated pressures.

12:32PM H39.00010 Vorticity Dynamics in Single and Multiple Swirling Reacting Jets, TRAVIS SMITH, MICHAEL AGUILAR, BENJAMIN EMERSON, DAVID NOBLE, TIM LIEUWEN, None — This presentation describes an analysis of the unsteady flow structures in two multinozzle swirling jet configurations. This work is motivated by the problem of combustion instabilities in premixed flames, a major concern in the development of modern low NOx combustors. The objective is to compare the unsteady flow structures in these two configurations for two separate geometries and determine how certain parameters, primarily distance between jets, influence the flow dynamics. The analysis aims to differentiate between the flow dynamics of single nozzle and triple nozzle configurations. This study looks at how the vorticity in the shear layers of one reacting swirling jet can affect the dynamics of a nearby similar jet. The distance between the swirling jets is found to have an effect on the flow field in determining where swirling jets merge and on the dynamics upstream of the merging location.

1. Graduate Student, School of Aerospace Engineering, Georgia Institute of Technology, Atlanta, GA
2. Propulsion Engineer, Delta Air Lines, Atlanta, GA
3. Research Engineer, School of Aerospace Engineering, Georgia Institute of Technology, Atlanta, GA
4. Research Engineer, School of Aerospace Engineering, Georgia Institute of Technology, Atlanta, GA
5. Professor, School of Aerospace Engineering, Georgia Institute of Technology, Atlanta, GA. AIAA Associate Fellow

Monday, November 23, 2015 10:35AM - 12:45PM – Session H40 Rarefied Flows and DSMC Sheraton Back Bay D - Sergey Averkin, Worcester Polytechnic Institute
10:35AM H40.00001 Janus-Particles in a rarefied gas: thermophoresis and orientation . TOBIAS BAIER, TU Darmstadt, SAMIR SHRESTHA, Kathmandu University, SUDARSHAN TIWARI, TU Kaiserslautern, STEFFEN HARDT, TU Darmstadt, AXEL KLAR, TU Kaiserslautern — Thermophoresis, the motion of a particle along a thermal gradient, has been used both for preventing and inducing the deposition of aerosols on heated or cooled surfaces. In the latter case it may be advantageous to induce the deposition with a preferred orientation of the particle, for example by utilizing non-uniform reflective properties on the particle surface. As a model system we investigate a spherical Janus particle on which gas molecules are reflected diffusively from one hemisphere and specularly from the other. In the limit of large Knudsen number this is studied analytically, focusing on the interplay between thermophoretic motion and alignment of the particle. Without motion, a torque orients the particle with its diffuse side towards the colder gas. However, any motion of the particle relative to the gas results in a preferred alignment with the specular side in direction of the particle velocity. Thus the thermophoretic motion, which is towards the colder side, results in a weakening of the particle alignment. The results are supported by Monte-Carlo simulations to extend the range of validity towards finite Knudsen numbers. These findings shed light on the efficiency of aligned deposition of nanoparticles from a gas stream on a cooled surface.

10:48AM H40.00002 ABSTRACT WITHDRAWN —

11:01AM H40.00003 Slow flow of a rarefied gas past a sphere: Numerical analysis of fundamental problem 1. SATOSHI TAGUCHI, TOSHIHIRO SUZUKI, University of Electro-Communications — A slow flow of a rarefied gas past a sphere with a uniform temperature is considered with a special interest in the drag exerted on the sphere. It was shown previously [S. Taguchi, J. Fluid Mech. 774, 363–394 (2015)] that the drag up to the second order of the Mach number is expressed in terms of two fundamental functions depending on the Knudsen number, which are obtained by solving the corresponding linearized problem. The present study aims to obtain these functions on the basis of the ellipsoidal-statistical (ES) model of the Boltzmann equation under the diffuse reflection boundary condition.

11:14AM H40.00004 Aerothermodynamics of compressible flow past a flat plate in the slip-flow regime . CHI-YANG CHENG, YI DAI, GENONG LI, YITAO HU, ANSYS, Inc, MING-CHIA LAI, Wayne State University — Compressible flow past a flat plate in the slip-flow regime features a very simple geometry and flow field, but it retains the most relevant and interesting physics in high-speed rarefied gas dynamics. In the slip-flow regime, the aerothermodynamic issues, especially the recovery factors and the convection heat transfer correlation, are the focus of this presentation. We present the detailed similarity equations, especially the transformed Maxwell’s slip and jump boundary conditions, and the equations for the Chapman-Rubesin parameter as well as how we incorporate the variable gas properties and the constitutive scaling model for the Knudsen layer in the similarity equations. The similarity solutions are compared with results published by E. R. van Driest [NACA Technical Note 2597, 1952]. We point out that van Driest’s solutions were computed by using no-slip and no-jump boundary conditions. The recovery factor and Nusselt number of the plate are shown as functions of the Reynolds number and the Mach number. Finally, the similarity solutions are also compared with the numerical solutions of a two-dimensional computational fluid dynamics model solving the full Navier-Stokes-Fourier equations with slip and jump boundary conditions.

11:27AM H40.00005 Numerical solution of Boltzmann equation using discrete velocity grids . PRAKASH VEDULA, University of Oklahoma, Norman — An importance sampling based approach for numerical solution of the (single species) Boltzmann equation using discrete velocity grids is proposed. This approach involves a stochastic method for evaluation of the collision integral based on sampling of deleeting/replenishing collisions and is designed to preserve important symmetries of the collision operator, including collision invariants. The underlying particle distribution function is represented as a collection of delta functions with associated weights that are non-negative. A key feature in the construction of the proposed method is that it ensures that the weights associated with the distribution function remain non-negative during collisional relaxation, thereby satisfying an important realizability condition. Performance of the proposed approach will be studied in two test problems involving spatially homogeneous collisional relaxation flow and microchannel flows. Results obtained from the proposed method will be compared with those obtained from the (deterministic) collisional Lattice Boltzmann Method (cLBM) and the traditional direct simulation Monte Carlo (DSMC) method for solution of Boltzmann equation. Extension of the proposed method using discrete velocity grids for multicomponent mixtures will also be discussed.

11:40AM H40.00006 A Kinetic 13-Moment Boundary Conditions Method for Particle Simulations of Viscous Rarefied Flows 1. SERGEY AVERKIN, NIKOLAOS GATSONIS, Worcester Polytechnic Institute — The kinetic 13-moment (Navier-Stokes-Fourier) boundary condition method is developed for direct simulation Monte Carlo (DSMC) simulations of rarefied gas flows. The particles are injected into the computational domain from the inlet and outlet following the first-order Chapman-Enskog distribution function. The unknown parameters of the Chapman-Enskog distribution function are reconstructed from the full 13-moment (Navier-Stokes-Fourier) equations discretized on the boundaries with the wave amplitudes calculated by the local one dimensional inviscid (LDOI) formula used in compressible (continuous) flow computations. The kinetic-moment boundary conditions are implemented in an unstructured 3D DSMC (U3DSMC) code and are supplemented with a neighboring-cell sampling approach and a time-average smoothing techniques to speed up convergence and reduce fluctuations. Simulations of a pressure-driven viscous subsonic flow in a circular tube are used for verification and validation of the boundary conditions. In addition, the present method is compared to the previously developed kinetic-moment boundary conditions derived from the five-moment (Euler) equations.

11:53AM H40.00007 DSMC-LBM hybrid scheme for flows with variable rarefaction conditions . GIANLUCA DI STASO, Technische Universität Eindhoven, SAURO SUCCI, Istituto per le Applicazioni del Calcolo - Consiglio Nazionale delle Ricerche - Roma, FEHRITTO SCHÖSSLER, Technische Universität Eindhoven — The kinetic description of gases, based on the Boltzmann equation, allows to cover flow regimes ranging from the rarefied to the continuum limit. The two limits are traditionally studied by numerically approximating the Boltzmann equation via Direct Simulation Monte Carlo (DSMC) method or the Lattice Boltzmann Equation method (LBM). While DSMC is suitable for rarefied flows, its computational cost makes it impractical to study hydrodynamic flows. The LBM has instead proved itself to be an efficient and accurate method in the hydrodynamic limit even though simulation of rarefied flows requires additional modeling. Here, results on the development of a hybrid scheme capable of coupling the LBM and the DSMC methods and able to efficiently simulate flows with variable rarefaction conditions are presented. The coupling scheme is based on Grad’s moment method approach and the local single particle distribution function at a given order of truncation is built by using the Hermite polynomials expansion approach and Gauss-Hermite quadratures. The capabilities of the hybrid approach for simulating flows in the transition regime are illustrated in the case of planar Couette and Poiseuille flows.
12:06PM H40.00008 Relaxation rates for inverse power law particle interactions and their variable hard sphere surrogates, ROBERT RUBINSTEIN, None — It is well known that collision models based on an assumed intermolecular potential (IPL, LJ, ...) can be successfully replaced by simplified surrogates (VHS, VSS, VS, ...) in DSMC calculations. But these surrogates only reproduce certain gross properties of the molecular model, for example, the temperature dependence of the viscosity; they do not approximate, and even mis-state, the details of the particle interactions. The success of the simplified models in problems at finite Knudsen number, where the Navier-Stokes approximation is not valid, may therefore seem surprising. To understand this success in a very special case, we showed that the first seven relaxation rates of the linearized Boltzmann equation for Maxwellian molecules are well approximated by the corresponding relaxation rates of its VHS surrogate. We will show that this analysis can be extended in somewhat less generality to IPL interactions, and to some extent to more realistic models including LJ. We believe that this analysis can help address the more general problem of identifying the properties of the collision model that dominate the predictions of the Boltzmann equation.

12:19PM H40.00009 DSMC Simulation of Microstructure Actuation by Knudsen Thermal Force, AARON PIKUS, ISRAEL SEBASTIAO, ANDREW STRONGRICH, ALINA ALEXEENKO, Purdue University — Compact, low-power and highly accurate vacuum sensors are needed for emerging applications such as high-altitude communication platforms, small satellites and i-vacuum manufacturing processes. A novel MEMS-based pressure and gas sensor — Microelectromechanical In-plane Knudsen Radiometric Actuator (MIKRA) — has been developed at Purdue. MIKRA is based on Knudsen thermal force generated by rarefied flow driven by thermal gradients within the microstructure. The goal of this work is to model the rarefied gas flow in the MIKRA sensor to validate the numerical modeling of rarefied thermally-driven flows and gain insights for sensor design. The Direct Simulation Monte Carlo (DSMC) solver SPARTA is employed to numerically calculate the distribution of the flowfield and surface properties. The resulting forces on the colder shuttle beam are calculated and compared to the available experimental data as well as other numerical solvers. Both DSMC and experimental results suggest that the maximum forces occur at a Knudsen number of approximately 1. The streamlines indicate the presence of two small vortexes between the heated beam and the colder shuttle beam, and a larger one above these two beams. The DSMC simulations, validated by experimental measurements, help understand the unique flow behaviors encountered in rarefied thermally-driven flows.

12:32PM H40.00010 Analytical and Numerical Modeling of Strongly Rotating Rarefied Gas Flows, SAHADEV PRADHAN, VISWANATHAN KUMARAN, Department of Chemical Engineering, Indian Institute of Science, Bangalore-560012, India — Centrifugal gas separation processes effect separation by utilizing the difference in the mole fraction in a high speed rotating cylinder caused by the difference in molecular mass, and consequently the centrifugal force density. These have been widely used in isotope separation because chemical separation methods cannot be used to separate isotopes of the same chemical species. More recently, centrifugal separation has also been explored for the separation of gases such as carbon dioxide and methane. The efficiency of separation is critically dependent on the secondary flow generated due to temperature gradients at the cylinder wall or due to inserts, and it is important to formulate accurate models for this secondary flow. The widely used Onsager model for secondary flow is restricted to very long cylinders where the length is large compared to the diameter, the limit of high stratification parameter, where the gas is restricted to a thin layer near the wall of the cylinder, and it assumes that there is no mass difference in the two species while calculating the secondary flow. There are two objectives of the present analysis of the rarefied gas flow in a rotating cylinder. The first is to remove the restriction of high stratification parameter, and to generalize the solutions to low rotation speeds where the stratification parameter may be O(1).

Monday, November 23, 2015 1:55PM - 2:30PM — Session J1 Invited Session: Microhydrodynamics of Deformable Particles: Surprising Responses of Drops and Vesicles to Uniform Electric Field or Shear Flow, PETIA VLAHOVSKA, Brown University — Particle motion in a viscous fluid is a classical problem that continues to surprise researchers. In this talk, I will discuss some intriguing, experimentally-observed behaviors of droplets and giant vesicles (cell-size lipid membrane sacs) in electric or flow fields. In a uniform electric field, a droplet deforms into an ellipsoid that can either be steadily tilted relative to the applied field direction or undergo unsteady motions (periodic shape oscillations or irregular flipping); a spherical vesicle can adopt a transient square shape or reversibly porate. In a steady shear flow, a vesicle can tank-tread, tumble or swing. Theoretical models show that the nonlinear drop dynamics originates from the interplay of Quincke rotation and interface deformation, while the vesicle dynamics stems from the membrane inextensibility. The practical motivation for this research lies in an improved understanding of technologies that rely on the manipulation of drops and cells by flow or electric fields.

Monday, November 23, 2015 1:55PM - 2:30PM — Session J34 Invited Session: Using Optimisation to Identify the ”Best” Way to Trigger Flow Transition, RICHARD KERSWELL, Bristol University — Understanding how fluids break down to turbulence has proved a fascinating and enduringly-difficult problem in fluid mechanics. Wall-bounded shear flows such as pipe flow, plane Couette flow and channel flow are particularly interesting as the observed transition depends on the environment and typically occurs abruptly leading immediately to complicated spatiotemporal flows. Theoretical efforts to gain understanding since the 1990s have either mostly focussed on a small-amplitude perspective of analysing the appropriate linear operator (modal and nonlinear analyses) or taken a fully nonlinear perspective of calculating exact solutions of the equations and studying how their stable and unstable manifolds structure phase space (a dynamical systems approach). I will discuss how this conceptual gap (in amplitude) can be bridged using an optimisation approach which can be used to probe the nonlinear stability of a flow state. If the alternative state is turbulence, this approach identifies the optimal disturbance the minimal seed - to trigger transition. The essential ideas will be illustrated in a simple 2-degree-of-freedom model system before results for the Navier-Stokes equations and pipe flow will be presented together with some future perspectives. (Joint work with C. Caulfield, C. Pringle, S. Rabin & A. Willis)
**Monday, November 23, 2015 2:45PM - 3:20PM –**

**Session K1 Invited Session: Flow Near Singular Elastic Interfaces: Lubrication, Wetting and Cusps**

Auditorium - Martin Maxey, Brown University

**2:45PM K1.00001 Flow near singular elastic interfaces: Lubrication, wetting & cusps**

JACCO SNOEJIER, University of Twente and Eindhoven University of Technology — Soft elastic surfaces exhibit intriguing similarities to liquid interfaces: they possess a surface tension and are susceptible to interfacial instabilities. The first part of this talk addresses why the wetting dynamics on such a very soft substrate is fundamentally different from the usual wetting hydrodynamics. The initially flat solid is deformed into a sharp ridge and the resulting contact line motion is governed by the rheology of the solid. We experimentally observe an irregular stick-slip motion of the drop, and explain this phenomenon by a viscoelastic theory for the dynamic contact angle. In the second part we address the morphology of sharp cusps that arise at elastic interfaces. Inspired by classical work on viscous cusps at liquid interfaces, we reveal the emergence of self-similar cusps in the regime of extremely nonlinear elasticity.

**Monday, November 23, 2015 2:45PM - 3:20PM –**

**Session K34 Invited Session: Instability and Turbulence of Propagating Particulate Flows**

Ballroom BC - Martin Maxey, Brown University

**2:45PM K34.00001 Instability and Turbulence of Propagating Particulate Flows**

BALACHANDAR, University of Florida — Propagation of particle-laden fluid into an ambient is a common fluid mechanical process that can be observed in many industrial and environmental applications. Sedimentation fronts, volcanic plumes, dust storms, powder snow avalanches, submarine turbidity currents, explosive powder dispersal, and supernovae offer fascinating examples of advancing particulate fronts. The propagating interface can undergo Rayleigh-Taylor, Kelvin-Helmholtz and double-diffusive instabilities and result in the formation of lobes and clefts, spikes and bubbles, and particulate fingers. The interplay between suspended particles and turbulence is often complex due to interaction of competing mechanisms. In problems such as turbidity currents, turbulence controls sediment concentration through resuspension and settling of particles at the bed. Also, turbulent entrainment at the propagating front is observed to be influenced by the sediments. Stable stratification due to suspended sediment concentration can damp and even kill turbulence. This complex turbulence-sediment interaction offers possible explanation for massive sediment deposits observed in nature. The talk will also address challenges and recent advancements in the modeling and simulation of such particle-laden turbulent flows.

**3:20PM - 3:20PM –**

**Session KP1 Poster Session**

Exhibit Hall D (Technical Poster Display Area) -

**KP1.00001 ACOUSTICS –**

KP1.00002 Exploration of the Use of Triple Correlation in the Analysis of Acoustic Data

GENEVIEVE STARKE, JACQUES LEVALLE, Syracuse University — Jet noise is a long-standing environmental problem. To better understand the underlying mechanisms, acoustic data from several microphones in the far field of a Ma = 0.6 round jet are being analyzed. Currently, cross-correlation is being used to compare two signals and find the largest peak value of the correlation coefficient. This indicates the best match for the same information reaching two microphones at different times; it also gives the lag of the sound events which is helpful in the location of the sources of the events. This is traditionally done with pairs of signals, either with entire signals or with event-level short excerpts. Then, multiple signals require the matching of the results of various pair-wise correlations. Here, we explore triple correlation as a way to make this process more efficient. Triple correlation uses a sliding method, adjusting the signals by time steps and graphing the mean of the product of the result. One of the signals is selected as a base, and the two other signals are shifted relative to the base signal. The result gives a graph giving the lags of both signals in relation to the base signal. The triple correlation algorithm is being tested on simple signals and will be applied to experimental data. We will report on the applicatio

KP1.00003 Addressing the likelihood of cumulative nonlinear distortion in supersonic jet noise using the effective Goldber number

WOUTJIN J. BAARS, University of Melbourne, CHARLES E. TINNEY, MARK F. HAMILTON, University of Texas at Austin — When replicating full-scale jets by way of sub-scale experiments, it is routine to aim for aerodynamic similarity; this is achieved by matching the jet’s geometry, Mach number, temperature ratio and Reynolds number. We here compute the effective Goldber number (Λ) to assess acoustic similarity for supersonic jets—whether the wave propagation obeys by linear or nonlinear theory. Cumulative nonlinear wave distortion may only appear when the jet flow and ambient surround are sufficient incubators for distortion. Noticeably, the imperative conditions for sustaining this distortion do not scale following aerodynamic similarity laws. A method for computing Λ encompasses a ray tube situated along the Mach angle where the sound is not only most intense, but advances from undergoing cylindrical- to spherical-decay in its pressure amplitude. Hence, values of Λ are computed separately for the cylindrically and spherically spreading regions, for a plethora of experimental databases. The findings demonstrate how for sub-scale jets, cumulative nonlinear distortion may be present in the region of cylindrical spreading alone. It is revealed that nonlinear distortion is likely to sustain in the region of spherical pressure decay when full-scale jets are concerned.

**KP1.00004 AERODYNAMICS –**

**KP1.00005 ABSTRACT WITHDRAWN –**
Large eddy simulation of a high speed train geometry under cross-wind with an adaptive lattice Boltzmann method. RALF DEITERDING, University of Southampton — Aerodynamics and Flight Mechanics Group, MORITZ M. FRANGER, German Aerospace Center (DLR) - Institute of Aerodynamics and Flow Technology. — Numerical investigations in order to determine the forces induced by side wind onto a train geometry are generally not sufficiently accurate to be used as a predictive tool for regulatory safety assessment. Especially for larger yaw angles, the turbulent cross-wind flow is characterized by highly instationary behavior, driven primarily by vortex shedding on the roof and underside geometric details, i.e., the bogie and wheel systems. While industry-typical Reynolds-averaged turbulence models are not well suited for this scenario, better results are obtained when large eddy simulation (LES) techniques are applied. Here, we employ recently self-developed weakly compressible lattice Boltzmann method (LBm) with Smagorinsky LES model on hierarchically adaptive block-structured Cartesian meshes. Using a train front-car of 1:25 scale at yaw angle 30° and Re = 250,000 as main test case, we compare the LBm results with incompressible large eddy and detached eddy simulations on unstructured boundary-layer type meshes using the OpenFOAM package. It is found that time averaged force and moment predictions from our LBm code compare better to available wind tunnel data, while mesh adaptation and explicit nature of the LBm approach reduce the computational costs considerably.

The Ultimate Flow Controlled Wind Turbine Blade Airfoil. AVRAHAM SEIFERT, DANNY DOLGOPYAT, ORI FRIENDLAD, LIOR SHIG, Tel Aviv Univ. — Active flow control is being studied as an enabling technology to enhance and maintain high efficiency of wind turbine blades also with contaminated surface and unsteady winds as well as at off-design operating conditions. The study is focused on a 25% thick airfoil (DU91-W2-250) suitable for the mid blade radius location. Initially a clean airfoil was fabricated and tested, as well as compared to Xfoil predictions. From these experiments, the evolution of the separation location was identified. Five locations for installing active flow control actuators are available on this airfoil. It uses both Piezo fluidic (“Synthetic jets”) and the Suspension and Oscillatory Blowing (SaOB) actuators. Then we evaluate both actuation concepts overall energy efficiency and efficacy in controlling boundary layer separation. Since efficient actuation is to be found at low amplitudes when placed close to separation location, distributed actuation is used. Following the completion of the baseline studies the study has focused on the airfoil instrumentation and extensive wind tunnel testing over a Reynolds number range of 0.2 to 1.5 Million. Sample results will be presented and outline for continued study will be discussed.

Design of Shrouded Airborne Wind Turbine & CFD Analysis. FAIQA ANBREEN, REZA TOOSSI, Cal State Univ- Long Beach, FAIQA ANBREEN COLLABORATION. — The focus is to design a shrouded airborne wind turbine, capable to generate 70 kW to propel a leisure boat. The idea of designing an airborne turbine is to take the advantage of different velocity layers in the atmosphere. The blades have been designed using NREL S826 airfoil, which has coefficient of lift C/L = 0.8. The rotor diameter is 7.4 m. The balloon (shroud) has converging-diverging nozzle design, to increase the mass flow rate through the rotor. The ratio of inlet area to throat area, A1/A2 is 1.31 and exit area to throat area, Ae/A2 is1.15. The Solidworks model has been analyzed numerically using CFD. The software used is StarCCM+. The Unsteady Reynolds Averaged Navier Stokes Simulation (URANS) K-ε model has been selected, to study the physical properties of the flow, with emphasis on the performance of the turbine. Stress analysis has been done using Nastran. From the simulations, the torque generated by the turbine is approximately 800N-m and angular velocity is 21 rad/s.

Toward Affordable, Theory-and-Simulation-Inspired, Models for Realistic Wind Turbine Aerodynamics and Noise. FULUSO LADEINDE, Stony Brook University, Stony Brook, NY 11794-2300 USA, KEN ALABI, WENHAI LI, TTC Technologies, Inc., Centereach, NY 11720 USA. — The problem of generating design data for the operation of a farm of wind turbines for clean energy production is quite complicated, if properly done. Potential flow theories provide some models, but they are not suitable for the massive aerodynamic separation and turbulence that characterize many realistic wind turbine applications. Procedures, such as computational fluid dynamics (CFD), which can potentially resolve some of the accuracy problems with the purely theoretical approach, are quite expensive to use, and often prohibit real-time design and control. In our work, we seek affordable and acceptably-accurate models derived from the foregoing approaches. The simulation used in our study is based on high-fidelity CFD, meaning that we use high-order (compact-scheme based), mostly large-eddy simulation methods, with due regards for the proper treatment of the stochastic inflow turbulence data. Progress on the project described herein will be presented.

ASTROPHYSICAL FLUID DYNAMICS —.

Implications of new planet discoveries to our knowledge of solar system. VICTOR CHRISTIANTO, Scireprint.org, FLORENTIN SMARANDACHE, University of New Mexico. — In recent years a number of new planetoids have been reported, in particular by M. Brown and his team. While new planet discoveries have been reported from time to time, known as

Design of Shrouded Airborne Wind Turbine & CFD Analysis. FAIQA ANBREEN, REZA TOOSSI, Cal State Univ- Long Beach, FAIQA ANBREEN COLLABORATION. — The focus is to design a shrouded airborne wind turbine, capable to generate 70 kW to propel a leisure boat. The idea of designing an airborne turbine is to take the advantage of different velocity layers in the atmosphere. The blades have been designed using NREL S826 airfoil, which has coefficient of lift C/L = 0.8. The rotor diameter is 7.4 m. The balloon (shroud) has converging-diverging nozzle design, to increase the mass flow rate through the rotor. The ratio of inlet area to throat area, A1/A2 is 1.31 and exit area to throat area, Ae/A2 is1.15. The Solidworks model has been analyzed numerically using CFD. The software used is StarCCM+. The Unsteady Reynolds Averaged Navier Stokes Simulation (URANS) K-ε model has been selected, to study the physical properties of the flow, with emphasis on the performance of the turbine. Stress analysis has been done using Nastran. From the simulations, the torque generated by the turbine is approximately 800N-m and angular velocity is 21 rad/s.

Toward Affordable, Theory-and-Simulation-Inspired, Models for Realistic Wind Turbine Aerodynamics and Noise. FULUSO LADEINDE, Stony Brook University, Stony Brook, NY 11794-2300 USA, KEN ALABI, WENHAI LI, TTC Technologies, Inc., Centereach, NY 11720 USA. — The problem of generating design data for the operation of a farm of wind turbines for clean energy production is quite complicated, if properly done. Potential flow theories provide some models, but they are not suitable for the massive aerodynamic separation and turbulence that characterize many realistic wind turbine applications. Procedures, such as computational fluid dynamics (CFD), which can potentially resolve some of the accuracy problems with the purely theoretical approach, are quite expensive to use, and often prohibit real-time design and control. In our work, we seek affordable and acceptably-accurate models derived from the foregoing approaches. The simulation used in our study is based on high-fidelity CFD, meaning that we use high-order (compact-scheme based), mostly large-eddy simulation methods, with due regards for the proper treatment of the stochastic inflow turbulence data. Progress on the project described herein will be presented.

ASTROPHYSICAL FLUID DYNAMICS —.

Implications of new planet discoveries to our knowledge of solar system. VICTOR CHRISTIANTO, Scireprint.org, FLORENTIN SMARANDACHE, University of New Mexico. — In recent years a number of new planetoids have been reported, in particular by M. Brown and his team. While new planet discoveries have been reported from time to time, known as

Simultaneous PLIF/PIV measurements for a single-mode inclined interface. MOHAMMAD MOHAGHR, DAVID REILLY, JOHN CARTER, Georgia Inst of Tech, JACOB MCFARLAND, University of Missouri, DEVESH RANJAN, Georgia Inst of Tech — The Shock Tube and Advanced Mixing Laboratory (STAML) at Georgia Institute of Technology is using a newly established inclined shock tube facility to study an inclined interface perturbation. This facility allows for simultaneous characterization of density and velocity fields by employing high-resolution, full-field Planar Laser-Induced Fluorescence (PLIF) and Particle Image Velocimetry (PIV), respectively. The incident shock strength of Mach 1.55 was used to impulsively accelerate a N2-Acetone mixture over CO2 inclined interface with an Atwood number of 0.23 and an 80° angle of inclination. This angle of inclination results in a linear perturbation as defined by the amplitude-to-wavelength ratio (η/λ = 0.097). The development of the turbulent mixing layer for both pre- and post-shock flow is determined by measuring several quantities, including two BHR model parameters: density self-correlation and turbulent mass flux.

BIOLOGICAL FLUID DYNAMICS —
KP1.00014 RANS and LES simulations of the airflow through nasal cavities, GIACOMO LAMBERTI, Columbia Univ — The prediction of detailed flow patterns in nasal cavities using computational fluid dynamics (CFD) can provide essential information on the potential relationship between patient-specific geometrical characteristics and health problems. The long-term goal of the OpenNOSE project is to develop a reliable open-source computational tool based on the OpenFOAM CFD toolbox that can assist surgeons in their daily practice. The objective of this study was to investigate the effect of the turbulence model and boundary conditions on simulations of the airflow in nasal cavities. The geometry, including paranasal sinuses, was reconstructed from a carefully selected CT scan, and RANS and LES simulations were carried out for steady inspiration and expiration. At a flow rate near 20 l/min, the flow is laminar in most of the domain. During the inspiration phase, turbulence develops in nasopharynx and oropharynx regions; during the expiration phase, another vortical region is observed down the nostrils. A comparison between different boundary conditions suggests the use of a total pressure condition, or alternatively a uniform velocity, at the inlet and outlet. In future work the same geometry will be used for setting up a laboratory experiment, intended to cross-validate the numerical results.

KP1.00015 Turbulence effects on hemolysis by revisiting experiments with LES computations, MESUDE OZTURK, EDGAR O’REAR, DIMITRIOS PAPAVASSILIOU, The University of Oklahoma — Determining mechanically stimulated red blood cell trauma as a function of turbulence properties is required to design prosthetic heart devices [1]. Because blood is typically exposed to turbulence in such devices, the design of prosthetic heart devices depends on determining the effect of turbulent stresses on hemolysis [2,3]. While turbulent stresses increase hemolysis when cells are exposed to them [3], turbulent flow characteristics in the vicinity of lysed blood cells, and the mechanism of cell damage remains uncertain [2,3]. In this work, LES computations are used to investigate the effect of turbulent eddy structure on cell damage. The flow was simulated for classic Couette and capillary tube experiments [3,4], in order to examine the relation between hemolysis turbulence properties related to the dissipation of turbulent kinetic energy. The hypothesis tested is that eddies that are close in size with the erythrocytes are the ones that are responsible for hemolysis, rather than Reynolds stresses or viscous stresses. We define extensive measures, like the eddy areas for small eddies comparable to the size of the red blood cells, to provide a more general understanding of the mechanical cause of blood trauma. References 1. Quinlan NJ, Dooley PN. Ann Biomed Eng. 2007;35:1347-56. 2. Aız A, et al. Ann Biomed Eng. 2007;35:2108-20. 3. Kameeva MV, et al., ASAIO, 2004;50:418-23. 4. Sutera SP, Mehrjardi MH., Biophysical J., 1975;15:1-10.

KP1.00016 Influence of substrate micropatterning on biofilm growth, STEPHAN KOEHLER, SEAS, Harvard, MA, YIWEI LI, SEAS, Harvard, MA & Huazhong University of Science and Technology, China, BI-FENG LIU LIU, Huazhong University of Science and Technology, China, DAVID WEITZ, SEAS, Harvard, MA — We culture triple reporter Bacillus Subtilis biofilm on micropatterned agar substrates. We track the biofilm development in terms of size, thickness, and phenotype expression. For a tiling composed of elevated rectangles, we observe the biofilm develops an oval shape or triangular shape depending on the rectangle’s aspect ratio and orientation. The motile cells are primarily located in the valleys between the rectangles and the matrix producing cells are mostly located on the rectangles. Wrinkles form at the edges of the elevated surfaces, and upon merging form channels centered on the elevated surface. After a few days, the spore-forming cells appear at the periphery. Since biofilms in nature grow on irregular surfaces, our work may provide insight into the complex patterns observed.

KP1.00017 The effect of low accurate vesicle suspensions on observables, BRYAN QUAIFE, Florida State University, GEORGE BIROS, University of Texas — Vesicle suspensions, which are experimental and numerical proxies for red blood cells and other biomembranes, are ubiquitous in various applications such as biological flows. The governing equations for vesicle suspensions include several challenging aspects such as non-local interactions, an inextensibility and incompressibility constraint, and a fourth-order arc-length derivative. The simulation of vesicle suspensions can be accelerated by introducing several approximations. For instance, vesicles can be discretized at a coarse resolution, the accuracy of the non-local interactions can be reduced, and the area and length of the vesicle can be locally corrected. First I will describe several algorithms that are required to maintain stability at low accuracies. Then, I will discuss the effect these approximations have on several observables, such as the statistics of the velocity field.

KP1.00018 MOVED TO R35.010 —

KP1.00019 Water Transport through Cohesion-Tension in Porous Structures, SRINIVAS KOSARAJU, Northern Arizona University — The predominant theory to explain water transport through plant xylem is the cohesion-tension theory. According to the theory, negative pressure is created due to water evaporation through millions of microscopic capillary pores from tree leaves. The negative pressures are large enough to lift water hundreds of feet against gravity. In an attempt to replicate the process, multiple structures with varying porosity are tested to create negative pressures through water evaporation. The negative pressure created is used to support a water column. The current research is aimed to create artificial leaves using porous structures and be able to transport water in high rise buildings using renewable energy sources such as solar power.

KP1.00020 Flow distributions and spatial correlations in human brain capillary networks, SYLVIE LORTHOIS, MYRIAM PEYROUNETTE, ANNE LARUE, Institut de Mécanique des Fluides de Toulouse, UMR CNRS INPT UPS 5502, TANGUY LE BORGNE, Université de Rennes 1 — The vascular system of the human brain cortex is composed of a space filling mesh-like capillary network connected upstream and downstream to branched quasi-fractal arterioles and venules. The distribution of blood flow rates in these networks may affect the efficiency of oxygen transfer processes. Here, we investigate the distribution and correlation properties of blood flow velocities from numerical simulations in large 3D human intra-cortical vascular network (10 000 segments) obtained from an anatomical database. In each segment, flow is solved from a 1D non-linear model taking account of the complex rheological properties of blood flow in microcirculation to deduce blood pressure, blood flow and red blood cell volume fraction distributions throughout the network. The network structural complexity is found to impart broad and spatially correlated Lagrangian velocity distributions, leading to power law transit time distributions. The origins of this behavior (existence of velocity correlations in capillary networks, influence of the coupling with the feeding arterioles and draining veins, topological disorder, complex blood rheology) are studied by comparison with results obtained in various model capillary networks of controlled disorder.

1 ERC BrainMicroFlow GA615102, ERC ReactiveFronts GA648377
The optimal method and timing of intervention remain uncertain especially for mild COA. A peak-to-peak trans-coarctation pressure gradient (P2-P1) of greater than 20 mmHg warns severe COA and the need for interventional/surgical repair. The optimal method and timing of intervention remain uncertain especially for mild COA (P2-P1 < 20 mmHg); even it is unclear if mild COA should be treated at all. Although it was recently suggested that treatment strategies for mild COA may need to be redefined as transcatheter interventions emerge, benefits of such interventions are unclear. We investigated the effects of transcatheter interventions on the aorta and left ventricle (LV) hemodynamics in 11 patients with mild COA using a developed computational fluid dynamics and lumped parameter modeling framework along with particle image velocimetry and clinical measurements. Such interventions can improve aortic hemodynamics to some extent (e.g., time-averaged wall shear stress and kinetic energy were reduced by about 20%). However, there is no concomitant effect on the LV hemodynamics (e.g., stroke work and LV pressure were reduced by only less than 4%). Our computational approach can effectively predict clinical conditions. Herein one must question intervention for mild COA, as it has limited utility in reducing myocardial strain.

A projection scheme for velocity field reconstruction in the left ventricle of Advanced Machines and Design, Seoul National University, — In nature, the size of the flow channels systematically decreases with multiple generations of branching, and a mother branch is ultimately divided into numerous terminal daughters. The authors acknowledge financial support from Turkish National Scientific and Technical Research Council (TUBITAK) through project numbers 111M332 and 214M293.
KP1.00028 Aeromechanics of the Spider Cricket Jump: How to Jump 60+ Times Your Body Length and Still Land on Your Feet, EMILY PALMER, Johns Hopkins University, NICOLAS DESHLER, Washington International School, DAVID GORMAN, CATARINA NEVES, RAJAT MITTAL, Johns Hopkins University — Flapping, gliding, running, crawling and swimming have all been studied extensively in the past and have served as a source of inspiration for engineering designs. In the current project, we explore a mode of locomotion that straddles ground and air: jumping. The subject of our study is among the most proficient of long-jumpers in Nature: the spider cricket of the family Rhaphidophoridae, which can jump more than 60 times its body length. Despite jumping this immense distance, these crickets usually land on their feet, indicating an ability to control their posture during “flight.” We employ high-speed videogrammetry, to examine the jumps and to track the crickets posture and appendage orientation throughout their jumps. Simple aerodynamic models are developed to predict the aerodynamic forces and moment on the crickets during flight. The analysis shows that these wingless insects employ carefully controlled and coordinated positioning of the limbs during flight so as to increase jump distance and to stabilize body posture during flight. The principles distilled from this study could serve as an inspiration for small jumping robots that can traverse complex terrains.

KP1.00029 Development of a New Method for Platelet Function Test and Its Shearing Condition in Microfluidic System, HOYOUN LEE, GYEHYU KIM, SEAWHAN CHOI, SEHYUN SHIN, Korea University, KOREA UNIVERSITY DEPARTMENT OF MECHANICAL ENGINEERING TEAM — Platelet is a crucial blood cell on hemostasis. As platelet exposed to high shear stress, it can be activated showing morphological and functional changes to stop bleeding. When platelet is abnormal, there is high risk of cardiovascular diseases. Thus, quick and precise assay for platelet function is important in clinical treatment. In this study, we design a microfluidic system, which can test platelet function exposed with the stimulation of shear and agonists. The microfluidic system consists of three parts: 1) a shear mechanism with rotating stirrer; 2) multiple microchannels to flow samples and to stop; 3) camera-interfaced migration distance (MD) analyzing system. When sheared blood is driven by pressure through the microchannel, shear-activated platelets adhere to a collagen-coated surface, causing blood flow to significantly slow and eventually stop. As the micro-stirrer speed increases, MD decreases exponentially at first, but it increases beyond a critical rpm after all. These results are coincident with data measured by FACS flowcytometry. These results imply that the present system could quantitatively measure the degree of activation, aggregation and adhesion of platelets and that blood MD is potent index for measuring the shear-dependence of platelet function.

KP1.00030 Visualization of pulsatile flow for magnetic nanoparticle based therapies1, ANDREW WENTZEL, PHILIP YECKO, The Cooper Union for the Advancement of Science and Art — Pulsatile flow of blood through branched, curved, stenosed, dilated or otherwise perturbed vessels is more complex than flow through a straight, uniform and rigid tube. In some magnetic hyperthermia and magnetic chemo-therapies, localized regions of magnetic nanoparticle laden fluid are deliberately formed in blood vessels and held in place by magnetic fields. The effect of localized magnetic fluid regions on blood flow and the effect of the pulsatile blood flow on such magnetic fluid regions are poorly understood and difficult to examine in vitro or by numerical simulation. We present a laboratory model that facilitates both dye tracer and particle imaging velocimetry (PIV) studies of pulsatile flow of water through semi-flexible tubes in the presence of localized magnetic fluid regions. Results on the visualization of flows over a range of Reynolds and Womersley numbers and for several different (water-based) ferrofluids are compared for straight and curved vessels and for different magnetic localization strategies. These results can guide the design of improved magnetic cancer therapies.

1Support from the William H. Sandholm Program of Cooper Union’s Kanbar Center for Biomedical Engineering is gratefully acknowledged.

KP1.00031 BOUNDARY LAYERS –

KP1.00032 Progress Towards an LES Wall Model Including Unresolved Roughness1, KYLE CRAFT, ANDREW REDMAN, KURT AIKENS, Houghton College — Wall models used in large eddy simulations (LES) are often based on theories for hydraulically smooth walls. While this is reasonable for many applications, there are also many where the impact of surface roughness is important. A previously developed wall model1 has been used primarily for jet engine aeroacoustics. However, jet simulations have not accurately captured thick initial shear layers found in some experimental data1. This may partly be due to nozzle wall roughness used in the experiments to promote turbulent boundary layers. As a result, the wall model is extended to include the effects of unresolved wall roughness through appropriate alterations to the log-law. The methodology is tested for incompressible flat plate boundary layers with different surface roughness. Correct trends are noted for the impact of surface roughness on the velocity profile. However, velocity deficit profiles and the Reynolds stresses do not collapse as well as expected. Possible reasons for the discrepancies as well as future work will be presented.

1This work used the Extreme Science and Engineering Discovery Environment (XSEDE), which is supported by National Science Foundation number ACI-1053575. Computational resources on TACC Stampede were provided under XSEDE allocation ENG150001.

KP1.00033 A Test of the Validity of Inviscid Wall-Modeled LES1, ANDREW REDMAN, KYLE CRAFT, KURT AIKENS, Houghton College — Computational expense is one of the main deterrents to more widespread use of large eddy simulations (LES). As such, it is important to reduce computational costs whenever possible. In this vein, it may be reasonable to assume that high Reynolds numbers flows with turbulent boundary layers are inviscid when using a wall model. This assumption relies on the grid being too coarse to resolve either the viscous length scales in the outer flow or those near walls. We are not aware of other studies that have suggested or examined the validity of this approach. The inviscid wall-modeled LES assumption is tested here for supersonic flow over a flat plate on three different grids. Inviscid and viscous results are compared to those of another wall-modeled LES as well as experimental data — the results appear promising. Furthermore, the inviscid assumption reduces simulation costs by about 25% and 39% for supersonic and subsonic flows, respectively, with the current LES application1. Recommendations are presented as are future areas of research.

1This work used the Extreme Science and Engineering Discovery Environment (XSEDE), which is supported by National Science Foundation number ACI-1053575. Computational resources on TACC Stampede were provided under XSEDE allocation ENG150001.

KP1.00034 Micro-Scale Simulation of Water Flow in Porous Media Coupled with Phase Change. SAHAND ETEMAD, ARASH BEHRANG, PEYMAN MOHAMMADNIAZ, HOSSEIN HEJAZI, APOSTOLOS KANTZAS, Chemical and Petroleum Engineering University of Calgary, Calgary, CANADA — Sub-pore scale modeling of flow in porous media is gaining momentum. The concept of Digital Core Analysis deals with measurements of virtual core and the purpose of such modeling is to replace conventional and special core analysis when the latter are not feasible. Single phase flow phenomena are nowadays fairly easy to model given a good representation of the porous medium by its digital counterpart. Two phase flow modeling has proven more difficult to represent due to the complexities introduced by the insert of interfaces. These problems were at least partially overcome by the implementation of the “Volume of Fluid” method. OpenFOAM is the CFD package of choice in this work. The aforementioned approach is currently being extended in the modeling of phase change within a porous medium. Surface roughness is introduced by the incorporation of wedges of variable density and amplitude on the pore surface. A further introduced complication is that the individual grains are of different mineralogy and thus of different wettability. The problem of steam condensation in such media is addressed. It is observed that steam condenses first in the smallest of wedges, which act as a nucleation sites. Water spreads on water-wet surfaces. Snap-off is observed in several cases leading to temporary trapping of vapor. Grid size effects are also addressed. The application of this modeling effort is the condensation of steam in thermal recovery methods.

KP1.00035 The scaling laws for the energy-containing range of second-order structure functions above a dense vegetation canopy. YING PAN, National Center for Atmospheric Research, MARCELO CHAMECKI, Pennsylvania State University — Theoretical and experimental results show that the energy-containing range of second-order streamwise spatial structure function within the logarithmic layer of moderate- and high-Reynolds-number wall turbulence is scaled by the dissipation length scale. We extend these scaling laws for turbulent flows above a dense vegetation canopy, where the structure of turbulence is more analogous to a free shear layer than a wall boundary layer. The imbalance between production and dissipation of turbulent kinetic energy (TKE) within the canopy shear layer is much greater than that within the logarithmic layer of wall turbulence. For evaluation of the scaling laws, we use large-eddy simulation (LES) results that well reproduce field experimental data of the second-order streamwise temporal structure function of filtered velocity above the canopy. Within the shear layer above the canopy, LES results of second-order streamwise spatial structure function of filtered velocity are correctly scaled by the dissipation length scale, confirming the theoretical extension of the scaling laws. This work is a preliminary step towards universal scaling laws for turbulent shear flows.

KP1.00036 BUBBLES

KP1.00037 Investigation of Manipulation Technique of Microbubbles Using Focused Ultrasound. TAICHI OSAKI, The University of Tokyo, KAZUHITO INOUE, YOICHIRO MATSUMOTO, Retired, SHU TAKAGI, TAKASHI AZUMA, The University of Tokyo, MITSUHISA ICHIYANAGI, Sophia University — Recently, it has been thought that the application of ultrasound and microbubbles (MB) is medical technology. Should MB be manipulated contactlessly, it will contribute to the mechanism investigation on the drug delivery system using MB as drug carrier. However no technique has yet to be established that can trap MB at any desired position, manipulate them along any desired path. Accordingly in this research, we investigated whether it was possible to trap MB at desired position, manipulate them along desired paths through experiments aimed at the development of MB manipulation tools that utilize ultrasound. Moreover, we analyzed the microbubble behaviors in ultrasound field. Bubbles in the ultrasound wave field are subjected to the primary Bjerknes force. Our method aimed that MB are trapped at the antinode or the node and manipulated with moving the antinode or node. We fabricated a concave transducer which radiates focused ultrasound and used sonazoid as MB and they were trapped at the focus as a cluster. The transducer moves its own position to move its focus and manipulate MB. Besides, we observed the trapped cluster with several incident frequencies. MB were trapped and manipulated along a locus of alphabet “M” about 100 m. From this result, it is implied that MB can be manipulated along any desired path. Moreover, there was the inverse correlation between the trapped cluster size and the incident frequency.

KP1.00038 Propagation and Dissolution of CO₂ bubbles in Algae-Photo-bioreactors. SRINIVAS KOSARAJU, Northern Arizona University — Research grade photo-bioreactors are used to study and cultivate different algal species for biofuel production. In an attempt to study the growth properties of a local algal species in rain water, a custom made bioreactor is designed and being tested. Bio-algae consumes dissolved CO₂ in water and during its growth cycle, the consumed CO₂ must be replenished. Conventional methods use supply of air or CO₂ bubbles in the growth medium. The propagation and dissolution of the bubbles, however, are strongly dependent on the design parameters of the photo-bioreactor. In this paper, we discuss the numerical modeling of the air and CO₂ bubble propagation and dissolution in the photo-bioreactor. Using the results the bioreactor design will be modified for maximum productivity.

KP1.00039 Coalescence preference in densely packed bubbles. YESEUL KIM, SU JIN LIM, SKKU Advanced Institute of Nanotechnology (SAINT), Sungkyunkwan University, BOPIL GIM, Department of Bio and Brain Engineering, Korea Advanced Institute of Science and Technology (KAIST), BYUNG MOOK WEON, School of Advanced Materials Science and Engineering, SKKU Advanced Institute of Nanotechnology (SAINT), Sungkyunkwan University — Coalescence preference is the tendency that a merged bubble from the contact of two original bubbles (parent) tends to be near to the bigger parent. Here, we show that the coalescence preference can be blocked by densely packing of neighbor bubbles. We use high-speed high-resolution X-ray microscopy to clearly visualize individual coalescence events which occur on a time scale seconds and inside dense packing of microbubbles with a local packing fraction of ~ 40%. Previous theory and experimental evidence predict a power of -5 between the relative coalescence position and the parent size. However, our new observation for coalescence preference in densely packed microbubbles shows a different power of -2. We believe that this result may be important to understand coalescence dynamics in dense packing of soft matter.

KP1.00040 Experimental technique for observing free oscillation of a spherical gas bubble in highly viscous liquids. TAKEHIRO NAKAJIMA, KEITA ANDO, Department of Mechanical Engineering, Keio University — An experimental technique is developed to observe free oscillations of a spherical gas bubble in highly viscous liquids. It is demonstrated that focusing a nanosecond laser pulse of wavelength 532 nm and energy up to 1.5 mJ leads to the formation of a spherical gaseous bubble, not a vaporous bubble (quickly condensed back to the liquid), whose equilibrium radius is up to 200 microns in glycerin saturated with gases at room temperature. The subsequent free oscillations of the spherical gas bubble is visualized using a high-speed camera. Since the oscillation periods are short enough to ignore bubble translation under gravity and mass transfer out of the bubble, the observed bubble dynamics can be compared to nonlinear and linearized Rayleigh-Plesset-type calculations that account for heat conduction and acoustic radiation as well as the liquid viscosity. In this presentation, we report on the measurements with varying the viscosity and comparisons to the theory to quantify damping mechanisms in the bubble dynamics.
KP1.00041 High fidelity simulation of nucleate boiling and transition to critical heat flux on enhanced structures, MIAH YAZDANI, ABBAS ALAYHYARI, THOMAS RADCLIFF, MARIOS SOTERIOU, United Tech Res Ctr — Surface enhancement is often the primary approach for improved heat transfer performance of two-phase thermal systems particularly when they operate in nucleate boiling regime. This paper exploits the modeling capability developed by Yazdani et. al. for simulation of nucleate boiling and transition to critical heat flux to study the nucleation phenomenon on various enhanced structures. The multi-scale of two-phase flow associated with boiling phenomena is addressed through combination of deterministic CFD for the macro-scale transport, asymptotic based representation of micro-layer, and stochastic representation of surface roughness so as to allow a high-fidelity simulation of boiling on an arbitrary surface. In addition, given the excessive complexity of surface structures often used for enhancement of boiling heat transfer, a phase-field-based method is developed to generate the structures where the numerical parameters in the phase-field model determine the topology of a given structure. The “generated” structure is then embedded into the two-phase flow model through virtual boundary method for the boiling simulation. The model is validated against experimental data for the boiling curve and the critical heat flux as well as nucleation and bubble dynamics characteristics.

KP1.00042 COMPRESSIBLE FLOW

KP1.00043 Turbulent energy flux generated by shock/homogeneous-turbulence interaction, KRISHNENDU SINHA, RUSSELL QUADROS, Department of Aerospace Engineering, Indian Institute of Technology Bombay, JOHAN LARSSON, Department of Mechanical Engineering, University of Maryland, College Park — High-speed turbulent flows with shock waves are characterized by high localized surface heat transfer rates. Computational predictions are often inaccurate due to the limitations in modeling of the unclosed turbulent energy flux in the highly non-equilibrium regions of shock interaction. In this paper, we investigate the turbulent energy flux generated when homogeneous isotropic turbulence passes through a nominally normal shock wave. We use linear interaction analysis where the incoming turbulence is idealized as being composed of a collection of two-dimensional planar vorticity waves, and the shock-wave is taken to be a discontinuous wave. Turbulent energy flux is predicted to be strongly dependent on the incidence angle of the incoming waves. The energy flux correlation is also decomposed into its vortical, entropy and acoustic contributions to understand its rapid non-monotonic variation behind the shock. Three-dimensional statistics, calculated by integrating two-dimensional results over a prescribed upstream energy spectrum, are compared with available direct numerical simulation data. A detailed budget of the governing equation is also considered in order to gain insight into the underlying physics.

KP1.00044 COMPUTATIONAL FLUID DYNAMICS

KP1.00045 Effect of the Convected Terms in the Transient Viscoelastic Flow, NARIMAN ASHRAPI, MEYSAM MOHAMADALI, Young Researchers and Elites Club, Science and Research Branch, Islamic Azad University, Tehran, Iran — The influence of fluid elasticity is examined for the plane Couette flow (PCF) of a Johnson Segalman (J.S) fluid. The model takes into account the interrelations of velocity gradients and stress components through introduction of appropriate coefficients in the elastic terms of constitutive equation. The flow field is obtained from the conservation and constitutive equations using the Galerkin projection method. Both inertia and normal stress effects are included. Effect of several values of governing parameters such as introduced coefficients, Reynolds number and Wiessenberg number on velocity and normal and shear stresses profiles are explored in detail. The results show that the oscillating behavior of the flow is increased to grow as the coefficients increase. The shear stress increases in terms of the convection terms and the flow properties. For higher Reynolds the shear stress reaches a maximum and then decreases to minimum. From a numerical point of view, the model also allows for the velocity and stress components to be represented by truncated series.

KP1.00046 High-performance and high-order numerical methods for 2D Navier-Stokes equations, VINICIUS HENRIQUE AURICHO, ATTILIO CUCCHIERI, MARIA LUISA BAMBOZZI DE OLIVEIRA, Univ de Sao Paulo - dup record — Since numerical simulation of a flow is a computationally-intensive problem, our main goal is to develop numerical methods - to solve the fluid equations of motion (compressible Navier-Stokes) in 2D - that are also suitable for the high-performance computing framework. We study known methods, such as flux-splitting, MacCormack, and compact schemes, to guide our search. In particular, we consider some high-order versions of these methods, since they allow for high-resolution with less grid points, possibly reducing the computation times. Our effort is focused on obtaining shock-capturing, multiscale, low-numerical dissipation methods.

1 CNPq-Brazil

KP1.00047 Simulation of High Re Boundary Layer Flows on Uniform Grids Using Immersed Boundaries with Vorticity Confinement, SUBHASHINI CHITTA, JOHN STEINHOFF, Wave CPC, Inc. — This paper describes the use of Vorticity Confinement (VC) to efficiently treat complex blunt bodies with thin shed vortex sheets and attached boundary layers. Because these flows involve turbulence in the vortical regions, there is currently no ab initio method to treat them on current or foreseeable computers. In fact, in spite of years of turbulence modeling efforts (such as LES or RANS), serious flaws in aerodynamic design involving vortex shedding may still be left undetected until the expensive prototype or production stage. Our basic premise is that, for a class of real-world problems requiring simulating ensembles of flow conditions for overall accuracy, conventional turbulence models suffer cost constraints. For these reasons, VC is used to rapidly simulate many operating conditions, as is often done in expensive testing programs for flying prototypes, and in realistic simulations. To achieve dramatically lower computational cost, VC treats the entire flow in a uniform, coarse grid with solid surfaces “immersed” in the grid so that they can be quickly generated for many configurations with no requirement for adaptive or conforming fine grids. Also, the VC method has the efficiency of panel methods, but the generality and ease of use of Euler equation methods.

1 We would like to thank Dr. Frank Caradonna for his suggestions and support.

KP1.00048 Numerical analysis of rough wall effect on lid-driven cavity flow using Lattice Boltzmann Method, ARMAN SAFDARI, Pusan Natl Univ, S. M. REZA ATTARZADEH, Concordia Univ, KYUNG CHUN KIM, Pusan Natl Univ — In this paper, the numerical investigation of two dimensional incompressible flow in a lid-driven cavity with series of squared roughness on the basal wall is carried out. Understanding the dynamic of fluid-particles interaction is of interest in different industrial applications such as sedimentation process. Two numerical methods are applied for validation purpose: Transient modeling based on Finite Volume Method, and Lattice Boltzmann method based on the discrete Boltzmann equation. The flow field is investigated for range of Reynolds number: 100, 700 and 1000 using a fine grid mesh around the roughened wall. The effect of wall-roughness as an influencing parameter on formation of the central vortex inside the cavity is investigated. It is shown that the size of the downstream secondary eddies become smaller with either increasing Reynolds number or increasing the number of roughed features. Dominant effect of secondary eddies were observed by increasing the size of the wall roughness which mimics the influence of sedimented particles inside a cavity. Interesting features of the flow, cavity features and the boundary conditions are discussed in details.
KP1.00049 Large-eddy simulation of vortex streets and dispersion behind high-rise buildings, BEOM-SOOK HAN, Seoul Natl Univ, SEUNG-BU PARK, Columbia university, JONG-JIN BAIK, Seoul Natl Univ — Understanding flow and dispersion in densely built-up urban areas is one of the important problems in the field of urban fluid mechanics. Nowadays, sophisticated numerical models and high-resolution urban morphology data enable us to study detailed flow structures in real urban areas. Simulations with high-resolution urban morphology data show very complex flow structures in several studies. Here, we examine turbulent flow patterns and associated pollutant dispersion near and, particularly, behind high-rise buildings using the parallelized large-eddy simulation model (PALM) and high-resolution urban morphology data. The study area selected is a highly built-up area of Seoul, South Korea. It is shown that turbulent wakes are produced behind high-rise buildings and vortex streets appear in the places where turbulent wakes occur. The vortex street seems to be related to strong updrafts and ejections that appear downwind of high-rise buildings. The vortex street is found to affect pollutant dispersion. Various factors that influence the evolution and structure of vortex streets will be presented and discussed along with involved dispersion mechanisms.

KP1.00050 A computational framework for the quantification of rare events in systems with instabilities, THEMISTOKLIS SAPSIS, MUSTAFA MOHAMAD, WILL COUSINS, MIT — We consider the problem of probabilistic quantification of dynamical systems exhibiting heavy tailed distributions. These heavy tail features are associated with rare transient re-sponces due to the occurrence of internal instabilities. Systems with these characteristics can be found in a variety of areas including mechanics, fluids, and waves. Here we are interested for the development of a computational approach, a probabilistic-decomposition-synthesis method that will take into account the nature of these internal instabilities and will inexpenisvely provide the non-Gaussian probability density function for the quantities of interest. Our approach relies on the decomposition of the statistics to a stable Gaussian core and a heavy-tailed distribution. Statistics in the stable region are analytically characterized using a Gaussian approximation approach, while the non-Gaussian distributions associated with the intermittently unstable region of the phase space, are inexpenisively quantified through reduced-order methods. Applications are presented for nonlinear water waves as well as subjected mechanical systems.

1Supported by ONR Grant # N000141410520.

KP1.00051 CONVECTION AND BUOYANCY DRIVEN FLOWS —

KP1.00052 Validity of classical scaling laws in laminar channel flow with periodic spacer-like obstacles, WILKO ROHLFS, RWTH - Aachen, JOHN H. LIENHARD, MIT - Massachusetts Institute of Technology — Laminar channel flows with periodic obstacles occur in different technical applications involving heat and mass transfer. They are present in membrane technologies such as electro-dialysis or spirally wound membrane modules. For process design, classical scaling laws of heat and mass transfer are typically used. The laws scale the transfer (Sherwood) number, Sh, to the hydrodynamic Reynolds, Re, the fluid specific Schmidt number, Sc, and to some dimensionless geometric parameters, G, in a classical form like $Sh = C_{Re}^{n}Sc^{2}G^{m}$. However, the validity of those classical scaling laws is limited to the region where the concentration boundary layer develops as it is well known that the transfer numbers approach a constant (Reynolds and Schmidt independent) value in the developed region of a laminar channel flow. This study examines numerically the validity of the scaling laws if the channel flow is interrupted periodically by cylindrical obstacles of different size and separation distance. In the developed region, a Schmidt and Reynolds number dependency is found and associated to wall-normal flow induced by the obstacles, for which this dependency varies with obstacle size and separation distance.

1Funding for WR was provided by the German Academic Exchange Service DAAD

KP1.00053 Numerical Modeling of Surface and Volumetric Cooling using Optimal T- and Y-shaped Flow Channels, SRINIVAS KOSARAJU, Northern Arizona University — The T- and Y-shaped flow channels can be optimized for reduced pressure drop and pumping power. The results of the optimization are in the form of geometric parameters, such as length and diameter ratios of the stem and branch sections. While these flow channels are optimized for minimum pressure drop, they can also be used for surface and volumetric cooling applications such as heat exchangers, air conditioning and electronics cooling. In this paper, we studied the heat transfer characteristics of multiple T- and Y-shaped flow channel configurations using numerical simulations. All configurations are subjected to same pumping power and heat generation constraints and their heat transfer performance is studied.

KP1.00054 Numerical Modeling and Optimization of Warm-water Heat Sinks, YASER HADAD, PAUL CHIAROT, State University of New York at Binghamton — For cooling in large data-centers and supercomputers, water is increasingly replacing air as the working fluid in heat sinks. Utilizing water provides unique capabilities; for example: higher heat capacity, Prandtl number, and convection heat transfer coefficient. The use of warm, rather than chilled, water has the potential to provide increased energy efficiency. The geometric and operating parameters of the heat sink govern its performance. Numerical modeling is used to examine the influence of geometry and operating conditions on the key metrics such as thermal and flow resistance. This model also facilitates studies on cooling of electronic chip hot spots and failure scenarios. We report on the optimal parameters for a warm-water heat sink to achieve maximum cooling performance.

KP1.00055 Buoyancy and blockage effects on transient laminar opposing mixed convection heat transfer from two horizontal confined isothermal cylinder in tandem, LORENZO MARTÍNEZ-SUÁSTEGUI, ESIEME Azcapotzalco, Instituto Politécnico Nacional, ERICK SALCEDO, Departamento de Termofluidos, Facultad de Ingeniería, UNAM, JUAN CAJAS, Barcelona Supercomputing Center (BSC-CNS), Edificio NEXUS 1, Campus Nord UPC, Gran Capitán 2-4, CÉSAR TREVÍNO, UMDI, Facultad de Ciencias, Universidad Nacional Autónoma de México, SISAL — Transient mixed convection in a laminar cross-flow from two isothermal cylinders in tandem arrangement confined inside a vertical channel is studied numerically using the vorticity-stream function formulation of the unsteady two-dimensional Navier-Stokes and energy equations. Numerical experiments are performed for a Reynolds number based on cylinder diameter of $Re = 200$, Prandtl number of $Pr = 7$, blockage ratio of $D/H = 0.2$, a pitch-to-diameter ratio of $L/D = 2$, and several values of buoyancy strength or Richardson number $Ri = Gr/Re^2$. The results reported herein demonstrate how the wall confinement, interference effects and opposing buoyancy affect the flow structure and heat transfer characteristics of the cylinder array.

1This research was supported by the Consejo Nacional de Ciencia y Tecnología (CONACYT), Grant number 167474 and by the Secretaria de Investigacion y Posgrado del IPN, Grant number SIP 20141309.

KP1.00056 DROPS —
Impact of a single drop on the same liquid: formation, growth and disintegration of jets, G. GILOU ABAGLAH, Cornell University, ROBERT DEEGAN, University of Michigan — One of the simplest splashing scenarios results from the impact of a single drop on the same liquid. The traditional understanding of this process is that the jetting phenomenon is more complicated than expected because multiple jets can be generated from a single impact event and there are bifurcations in the multiplicity of jets. First, we study the formation, growth and disintegration of jets following the impact of a drop on a thin film of the same liquid using a combination of numerical simulations and linear stability theory. We obtain scaling relations from our simulations and use these as inputs to our stability analysis. We also use experiments and numerical simulations of a single drop impacting on a deep pool to examine the bifurcation from a single jet into two jets. Using high speed X-ray imaging methods we show that vortex separation within the drop leads to the formation of a second jet long after the formation of the ejecta sheet.

Aqueous Polymer in water alter the Coffee-ring effect, CHANGDEOK SEO, DAEO JANG, School of mechanical engineering, Korea University, WONHWI NA, Department of Micro/Nano systems, Korea University, SERA PARK, SEHYUN SHIN, School of mechanical engineering, Korea University — When evaporating in droplet system, small particles move toward an edge by outward capillary flow. This phenomenon is known as coffee-ring effect. In experiments that are required to uniformly accumulate particles, this effect can be fatal. In spite of recent challenges for suppressing the coffee-ring effect, it is still insufficiently controlled in film and droplet with various solutions. For deliberate applications, various materials have been used to control the initial spreading of colloidal particles. In this research, we used a bio-compatible and aqueous polymer, polyethylene glycol (PEG) for altering the coffee-ring effect. The influence of PEG on the evaporation of drying colloidal droplets is examined in a wide range of initial concentrations. Adding PEG to water causes a strong vortex flow near the edge of droplet and subsequently leads to significantly uniform patterns of colloidal particle deposition after evaporation. We found the vortex phenomenon by combination of radially outward capillary flow and radially inward Marangoni flows are induced by the radial variation of polymer concentration along the air/water interface. Furthermore, increasing polymer concentration significantly alters the characteristic of Marangoni Vortex and leads to reproducible patterning of conical structures.

Superhydrophobic-like tunable droplet bouncing on slippery liquid interfaces, CHONGLEI HAO, ZUANKAI WANG, City Univ of Hong Kong — Droplet impacting on solid or liquid interfaces is a ubiquitous phenomenon in nature. Although complete rebound of droplets is widely observed on superhydrophobic surfaces, the bouncing of droplets on liquid is usually due to the easy collapse of entrapped air pocket underneath the impinging droplet. Here, we report a superhydrophobic-like bouncing regime on thin liquid film, characterized by the contact time, the spreading dynamics, and the restitution coefficient independent of underfilm liquid film thickness. Through an experimental and theoretical analysis, we demonstrate that the manifestation of such a superhydrophobic-like bouncing necessitates an intricate interplay between the Weber number, the thickness and viscosity of liquid film. Such insights allow us to tune the droplet behaviors in a well-controlled fashion. We anticipate that the combination of superhydrophobic-like bouncing with inherent advantages of emerging slippery liquid interfaces will find a wide range of applications.

Effect of surface morphology on anti-icing, DUCK-GYU LEE, THANH-BINH NGUYEN, WAN-DOO KIM, HYUNEUI LIM, Korea Institute of Machinery & Materials — A water drop on a sub-cooled surface undergoes solidification, and it is well known that the anti-icing effects such as delayed freezing time and low adhesion force are determined by surface morphology. To quantify the effect of surface morphology and the effect of surfactant treatment on the surface behavior, we first theoretically predict the freezing time of a water drop on a sub-cooled micro patterned substrate and show that the time delay is in good agreement with experimental results. Then we develop a simple theory for the work of adhesion upon consideration the substrate geometrical condition in order to prevent it from being broken due to the adhesion. Finally, we provide the morphological conditions for the pattern under which the freezing time delay is maximized and the work of adhesion is minimized.

Spreading of Electrolyte Drops on Charged Surfaces: Electric Double Layer Effects on Drop Dynamics, KYEONG BAE, SHAYANDEV SINHA, GUANG CHEN, SIDDHARTHA DAS, Univ of Maryland-College Park — Drop spreading is one of the most fundamental topics of wetting. Here we study the spreading of electrolyte drops on charged surfaces. The electrostatic force in contact with the charged solid triggers the formation of an electric double layer (EDL). We develop a theory to analyze how the EDL affects the spreading dynamics. The drop dynamics is studied by probing the EDL effects on the temporal evolution of the contact angle and the base radius (r). The EDL effects are found to hasten the spreading behaviour — this is commensurate to the EDL effects causing a ‘phlic’ tendency in the drops (i.e., drops attaining a contact angle smaller than its equilibrium value), as revealed by some of our recent papers. We also develop scaling laws to illustrate the manner in which the EDL effects make the r versus time (t) variation deviate from the well known r ∼ tⁿ variation, thereby pinpointing the attainment of different EDL-mediated spreading regimes.

Electrokinetic Flows —
KP1.00065 Electric field control of a fluid transfer between freely suspended and sessile droplets, SUHWHAN CHOI, ALEXEI SAVELEV, North Carolina State University — This work explores direct fluid transfer between microdroplets using liquid bridges stabilized by ac electric field. Experiments are performed with freely and sessile microdroplets of pure glycerol and water with dye. The droplets are placed along electric field directions in a cell with parallel plate electrodes filled with silicone oil. The electrical conductivity of droplets is changed from 1 to 200 μS/cm by adding dye solutions. Liquid bridges interconnecting two microdroplets can be created using an alternating electric field from 0.3 to 0.7 kV/mm with a frequency of 10.3 kHz. For such bridging fluid can be transferred through the liquid bridge from one droplet to another due to the pressure difference. The process is recorded using a CCD camera. The fluid flowrates in the range from ~100 to 10 nL/s are recorded with different electric fields and liquid conductivity. We propose that the manipulation of the liquid bridge will be the method in which small fluid volumes are dispensed.

KP1.00066 The capacitance of ionic liquid electric double layer near nanostructured electrodes, YUN SUNG PARK, MYUNG MO AHN, IN SEOK KANG, Pohang Univ of Sci & Tech — The electric double layer capacitators with nanostructured electrodes have attracted much attention of researchers due to their high power density and long life time. Recently, the ionic liquids are used as an electrolyte of EDLC owing to their electrochemical stability. When ionic liquids are used as an electrolyte, the interrelations between the electric double layer of ionic liquids and the nanostructured electrode must be studied. In this study, the EDLC systems with nanostructured electrodes and ionic liquids are simulated by solving the modified Poisson-Boltzmann equation proposed by Bazant, Storey, and Kornyshev (Phys. Rev. Lett. 106, 046102 (2011)) with COMSOL Multiphysics. Several electrode geometries including exohedral, endohedral and arrayed shapes with different length scales are simulated. The potential and charge distributions in the normal direction to the electrode surface are analyzed. The capacitance per unit area is obtained and compared to that of flat electrode. The structure determines the space for counter-ion packing and co-ion gathering, thus has crucial effects on electric double layer capacitance. The critical increase of capacitance with nanoscale confined space is observed with low electrode potential.

1This research was supported by Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Science, ICT and Future Planning (Grant Number: 2013R1A1A2011956).

KP1.00067 Electroosmotic Flow in Rigid and Soft Nanochannels: Effects of Solvent Polarization, LUCAS MYERS, SHAYANDEV SINHA, SIDDHARTHA DAS, Univ of Maryland-College Park — Electroosmotic (EOS) flow, triggered by the interaction of an applied electric field and the charge density gradient generated at the interface of a solid-liquid interface, has evolved as an extremely popular technique of driving liquid in micro-nanochannels. Unlike the Poisson-Boltzmann (PB) approach based analysis of the EOS transport, there is relatively little work on studying EOS flows in a framework beyond the PB approach. Here we provide a theory for the EOS transport using a Langevin-Bikeman (LB) model that simultaneously accounts for two important non-PB elements, namely solvent polarization and finite ion sizes. Our analysis reveals new non-dimensional parameters that influence the EOS flow. More importantly, we identify an effective electric double layer (EDL) thickness that dictates the flow characteristics. The central finding of our calculations is that for the realistic set of parameters, non-PB influences always enhances the electroosmotic flow. In the next part of the study, we highlight the non-trivialities associated with the case where the nanochannels become “soft,” i.e., the nanochannel walls are grafted with polyelectrolyte layers that affect both the electrostatic potential distribution as well as the drag force associated with the electroosmotic flow.

KP1.00068 Instantaneous velocity measurement of AC electroosmotic flows by laser induced fluorescence photobleaching anemometer with high temporal resolution, WEI ZHAO, FANG YANG, Department of Mechanical Engineering, University of South Carolina, Columbia, RUI QIAO, Department of Mechanical Engineering, Virginia Tech, GUIREN WANG, Department of Mechanical Engineering & Biomedical Engineering Program, University of South Carolina, Columbia, RUI QIAO COLLABORATION — Understanding the instantaneous response of flows to applied AC electric fields may help understand some unsolved issues in induced-charge electrokinetics and enhance performance of microfluidic devices. Since currently available velocimeters have difficulty in measuring velocity fluctuations with frequency higher than 1 kHz, most experimental studies so far focus only on the average velocity measurement in AC electrokinetic flows. Here, we present measurements of AC electroosmotic flow (AC-EOF) response time in microchannels by a novel velocimeter with submicrometer spatial resolution and microsecond temporal resolution, i.e. laser-induced fluorescence photobleaching anemometer (LIFPA). Several parameters affecting the AC-EOF response time to the applied electric signal were investigated, i.e. channel length, transverse position and solution conductivity. The experimental results show that the EOF response time under a pulsed electric field decreases with the reduction of the microchannel length, distance between the detection position to the wall and the conductivity of the solution. This work could provide a new powerful tool to measure AC electrokinetics and enhance our understanding of AC electrokinetic flows.

KP1.00069 Investigation of liquid properties in extended nanospaces using streaming potential/current system, KYOJIRO MORIKAWA, Tokyo Institute of Technology, YUTAKA KAZOE, CHI-CHANG CHANG, The University of Tokyo, TAKEHIKO TSUKAHARA, Tokyo Institute of Technology, KAZUMA MAWATARI, TAKEHIKO KITAMORI, The University of Tokyo — Understanding liquid properties in extended nanospace (10-1000 nm) is important for the evolution of nanofluidic devices. Liquid properties are expected to be changed by the nano-confinement, because the extended nanospace represents a transitional regime from single molecules to the bulk condensed phase. In this study, we developed non-probe measurement system of dielectric constant and electric conductivity of water in the extended nanospaces using streaming potential/current system. The results showed that dielectric constant in extended nanospaces was approximately 3 times lower than that in bulk, and that conductivity in extended nanospaces was approximately 500 times higher than that in bulk. The measured conductivity was consistent with the calculated one, which was determined using dielectric constants measured in extended nanospaces and electric double layer (EDL) model. It will be important information for nanofluidics.

KP1.00070 ENERGY —
The Hybrid Sterling Engine: boosting photovoltaic efficiency and deriving mechanical work from fluid expansion and heat capture. NATHAN BEETS, Wake Forest University, WAKE FOREST CENTER FOR NANOTECHNOLOGY AND MOLECULAR MATERIALS TEAM, FRAUNHOFER INSTITUTE COLLABORATION — Two major problems with many third generation photovoltaics is their complex structure and greater expense for increased efficiency. Spectral splitting devices have been used by many with varying degrees of success to collect more and more of the spectrum, but simple, efficient, and cost-effective setups that employ spectral splitting remain elusive. This study explores this problem, presenting a solar engine that employs stokes shifting via laser dyes to convert incident light to the wavelength bandgap of the solar cell and collects the resultant infrared radiation unused by the photovoltaic cell as heat in ethylene glycol or glycerin. When used in conjunction with micro turbines, fluid expansion creates mechanical work, and the temperature difference between the cell and the environment is made available for use. The effect of focusing is also observed as a means to boost efficiency via concentration. Experimental results from spectral scans, vibrational voltage analysis of the PV itself and temperature measurements from a thermocouple are all compared to theoretical results using a program in Mathematica written to model refraction and lensing in the devices used, a quantum efficiency test of the cells, the absorption and emission curves of the dyes used to determine the spectrum shift, and the various equations for fill factor, efficiency, and current in different setups. An efficiency increase well over 50% from the control devices is observed, and a new solar engine proposed.

**Experimental Techniques**

A rapid filtering and reconstruction method of two-dimensional image velocimetry signals using a non-iterative POD-method. JONATHAN HIGHAM, WERNHER BREVIS, CHRISTOPHER KEYLOCK, Univ of Sheffield — A method is presented, based on Proper Orthogonal Decomposition (POD), for the detection and estimation of outliers in two-dimensional signals. In experimental fluid mechanics, for a number of reasons, two dimensional data obtained using techniques such as Particle Image Velocimetry often contain outliers. The proposed methodology is based on the assumption that statistically significant outliers can be identified as abnormalities in the evolution of the temporal POD coefficients and as changes to the eigenvalues. Unlike previous methods, the estimation technique in the current method is non-iterative. It is instead dependent on a correction of a parameter introduced to search for abnormal, outlier induced magnitudes in the modal decomposition. The method is benchmarked by synthetically simulating outliers applied to two data sets: One data set is obtained experimentally using Particle Image Velocimetry; the other is based on a numerical simulation. The results demonstrate that the proposed approach is able to identify the outliers reliably and correct them with acceptable accuracy.

Development of threedimensional optical correction method for reconstruction of flow field in droplet. HAN SEO KO, YEONGHEON GIM, SEUNG-HWAN KANG, Sungkyunkwan University — A three-dimensional optical correction method was developed to reconstruct droplet-based flow fields. For a numerical simulation, synthetic phantoms were reconstructed by a simultaneous multiplicative algebraic reconstruction technique using three projection images which were positioned at an offset angle of 45. If the synthetic phantom in a conical object with refraction index which differs from atmosphere, the image can be distorted because a light is refracted on the surface of the conical object. Thus, the direction of the projection ray was replaced by the refracted ray which occurred on the surface of the conical object. In order to prove the method considering the distorted effect, reconstruction results of the developed method were compared with the original phantom. As a result, the reconstruction result of the method showed smaller error than that without the method. The method was applied for a Taylor cone which was caused by high voltage between a droplet and a substrate to reconstruct the three-dimensional flow fields for analysis of the characteristics of the droplet.

This work was supported by the Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Korean government (MEST) (No. 2013R1A2A2A01068653).

Experimental Analysis of Flow over a Highly Maneuverable Airframe. JONATHAN SPIRNAK, MICHAEL BENSON, BRENT VAN POPPEL, United States Military Academy Department of Mechanical Engineering, CHRISTOPHER ELKINS, Stanford University Department of Mechanical Engineering, JOHN EATON, Stanford Department of Mechanical Engineering, TEAM HMA TEAM — One way to reduce the collateral damage in war is by increasing the accuracy of indirect fire weapons. The Army-Research Laboratory is currently developing a Highly Maneuverable Airframe (HMA) consisting of four deflection canards to provide in-flight maneuverability while fins maintain short duration aerodynamic stability. An experiment was conducted using Magnetic Resonance Velocimetry (MRV) techniques to gather three dimensional, three-component velocity data for fluid flow over a scaled down HMA model. Tests were performed at an angle of attack of 2.3° and canard deflection angles of 0° and 2°. The resulting data serve to both validate computational fluid dynamics (CFD) simulations and understand the flow over this complex geometry. Particular interest is given to the development of the tip and inboard vortices that originate at the canard/body junction and the canard tips to determine their effects on airframe stability. Results show the development of a strong tip vortex and four weaker inboard vortices off each canard. Although the weaker inboard vortices dissipate rapidly downstream of the canard trailing edges, the stronger tip vortices persist until reaching the fins approximately six chord lengths downstream of the canard trailing edges.

Photochromic flow visualization in silicone oil for demonstrations and experiments. ENRICO FONDA, New York University, STEPHEN R. JOHNSTON, DEVESH RANJAN, Georgia Institute of Technology, KATEPALLI R. SREENIVASAN, New York University — Photochromic dyes change color when illuminated, usually with UV light. As tracers for flow visualization they are non-intrusive, can selectively color the fluid, and are suitable for complex and confined flows. Availability of cheap 405nm high-power lasers combined with advances in image acquisition and image-processing technology, make these tracers particularly effective in creating convenient and engaging educational demonstrations as well as in qualitatively exploring flow structures. We present two low-cost demonstrations: laminar-flow reversibility using a Taylor-Couette device and a thermal convection flow. We also report our experience in studying large scales in high Prandtl numbers Rayleigh-Bénard convection.
KP1.00077 Effects of Adding Nanoparticles on Boiling and Condensing Heat Transfer inside a horizontal round tube. MOHSEN SHEIKHOLESLAMI, Department of Mechanical Engineering, Babol University of Technology, Babol, Islamic Republic of Iran. MOHAMMADKAZEM SADOUGHI, Department of Mechanical Engineering, Iowa State University of Science and Technology, Ames, IA 50011, United States, HAMED SHARIATMADAR, MOHAMMAD ALI AKHAVAN-BEHABADI, School of Mechanical Engineering, College of Engineering, University of Tehran, Tehran 1439955961, Iran — An experimental investigation is performed on heat transfer evaluation of a nano-refrigerant flow during condensation and evaporation inside a horizontal round tube. Experiments are carried out for three working fluid types including: (i) pure refrigerant (R600a); (ii) refrigerant/lubricant (R600a/oil); and (iii) nano-refrigerant: refrigerant/lubricant/nanoparticles (R600a/oil/CuO). Nanoparticles are added to the lubricant and their mixture is mixed with pure refrigerant. Therefore, nano-refrigerants (R600a/oil/CuO) are prepared by dispersing CuO nanoparticles with different fractions of 0.5%, 1% and 1.5% in the baseline mixture (R600a/oil). Effects of different factors including vapor quality, mass flux, and nanoparticles on the heat transfer coefficient are examined for both of condensation and evaporation flows, separately. The results shows that maximum heat transfer augmentation of 79% and 83% are achieved by using the refrigerant/lubricant/nanoparticles mixture, in comparison with the pure refrigerant case in condensation and evaporation, respectively which are occurred for nano-refrigerant with 1.5% mass fraction in both of them.

KP1.00078 Shear velocity and wall position determination from particle image velocimetry data with seed centroid correction. JEFF HARRIS, Pennsylvania State Univ, BLAKE LANCE, Utah State University, RICHARD SKIPFTON, University of Idaho, BARTON SMITH, Utah State University — Two methods of computing the wall shear velocity from high-resolution particle image velocimetry (PIV) measurements are compared with and without a correction that accounts for seed gradient near the wall. It is crucial to know the wall position when computing the wall shear stress, but this can be difficult due to laser scatter on a wall. Furthermore, PIV is well known to be biased near walls due to seeding gradients. We compensate for these effects by replacing the cross-stream location of each vector with a value based on the centroid of the seeding in each interrogation region. The shear velocity and wall position resulting from methods outlined in the literature are presented. The boundary layer cases presented are influenced by buoyancy and the efficacy of these methods for convective flow will be discussed.

KP1.00079 Inductively Coupled Discharge and Post-Discharge with RF of Ne. NESLHAN SAHIN, MURAT TANISLI, SERCAN MERTADAM, Anadolu University — Plasma, which is the fourth state of matter, can be produced in laboratories in different ways. In this study, the inductively radio frequency (RF) plasma at low pressure in the quartz glass reactor prepared for special conditions is obtained. This generated plasma is non-local thermodynamics plasma and it includes different particles such as positive ions, electrons and neutral particles. Inductively coupled neon’s discharge and post-discharge characteristic properties are examined with optical emission spectroscopy (OES) and then OES is used for determining electron temperature. Differences between discharge and post-discharge zone under the same conditions are obtained. It is investigated how gas filling pressure, the applied RF power to gases and gas flowing rate affect neon inductively coupled RF discharge and post-discharge.

KP1.00080 FREE-SURFACE FLOWS —

KP1.00081 Generating a soliton splash through variational modelling and experiments. ANNA KALOGRIOU, ONNO BOKHOVE, University of Leeds — Mathematical modelling of water waves in tanks with wave generators is demonstrated by investigating variational methods asymptotically and numerically. A reduced potential flow water wave model is derived using variational techniques, which is based on the assumptions of waves with small amplitude and large wavelength. This model consists of a set of modified Benney-Luke equations describing the deviation from the still water surface $\eta(x, y, t)$ and the bottom potential $\Phi(x, y, t)$, and includes a time-dependent gravitational potential mimicking a removable “sluice gate”. The asymptotic model is solved numerically using the automated system Firedrake. In particular, a (dis)continuous Galerkin finite element method is used, together with symplectic integrators for the time discretisation. As a validation, the numerical results are compared to a soliton splash experiment in a long water channel with a contraction at its end, resulting after a sluice gate is removed at a finite time.

KP1.00082 FLOW CONTROL —

KP1.00083 Effect of synthetic roughness on a turbulent channel flow. JAVIER COMBARIZA, JESUS RAMIREZ-PASTRAN, CARLOS DUQUE-DAZA, Department of Mechanical & Mechatronics Engineering, Universidad Nacional de Colombia, Bogota, Colombia — A turbulent channel flow featuring single step synthetic roughness at the bottom wall was examined using numerical experiments. An incompressible flow solver using LES as turbulence model was employed to study some turbulence variables as well as Q-criterion coherent structures. Roughness was attained by inserting a small step, stretching along the channel spanwise direction in the bottom wall. Three different values for the step width were used. Correlations between the steps width and skin-friction coefficients are calculated. Coherent structures using the Q-Criterion are constructed using different threshold values. By examining the evolution of the Q-structures, the effect of the perturbation is characterized in the near-wall region. Consistency between skin-friction coefficients and Q-structures evolution trends is observed in each case. Comparison between some of the TKE terms in the modified channel flow and those in a smooth channel flow, allow to identify the effect of the synthetic roughness on the turbulent behaviour. Finally, a simple description of the overall effect of the presence of the perturbation on the turbulent flow is brought about by associating the Q-structures with the strong recirculation zones formed in the near-wall region close to the steps.

KP1.00084 Sweeping and redevelopment of longitudinal vortices in a turbulent boundary layer by injecting bubble swarms. HYUN JIN PARK, YUJI TASAKA, YUICHI MURAI, Hokkaido Univ — We performed injection of bubble swarms, which consist of leading air films and following smaller bubbles, into a turbulent channel flow to investigate interaction between turbulent vortices and small air films. Advection of the air films flowing along the channel wall is faster than the streamwise vortices in a turbulent boundary layer, and thus the vortices in the boundary layer are swept by the air films. Our question is what happens on the vortices after sweeping? We visualized the vortices, and it elucidated that the swept vortices survive beneath the air films. After that the redevelopment of the vortices occurs and original condition of the boundary layer is restored. Reduction of Reynolds shear stress is still continued even beneath smaller bubbles in middle part of the bubble swarms and it suggests that redevelopment of Reynolds shear stress event, bursting of streamwise vortices mainly, cannot occur quickly even with survival of the vortices. As a result, the bubble swarms reduce frictional drag more than continuously injected bubbles at the same volume fraction of bubbles.
KP1.00085 Reduction of aerodynamic friction drag of moving bodies using a Microwave-Dielectric-Barrier-Discharge actuator controlling the boundary layer. THIERRY PIERRE, CNRS Paris & Marseille — A new plasma device named M-DBD (Microwave Dielectric Barrier Discharge) is used for controlling the boundary layer in order to reduce the drag force. A compact resonant UHF structure comprising a resonant element in the form of a quarter-wave antenna creates a mini-plasma insulated from the UHF electrodes by mica sheets. Additional electrodes induce an electric field in the plasma and transiently move the ions of the plasma. The high collision rate with the neutral molecules induce the global transient flow of the neutral gas. The temporal variation of the applied electric field is chosen in order to obtain a modification of the local boundary layer. First tests using an array of M-DBD plasma actuators are underway (see Patent ref. WO 2014111469 A1).

KP1.00086 FLOW INSTABILITY —

KP1.00087 Transition in Hypersonic Boundary Layers: Role of Dilatational Waves. YIDING ZHU, CHUANHONG ZHANG, QING TANG, HUJING YUAN, JIEZHI WU, SHIYI CHEN, CUNBIAO LEE, Peking University, MOHAMED GAD-EL-HAK, Virginia Commonwealth University — Transition and turbulence production in a hypersonic boundary layer is investigated in a Mach 7.9 quiescent tunnel using Rayleigh-scattering visualisation, fast-response pressure measurements, and particle image velocimetry. It is found that the second-mode instability is a key modulator of the transition process. Although the second mode is primarily an acoustic wave, it causes the formation of high-frequency vortical waves. While the growing acoustic wave itself is rapidly annihilated due to its large and sharp dissipation peak that is enhanced by the bulk viscosity, the acoustically generated high-frequency vortical wave keeps growing and triggers a fast transition to turbulence.

KP1.00088 Effect of surfactant on kinetics of thinning of capillary bridges1. EMILIA NOWAK, NINA KOVALCHUK, MARK SIMMONDS, University of Birmingham — Kinetics of thinning of capillary bridges is of great scientific and industrial interest being of vital importance for example in various emulsification and microfluidic processes. It is well known that the rate of bridge thinning is proportional to the interfacial tension. Therefore it is expected that the process should slow down by addition of surfactant. The kinetics of capillary bridges in the presence of surfactant was studied by the dripping of liquid from a capillary tip under conditions of nearly zero flow rate (We < 1). The tested liquids were aqueous solutions of sodium lauryl ether sulphate (SLES), which is broadly used in personal care products. The viscosity, surfactant activity and adsorption kinetics have been controlled by addition of glycerol and sodium chloride. The paper has shown that the kinetics of capillary bridges are determined by dynamic surface tension rather than by its equilibrium value. In particular, the kinetics of the bridge thinning for the 0.1 g L-1 aqueous SLES solution is practically the same as that of pure water despite twice lower equilibrium surface tension.

1 EPSRC Programme Grant, MEMPHIS, EP/K0039761/1

KP1.00089 Schlieren Imaging of Gravitational Instabilities during Miscible Viscous Fingering of Glycerol-Water Systems1. DANIELA MARIN, SIMONE STEWART, PATRICK BUNTON, Department of Physics, William Jewell College, Liberty MO U.S.A., ECKART MEIBURG, Department of Mechanical Engineering, Center for Interdisciplinary Research, in Fluids University of California at Santa Barbara Santa Barbara, U.S.A., ANNE DE WIT, Nonlinear Physical Chemistry Unit, Service de Chimie Physique et Biologie Thorique, Universit Libre de Bruxelles, Brussels, Belgium — Viscous fingering occurs when a lower viscosity fluid displaces a higher viscosity fluid causing interfacial instabilities creating finger-like patterns. In a typical flow, the less viscous fluid is injected into the higher viscous fluid that is between the plates of a Hele-Shaw cell. In most cases for transparent flows, dye is dissolved into the displacing fluid in order to observe it. This work uses Schlieren imaging of miscible fluid displacements in a horizontal Hele-Shaw Cell, which reveals new information about the three-dimensional nature of VF. A Schlieren system is composed of a parallel light beam, a lens that brings the light to a focus, a cutoff of some type, and a camera. Schlieren does not require dye, ensuring the natural flow of the fluids is undisturbed. Here the imaging system is described followed by results of miscible flows of water in to aqueous glycerol solutions. Structures attributable to three-dimensional buoyancy-driven flows are readily observed. These results are interpreted in light of recent three-dimensional calculations.

1 Supported by National Science Foundation CBET-1335739

KP1.00090 Spontaneous Formation of Nanopatterns in Velocity-Dependent Dip-Coated Organic Films: From Dragonflies to Stripes. P. HUBER, Institute of Materials Physics and Technology, Hamburg University of Technology, M. BAI, Department of Physics and Astronomy, University of Missouri (USA), V. DEL CAMPO, P. HOMM, P. FERRARI, A. DIAMA, Facultad de Fisica, Pontificia Universidad Catolica de Chile, C. WAGNER, Experimental Physics, Saarland University (Germany), H. TAUB, Department of Physics and Astronomy, University of Missouri (USA), K. KNORR, Experimental Physics, Saarland University (Germany), M. DEUTSCH, Physics Department and Institute of Nanotechnology and Advanced Materials, Bar-Ilan University (Israel), M. RETAMAL, U. VOLKMANN, T. CORRALES, Facultad de Fisica, Pontificia Universidad Catolica de Chile — We present the structure of thin, n-alkane films on the oxide layer of a silicon surface, prepared by dip-coating in a n-C14H29/n-heptane solution. Electron micrographs reveal two adsorption morphologies depending on the substrate withdrawal speed v. For small v, dragonfly-shaped molecular islands are observed. For a larger v value, the islands are pulled to the withdrawal direction. These have a diameter of a few hundred micrometers and a few-micrometer lateral separation. With increasing v, the surface coverage first decreases, then increases for v > vcr ~ 0.15 mm/s. The critical vcr marks a transition between the evaporation regime and the entrainment regime. The stripes’ strong crystalline texture and the well defined separation are due to an anisotropic 2D crystallization in narrow liquid fingers, which presumably results from a Marangoni-flow-driven hydrodynamic instability in the evaporating dip-coated films.

KP1.00091 Effects of inclination and vorticity on interfacial flow dynamics in horizontal and inclined pipes. ARETI KIARA, KELLI HENDRICKSON, YUMING LIU, Massachusetts Institute of Technology — The transport of oil and gas in long horizontal pipes can be significantly affected by the development of violent roll waves and slugs, but the mechanics causing such transitions have not been well understood. To enable the improvement of the prediction of flow transition criteria in long pipelines we perform theoretical analysis and direct numerical simulations of multiphase pipe flows to quantify the roles of inclination and vorticity in the flow dynamics. We find that backflow or flooding may occur even in the absence of disturbances due to inclination effects and long pipelines we perform theoretical analysis and direct numerical simulations of the predictions for spectrum evolutions for broad-banded interfacial disturbances in inclined pipes.
 KP1.00092 Rayleigh-Taylor mixing in supernova experiments1, NORA SWISHER, Carnegie Mellon University, CAROLYN KURANZ, University of Michigan at Ann Arbor, DAVID ARNETT, University of Arizona, OMAR HURRICANE, BRUCE REMINGTON, HARRY ROBEY, Lawrence Livermore National Laboratory, SNEZHANA ABARZHI, Carnegie Mellon University — We report a scrupulous analysis of data in supernova experiments that are conducted at high power laser facilities in order to study core-collapse supernova SN1987A. Parameters of the experimental system are properly scaled to investigate the interaction of a blast-wave with helium-hydrogen interface, and the induced Rayleigh-Taylor (RT) mixing of the denser and lighter fluids with time-dependent acceleration. We analyze all available experimental images of RT flow in supernova experiments, and measure delicate features of the interfacial dynamics. A new scaling is identified for calibration of experimental data to enable their accurate analysis and comparisons. By proper accounting for the imprint of the experimental conditions, the data set size and statistics are substantially increased. New theoretical solutions are identified to describe asymptotic dynamics of RT flow with time-dependent acceleration by applying theoretical analysis. Good qualitative and quantitative agreement is achieved of the experimental data with the theory and simulations. Our study indicates that in supernova experiments, the RT flow is in the mixing regime, the interface amplitude contributes substantially to the characteristic length scale for energy dissipation; the mixing flow may keep order.

1Support of the National Science Foundation is warmly appreciated

 KP1.00093 Qualitative and quantitative features of Rayleigh-Taylor mixing dynamics, AKLANT BHOWMICK, SNEZHANA ABARZHI, Carnegie Mellon University, PRAVEEN RAMAPrabhu, VARAD KARKHANIS, University of North Carolina at Charlotte, ANDREW LAWRIE, University of Bristol, RTI COLLABORATION — We consider dynamics of Rayleigh-Taylor (RT) flow in a large aspect ratio three-dimensional domain with square symmetry in the plane for fluids with contrasting densities. In order to quantify the interface evolution from a small amplitude single-mode initial perturbation to advanced stage of RT mixing, we apply numerical simulations using the MOBILE code, theoretical analyses, including group theory and momentum model, as well as parameters describing the interplay between acceleration and turbulence. We find: In RT flow, the fluid motion is intense near the interface and is negligible far from the interface. At late times the growth rates of RT bubbles and spikes may increase without a corresponding increase of length-scales in the direction normal to acceleration. The parameters describing the interplay between acceleration and turbulence in RT mixing are shown to scale well with the flow Reynolds number and Froude number.

 KP1.00094 Exploring Model Assumptions Through Three Dimensional Simulations Using a High-order Hydro Option in the Ares Code1, JUSTIN WHITE, University of Missouri-Columbia, BRITTON OLSON, BRANDON MORGAN, Lawrence Livermore National Laboratory, JACOB MCFARLAND, University of Missouri-Columbia, LAWRENCE LIVERMORE NATIONAL LABORATORY TEAM, UNIVERSITY OF MISSOURI-COLUMBIA TEAM — This work presents results from a large eddy simulation of a high Reynolds number Rayleigh-Taylor instability and Richtmyer-Meshkov instability. A tenth-order compact differencing scheme on a fixed Eulerian mesh is utilized within the Ares code developed at Lawrence Livermore National Laboratory. (LLNL) We explore the self-similar limit of the mixing layer growth in order to evaluate the k-L-a Reynolds Averaged Navier Stokes (RANS) model (Morgan and Wickett, Phys. Rev. E, 2015). Furthermore, profiles of turbulent kinetic energy, turbulent length scale, mass flux velocity, and density-specific-volume correlation are extracted in order to aid the creation a high fidelity LES data set for RANS modeling.

1Prepared by LLNL under Contract DE-AC52-07NA27344.

 KP1.00095 Effect of Inhomogeneous Flow on Rayleigh Taylor Instability, SUDIP SEN, College of William & Mary, National Institute of Aerospace (NASA) & Jarvis Christian College — The effect of inhomogeneous flow on the stability of Rayleigh-Taylor (RT) mode is investigated in the presence of realistic flow profile which includes both flow shear (first order radial derivative) and flow curvature (second order radial derivative). It is found that contrary to the usual believe the flow curvature has robust effect on the stability of the RT mode - depending on the sign the flow curvature could be stabilizing or destabilizing. The consequence of this novel finding in various interdisciplinary areas will be discussed.

 KP1.00096 High- and low-symmetric coherent structures and dimensional crossover in Richtmyer-Meshkov flows1, AKLANT BHOWMICK, SNEZHANA ABARZHI, Carnegie Mellon University — We study the three-to-two dimensional crossover for the nonlinear structures appearing in the nonlinear regime of Richtmyer Meshkov instability (RMI). This large-scale coherent structure is an array of bubbles and spikes that is periodic in the plane normal to the direction of an initial shock (impulsive acceleration) [1]. The flow is assumed to be anisotropic in the plane with the symmetry group p2mm. For the bubbles, there is a two-parameter family of regular asymptotic solutions. Stability of these solutions is studied. The transitions to the flows with the group p4mm and pm as well as properties of the dimensional crossover are analyzed. We find that 3D bubbles in RMI tend to conserve a near-symmetric-shape, and cannot be transformed into 2D bubbles continuously. We discuss the mechanism of secondary instabilities in anisotropic RM flows and the discontinuity of the dimensional crossover, as well as their dependence of the density ratio.

1Support of the National Science Foundation is warmly appreciated

 KP1.00097 On reliable quantification of Richtmyer-Meshkov flows1, NORA SWISHER, Carnegie Mellon University, MILOS STANIC, Technical University of Munich, ROBERT STELLINGWERF, Stellingwerf Consulting, JASON OAKLEY, RICCARDO BONAZZA, University of Wisconsin-Madison, SNEZHANA ABARZHI, Carnegie Mellon University — We report an integrated study of the dimensional crossover, as well as their dependence of the density ratio. We analyze quantitative and qualitative features of RT dynamics including the vector and scalar flow fields, the bulk and interface velocities, the large-scale interfacial structures and small-scale non-uniformities (reverse jets, hot spots) in the bulk. We argue that a systematic interpretation of RM dynamics from the data and a reliably quantification the RM evolution requires a synergy of the experiments, simulation, and theory.

1Support of the National Science Foundation is warmly appreciated

 KP1.00098 GEOPHYSICAL FLUID DYNAMICS —
the de-agglomeration of particles in the first cycle while working a high-pressure and high-shear condition. Viscosity values of nanofluids are observed to have reduced. This reduction in viscosity at the second pressure cycle could have been caused by the rheological characteristics of nanofluids are measured and are compared with that of the basefluid under increasing and decreasing pressures. Based nanofluids (prepared with alumina nanoparticles) under pressures up to 1000 bar are investigated using a high-pressure viscometer. The drilling fluids under cyclic high-pressure loadings have not been investigated so far. In the present work, rheological characteristics of silicon oil dispersed in a base fluid (or drilling fluid) are commonly used in oil industry to aid the drilling of oil well into the ground. Nanofluids, the colloidal suspensions of particles or wind erosional processes. Here we focus on dissolution processes. We perform laboratory experiments on hard caramel bodies, which dissolve on a short timescale, compared to geological material such as limestone. We highlight the spontaneous appearance of a dissolution pattern with no external flow. When a tilted hard caramel block dissolves, the syrup (denser than water) sinks in the bath and induces a flow, which results in a pattern on the bottom of the block. First parallel stripes appear, which evolve to transversal scallops in about one hour. The whole pattern moves upstream at a slow velocity. The stripes appearance is due to a buoyancy-driven instability. By varying the density and the viscosity of the bath, we show that the initial wavelengths of the pattern are in agreement with those given by the solutal Rayleigh-Bénard number. Later pattern evolution to scallops results from complex interactions between the flow and the topography. Finally we emphasize that similar mechanism of patterns formation can occur in the dissolution of minerals like salt, but also in the shaping of the bottom face of melting icebergs in the cold seas.

KP1.00100 ABSTRACT WITHDRAWN —

KP1.00101 Wind Characteristics of Coastal and Inland Surface Flows1. CHELAKARA SUBRAMANIAN, STEVEN LAZARUS, TETSUYA JIN, Florida Institute of Technology — Lidar measurements of the winds in the surface layer (up to 80 m) inland and near the beach are studied to better characterize the velocity profile and the effect of roughness. Mean and root-mean-squared profiles of horizontal and vertical wind components are analyzed. The effects of variable time (18, 60 and 600 seconds) averaging on the above profiles are discussed. The validity of common surface layer wind profile models to estimate skin friction drag is assessed in light of these measurements. Other turbulence statistics such as auto- and cross- correlations in spatial and temporal domains are also presented.

1The help of FIT DMES field measurement crew is acknowledged.

KP1.00102 Direct Statistical Simulation: Ensemble Averaging and Basis Reduction1. ALTAN ALLAWALA, BRAD MARSTON, Brown University — Low-order statistics of models of geophysical fluids may be directly accessed by solving the equations of motion for the equal-time cumulants themselves. We investigate a variant of the second-order cumulant expansion (CE2) in which zonal averaging is replaced by ensemble averaging. Proper orthogonal decomposition (POD) of the second cumulant is used to reduce the dimensionality of the problem. The approach is tested on a quasi-geostrophic 2-layer baroclinic model of planetary atmospheres by comparison to the traditional approach of accumulating statistics via numerical simulation, and to zonal averaged CE2.

1Supported in part by NSF DMR-1306806 and NSF CCF-1048701

KP1.00103 Large Eddy Simulations of Kelvin Helmholtz instabilities at high Reynolds number stratified flows. DANA BROWN, LOU GOODMAN, MEHDI RAESSI, University of Massachusetts Dartmouth — Simulations of Kelvin Helmholtz Instabilities (KHI) at high Reynolds numbers are performed using the Large Eddy Simulation technique. Reynolds numbers up to 100,000 are achieved using our model. The resulting data set is used to examine the effect of Reynolds number on various statistics, including dissipation flux coefficient, turbulent kinetic energy budget, and Thorpe length scale. It is shown that KHI are qualitatively different at high Re, up to and including the onset of vortex pairing and billow collapse and quantitatively different afterward. The effect of Richardson number is also examined. The results are discussed as they apply to ocean experiments.

KP1.00104 Control of mixing hotspots over the vertical turbulent flux in the Southern Ocean. ALI MASHAYEK, RAFFAELE FERRARI, MIT, JIM LEDWELL, Woods Hole Oceanographic Institution, SOPHIA MERRIFIELD, MIT, LOUIS ST. LAURENT, Woods Hole Oceanographic Institution — Vertical turbulent mixing in the Southern Ocean is believed to play a role in setting the rate of the ocean Meridional Overturning Circulation (MOC), one of the key regulators of the climate system. The extent to which mixing influences the MOC, however, depends on its strength and is still under debate. To address this, a passive tracer was released upstream of the Drake Passage in 2009 as part of the Diapycnal and Isopycnal Mixing Experiment in the Southern Ocean (DIMES). Vertical dispersion of the tracer was measured in subsequent years to estimate vertical mixing. The inferred effective turbulent diffusivity values have proven larger than those obtained from localized measurements of shear made at various locations along the path of the tracer. While the values inferred from tracer imply a key role played by mixing in setting the MOC, those based on localized measurements suggest otherwise. In this work, we use data from DIMES in combination with a high resolution numerical ocean model to investigate whether these discrepancies are the result of different sampling strategies; the microstructure profiles sampled mixing only in a few regions, while the tracer sampled mixing over a much wider area as it spread spatially.

KP1.00105 Geometrodynamical Fluid Theory Applied to Dynamo Flows in Planetary Interiors. KAYLA LEWIS, Monmouth University, DIEGO MIRAMONTES, College of Wooster, DILLON SCOFIELD, University of Oklahoma — Due to their reliance on a Newtonian viscous stress model, the traditional Navier-Stokes equations are of parabolic type; this in turn leads to acausal behavior of solutions to these equations, e.g., a localized disturbance at any point instantaneously affects the solution arbitrarily far away. Geometrodynamical fluid theory (GFT) avoids this problem through a relativistically covariant formulation of the flow equations (Phys. Lett. A 374 3476-82 (2010)). Using GFT, we derive the magnetohydrodynamic equations describing the balance of energy-momentum appropriate for dynamo flows in planetary interiors. These equations include interactions between magnetic and fluid vortex fields. We derive scaling laws from these equations and compare them with scaling laws derived from the traditional approach. Finally, we discuss implications of these scalings for flows in planetary dynamos.

KP1.00106 INDUSTRIAL APPLICATIONS —

KP1.00107 Pressure cycle rheology of nanofluids at ambient temperature, ANOOP KANJIRAKAT, REZA SADR, ROMMEL YRAC, MAHMOOD AMANI, Texas A and M University at Qatar — Colloidal suspensions of particles dispersed in a base fluid (or drilling fluid) are commonly used in oil industry to aid the drilling of oil well into the ground. Nanofluids, the colloidal suspensions of nano-sized particles dispersed in a basefluid, have also shown potentials as cooling and abrasive fluids. Utilizing them along with drilling fluids under cyclic high-pressure loadings have not been investigated so far. In the present work, rheological characteristics of silicon oil based nanofluids (prepared with alumina nanoparticles) under pressures up to 1000 bar are investigated using a high-pressure viscometer. The rheological characteristics of nanofluids are measured and are compared with that of the basefluid under increasing and decreasing pressures. Relative viscosity variations of nanofluids were observed to have influenced by the shear rate. In addition, under cyclic high-pressure loading viscosity values of nanofluids are observed to have reduced. This reduction in viscosity at the second pressure cycle could have been caused by the de-agglomeration of particles in the first cycle while working a high-pressure and high-shear condition.
KP1.00108 JETS —

KP1.00109 ABSTRACT WITHDRAWN —

KP1.00110 Computational analysis for dry-ice sublimation assisted CO$_2$ jet impingement flow, SONGMI KWAK, JAESEON LEE, UNIST — The flow and heat transfer characteristics of the novel gas-solid two-phase jet impingement are investigated computationally. When the high pressure carbon dioxide (CO$_2$) flow passes through a nozzle or orifice, it experiences the sudden expansion and the rapid temperature drop occurred by Joule-Thomson effect. This temperature drop causes the lower bulk jet fluid temperature than the CO$_2$ sublimation line, so dry-ice becomes formed. By using CO$_2$ gas-solid mixture as a working fluid of jet impingement, it is expected the heat transfer enhancement can be achieved due to the low bulk temperature and the additional phase change latent heat. In this study, 2D CFD model is created to predict the cooling effect of gas-solid CO$_2$ jet. The gas-solid CO$_2$ flow is considered by Euler-Lagrangian approach of mixed phase and the additional heat transfer module is embedded to account for the sublimation phenomena of the solid state CO$_2$. The jet flow and heat transfer performance of gas-solid CO$_2$ jet is investigated by the variance of flow parameter like Reynolds number, solid phase concentration and jet geometries.

KP1.00111 MAGNETOHYDRODYNAMICS —

KP1.00112 Helical mode interactions and spectral transfer processes in magneto-hydrodynamic turbulence, MORITZ LINKMANN, ARJUN BERERA, MAIRI MCKAY, JULIA JÄGER, Univ of Edinburgh — Spectral transfer processes in magnetohydrodynamic (MHD) turbulence are investigated analytically by decomposition of the velocity and magnetic fields in Fourier space into helical modes. Steady solutions of the dynamical system which governs the evolution of the helical modes are determined, and a stability analysis of these solutions is carried out. The interpretation of the analysis is that unstable solutions lead to energy transfer between the interacting modes while stable solutions do not. From this, a dependence of possible interscale energy and helicity transfers on the helicities of the interacting modes is derived. The direction of energy transfer not only depends on magnetic and kinetic helicities but also on the ratio of magnetic to kinetic energy and on the cross-helicity. As expected from the inverse cascade of magnetic helicity in 3D MHD turbulence, mode interactions with like helicities lead to transfer of energy and magnetic helicity to smaller wavenumbers. However, some interactions of modes with unlike helicities also contribute to an inverse energy transfer. As such, an inverse energy cascade for nonhelical magnetic fields is shown to be possible.

KP1.00113 Finite dissipation and nonuniversality in magnetohydrodynamic turbulence, MORITZ LINKMANN, ARJUN BERERA, MAIRI MCKAY, ERIN GOLDSTRAW, Univ of Edinburgh, W. DAVID MCCOMB, Retired — A model equation for the Reynolds number dependence of the dimensionless dissipation rate $C_\varepsilon$ in homogeneous magnetohydrodynamic turbulence in the absence of a mean magnetic field is derived from the real-space energy balance equation, leading to $C_\varepsilon = C_{\varepsilon,\infty} + C/R_- + O(1/R_-^2)$, where $R_-$ is a generalized Reynolds number. The constant $C_{\varepsilon,\infty}$ is here defined in terms of the Elsässer fields and is shown to describe the total energy transfer flux. This flux depends on magnetic and cross helicities, because these affect the nonlinear transfer of energy, suggesting that the value of $C_{\varepsilon,\infty}$ is not universal. Direct numerical simulations for freely decaying and stationary MHD turbulence were conducted on up to 2048$^3$ grid points, showing good agreement between data and the model for both cases, different initial values of cross and magnetic helicities and different forcing schemes. The ideas introduced here can be used to derive similar model equations for other turbulent systems.

KP1.00114 MICROSCALE FLOWS —

KP1.00115 Three-dimensional simulation of droplet migration in a Hele-Shaw microchannel, YUE LING, JOSE-MARIA FULLANA, STEPHANE POPINET, CHRISTOPHE JOSSE RAND, Institut d’Alembert, UPMC-Paris 6 — Droplet-based microfluidics is a promising tool for performing biomechanical and chemical assays. Three-dimensional simulations are performed in this work to investigate the migration of a droplet in a confined microchannel (a Hele-Shaw cell). As the droplet moves in the channel, a thin film is formed between the droplet and the wall. The thickness of the film can be two orders of magnitude smaller than the channel height. Furthermore, the time step which is mainly controlled by the surface tension effect becomes very small for low Capillary number. Therefore, numerical simulation of droplet migration in microchannel is challenging. The present simulations are conducted with a two-phase flow solver (GERRIS) on an adaptive mesh. The interface between the two phases is captured by the Volume-of-Fluid method. The droplet dynamics are very different as the aspect ratio (the ratio between the droplet diameter and the channel height) varies from smaller to larger than unity. For droplets of large aspect ratio, the droplet velocity is mainly dictated by the dynamics of the thin film. The simulations also show that the flow around the droplet is three dimensional and has a significant impact on the droplet shape.

1ANR-13-BS09-0011

KP1.00116 Hemorheology in PDMS Micro channel with varied surface roughness, BHARATH BABU NUNNA, SHIQIANG ZHUANG, EON SOO LEE1. New Jersey Inst of Tech — Hemorheology in micro channel is studied in order to enhance the diagnostic phenomenon of micro assay. Blood consists of formed elements (RBC, WBC & Platelets) and Plasma (Water, Plasma proteins & other solutes). Blood due to existence of RBC will behave as a Non-Newtonian fluid. The flow of blood varies on surface roughness of the passage. In this presentation the blood flow characteristics is examined in the micro channel with varied surface roughness on the walls of PDMS micro channels. The micro channel considered for this experiment is fabricated with a cross section close to a rectangular shape with width of 200 mm-1000mm and depth of 100 mm and optically transparent. The analysis of Surface roughness impact on blood flow will help to define and design the micro channel with specified surface treatment on the walls of the channel.

1Prof. Eon Soo Lee is the “Principal Investigator” of our lab.
KP1.00117 Numerical analysis of mixing by sharp-edge-based acoustofluidic micromixer, NITESH NAMA, PO-HSUN HUANG, TONY JUN HUANG, FRANCESCO COSTANZO, Pennsylvania State Univ — Recently, acoustically oscillated sharp-edges have been employed to realize rapid and homogeneous mixing at microscales (Huang, Lab on a Chip, 13, 2013). Here, we present a numerical model, qualitatively validated by experimental results, to analyze the acoustic mixing inside a sharp-edge-based micromixer. We extend our previous numerical model (Nama, Lab on a Chip, 14, 2014) to combine the Generalized Lagrangian Mean (GLM) theory with the convection-diffusion equation, while also allowing for the presence of a background flow as observed in a typical sharp-edge-based micromixer. We employ a perturbation approach to divide the flow variables into zeroth-, first- and second-order fields which are successively solved to obtain the Lagrangian mean velocity. The Langrangian mean velocity and the background flow velocity are further employed with the convection-diffusion equation to obtain the concentration profile. We characterize the effects of various operational and geometrical parameters to suggest potential design changes for improving the mixing performance of the sharp-edge-based micromixer. Lastly, we investigate the possibility of generation of a spatio-temporally controllable concentration gradient by placing sharp-edge structures inside the microchannel.

KP1.00118 Modeling wrinkled-assisted assembly of ordered nanoparticles and nanorods on a wavy substrate. CAMILA LUPPI SATO, PETER YEH, ALEXANDER ALEXEEV, Woodruff School of Mechanical Engineering, Georgia Institute of Technology, MARTIN MAYER, PATRICK PROBST, ANDREAS FERY, Physical Chemistry II, University of Bayreuth, Germany — Wrinkle-assisted assembly is a technique that allows for fabrication of ordered structures of nanoparticles and nanorods on hydrophilic substrates. As an intermediate step in this process, nanoparticles are deposited within microscopically wrinkled surfaces, where they organize into patterned structures upon solvent evaporation. However, the dependence of the resulting pattern on nanoparticle concentration, particle size and shape, and substrate geometry is not well understood. We develop a model of the ordering process using dissipative particle dynamics (DPD) to predict the resulting nanostructures. We approximate the wavy sheet as a sinusoidal surface. One layer of DPD liquid containing nanoparticles fills the surface, while another layer of DPD fluid acts as the gaseous phase. We model the evaporative process by gradually replacing DPD liquid particles with DPD gaseous particles. The results of our work are useful in designing surface patterns that exhibit strong plasmonic coupling.

1Financial support from NSF CAREER Award DMR-1255288 is gratefully acknowledged.

KP1.00119 MULTIPHASE FLOWS —

KP1.00120 Spectral Analysis of Cluster Induced Turbulence, RAVI PATEL, PETER IRELAND, Cornell University, JESSE CAPECELATRO, University of Illinois at Urbana-Champaign, RODNEY FOX, Iowa State University, OLIVIER DESJARDINS, Cornell University — Particle laden turbulent flows are an important feature of many industrial processes such as fluidized bed reactors. The study of cluster-induced turbulence (CIT), wherein particles falling under gravity generate turbulence in the carrier gas via fluctuations in particle concentration, may lead to better models for these processes. We present a spectral analysis of a database of statistically stationary CIT simulations. These simulations were previously performed using a two way coupled Eulerian-Lagrangian approach for various mass loadings and particle-scale Reynolds numbers. The Lagrangian particle data is carefully filtered to obtain Eulerian fields for particle phase volume fraction, velocity, and granular temperature. We perform a spectral decomposition of the particle and fluid turbulent kinetic energy budget. We investigate the contributions to the particle and fluid turbulent kinetic energy by pressure strain, viscous dissipation, drag exchange, viscous exchange, and pressure exchange over the range of wavenumbers. Results from this study may help develop closure models for large eddy simulation of particle laden turbulent flows.

KP1.00121 NANO FLOWS —

KP1.00122 Electrospinning of Biodegradable and Biocompatible Nanofiber Patches from Solutions of "Green" Materials for Plant Protection against Fungi Attack, SOUMYADIP SETT, MINWOOK LEE, ALEXANDER YARIN, Univ of Illinois - Chicago, S.M. ALAVI MOGHADAM, MATTHIAS MEINKE, WOLFGANG SCHROEDER, Institute of Aerodynamics, RWTH Aachen University — Biodegradable and biocompatible soy protein/petroleum-derived polymer monolithic fibers containing adhesives were electrospun on commercial rayon pads. The polymers used, PVA and PCL, are widely used in the biomedical industry, including such applications as drug delivery and scaffold manufacturing. Soy protein is an abundant waste of SoyDiesel production, and is widely used as a nutrient. The soy content in our fibers was as high as 40 %w/w. Four different adhesives, including ordinary wood glue, repositionable glue and FDA-approved pressure-sensitive glue were used for electrospinning and electrospaying. The normal and shear adhesive strengths of the patches developed in this work were measured and compared. The adhesive strength was sufficient enough to withstand normal atmospheric conditions. These biodegradable and biocompatible nano-textured patches are ready to be used on prune locations without being carried away by wind and will protect plants against fungi attack at these locations, preventing diseases like Vine Decline.

KP1.00123 Non-equilibrium molecular dynamics simulation of the unstirred flow in the osmotically driven flow, KEITO KONNO, TOMOAKI ITANO, MASAKO SEKI, Kansai Univ — We studied the solvent flows driven by the osmotic pressure difference across the semi-permeable membrane. The flow penetrating from the low concentration side transports away solutes adjacent to the membrane, so that the concentration is reduced significantly only at the vicinity of the membrane. It is expected that the relatively low solute concentration develops into a thin boundary layer in the vicinity of the membrane in the case of absence of external stirring process, which is termed as un-stirred layer (USL). To investigate concentration distribution in USL, we carried out non-equilibrium molecular dynamics simulations. The flows driven by osmotic pressure are idealized as 2 dimensional hard disk model, which is composed of solvent and solute molecules. The membrane is modeled as a medium composed of stationary parallel rods distributed by a spatial interval, which is less than the diameter of the solute molecules. The following results were obtained from the numerical simulation. First, the thickness of USL, which was estimated from the obtained concentration distribution, is on the order of a length determined by mean free path. Second, USL was semicircle the center of which is on the end of pore of membrane.

KP1.00124 NONLINEAR DYNAMICS —

KP1.00125 ABSTRACT WITHDRAWN —
The flow of slurry in the inclined closed channel was examined. The viscoelastic fluid is modeled as a derivative of a typical Oldroyd-B relation of stress and velocity gradient. First, gravity is considered as the driving force for the fluid flow to simulate the existing sewage system. The complete flow field is evaluated for this case. Next, a pressure gradient is introduced to observe its effects on the flow. Velocity profile as well as stress distributions are given for different scenarios of the nonlinear fluid flowing in a closed channel with and without pressure gradient.

Magnetorheological rotational flow between coaxial cylinders was considered. The effects of a magnetic field and fluid nonlinearity are investigated for the rotational flow of a nonlinear viscoelastic fluid following the Carreau model while viscous dissipation is taken into account. The governing motion and energy balance equations are coupled, adding complexity to the already highly correlated set of differential equations. The numerical solution is obtained for the narrow gap limit and steady state base flow. Magnetic field effect on local entropy generation due to steady two-dimensional laminar forced convection flow was investigated. This study was focused on the entropy generation characteristics and its dependency on various dimensionless parameters. The effects of the Hartmann number, the Brinkman number, and the Deborah number on the stability of the flow were investigated. Introduction of the magnetic field induces resistive force acting in the opposite direction of the flow, thus causing its deceleration. Moreover, the study shows that the presence of magnetic field tends to slow down the fluid motion. It, however, increases the fluid temperature. Moreover, the total entropy generation number decreases as the Hartmann number and fluid elasticity increase and increases with increasing Brinkman number.

Poromechanics modeling of fault stability under the influence of fluid pressure changes was considered. We demonstrate the ability of the two-way coupled poromechanic model to reproduce rock deformation behavior measured in triaxial laboratory tests under the influence of pore pressure. We then study the fault stability in the case of a preexisting impermeable fault, across which there exist a pressure discontinuity due to fluid injection on one side. Numerical results are discussed with a focus on the fault stability criterion and the slip behavior.

Gravity-Driven Particle-Laden Flow on an Incline was studied. We present experimental results of the height profile of particle-laden viscous thin films with finite volume on an incline. For high angles of inclination and high concentrations of mixtures, negatively buoyant particles undergo resuspension then accumulate at the front of the suspending fluid; this leads to the development of a particle-rich ‘ridge’. Theoretically, the ridge corresponds to the shocks which take on two characteristic shapes: singular and double shocks. We observe the presence of both formations experimentally by varying the volume of the slurry and compare our results to the theoretical model. Our research also investigates the dependence of the fingering instability as the inclination angle or particle to liquid concentration is changed. The slurries have similar dynamics to those used in coating flow techniques and other industrial applications.
KP1.00139 Molecular dynamics analysis of reflected gas molecules on self-assembled monolayers1, HIDEKI TAKEUCHI, National Institute of Technology, Kochi College — In order to investigate the gas flow of high Knudsen number, it is necessary to specify the boundary condition for the reflected gas molecules at a solid surface. In most cases of the analysis, the diffuse reflection is generally assumed, but there are many cases for which this reflection cannot be applied. The characteristics of the reflected gas molecules depend on the state of the solid surface as well as the gas-surface interaction. The present author analyzed the scattering properties of monoatomic and diatomic gases on various solid surfaces based on the molecular dynamics (MD) method and proposed the boundary condition of reflected gas molecule (Phys. Fluids 18, 046103, 2006). Recently, self-assembled monolayers (SAMs) for the functionalization of the solid surface have been used in the development of micro/nano devices such as microarray and nanosensor. Therefore, it is interesting to study the scattering behavior of the reflected gas molecules on the SAM surface and make the scattering model of gases for the boundary condition. In this study, the angular distribution and the trapping probability for gas molecule on the SAM surface are observed by using MD simulation. The scattering probability at different incident energies is also discussed.

1JSPS KAKENHI Grant Number 26870813

KP1.00140 REACTING FLOWS —

KP1.00141 Simulations of Small-Scale Liquid Film Combustors1, PAVEL POPOV, WILLIAM SIREGNANO, University of California at Irvine, Mechanical and Aerospace Engineering — Recent technological advances have generated need for small-scale combustor designs. The reduction of scale, however, leads to a higher area to volume ratio and thus greater relative heat loss. Liquid film combustors are one proposed design which aims to overcome this obstacle. In them, the fuel is injected as a liquid film on the combustor wall, and heat transfer is reduced due to evaporative cooling of the liquid film leading to reduced temperature gradients at the combustor walls. In this work, we present simulation results for a cylindrical small scale liquid film combustor, in which the reactants are liquid heptane and gaseous air. A computational procedure has been developed to simulate this two-phase combustion problem, using detailed chemical mechanisms. A cubic equation of state is applied for the simulation of the gaseous phase at high pressures. The present study examines the structure of the triple flame inside this combustor design, which has been analyzed in previous experimental work. Comparison between simulation and experimental work is made, with particular emphasis on the influence of the chemical mechanism, high-pressure equation of state, and the effect of swirl amplitudes in the liquid and gas phases on the structure of the flame.

1Supported by AFOSR grant FA9550-12-1-0156, AFOSR scientific manager: Dr. Mitat Birkan


1The author acknowledge financial support from SABIC, #SB101018, through the Dean of Scientific Research at KFUPM

KP1.00143 SEPARATED FLOWS —

KP1.00144 A Study of Pulsed Blowing Effect on Flow Separation over Flap1, YANKUI WANG2, PING ZHOU3, QIAN LI4, BeiHang University — With the development of the modern aircraft, such as tailless flying configuration, traditional flaps are also the main control surfaces for flight controlling. However, the efficiency of the flap is not only descent quickly due to flow separation over itself under higher deflection angle of flap, but also is evidently influenced by the flow coming down from the upstream wing. A novel flow control technique to improve the flow separation over the flap by pulsed blowing is investigated in this paper by wind tunnel test under Reynolds number of 0.6*10E6 - 2.4*10E6. To begin with, the control performance for flow separation over the flap is very sensitive to the blowing position and direction and the flow separation can be recovered by the pulsed blowing evidently. Secondly, the pulsed blowing efficiency is 30% higher than that of continuous blowing with the same consumption. In addition, the pulsed blowing efficiency increases quickly with the increasing of pulsed blowing frequency and keep constant gradually when the pulsed blowing Strou number is bigger than 0.6.

1National Natural Science Foundation of China(11272035)
2Prof Yankui Wang received a PhD degree in Fluid Mechanics from BeiHang University and served as a full Professor from 2005. His current research interest is fluid mechanics and complex flow.
3Ping Zhou is currently a Ph.D. degree of fluid Mechanics in BeiHang University.
4Qian Li is currently a Ph.D. candidate of fluid mechanics in BeiHang University.

KP1.00145 SUSPENSIONS —

KP1.00146 Two-scale evolution during shear reversal in dense suspensions, CHRISTOPHER NESS, JIN SUN, University of Edinburgh — We use shear reversal simulations to explore the rheology of dense, non-Brownian suspensions, resolving lubrication forces between neighbouring particles and modelling particle contacts as linear springs. The transient stress response to an abrupt reversal of the direction of shear shows rate-independent, nonmonotonic behaviour, capturing the salient features of the corresponding classical experiments. Based on analyses of the hydrodynamic and particle contacts, we demonstrate distinct responses at small and large strains, associated with contact breakage and structural re-orientation, respectively, emphasising the importance of particle contacts. Consequently, the hydrodynamic and contact stresses evolve over disparate strain scales and with opposite trends, resulting in nonmonotonic behaviour when combined. We further elucidate the roles of particle roughness and repulsion in determining the microstructure and hence the stress response at each scale.


**KP1.00147 TURBULENCE** –

**KP1.00148 Analytical damped-oscillator models for unsteady atmospheric boundary layers**¹, MOSTAFA MOMEN, ELIE BOU-ZEID, Princeton University — Geophysical flows are dynamical systems that are evolving nonlinearly with time. Non-stationary shear and buoyancy forces are the main sources that drive the unsteadiness of such flows. However, due to their inherent complexity, most previous studies focused on steady-state conditions. In these boundary layers, the pressure gradient, buoyancy, Coriolis, and friction forces interact. The mean PDEs governing the unsteady version of the problem, which emerges when these forces are not in equilibrium, are solvable only for a limited set of forcing variability modes, and the resulting solutions are intricate and difficult to interpret. Here we derive a simpler physical model that reduces the governing RANS equations into a first-order ODE with non-constant coefficients. The origin of the non-stationarity of turbulence can be buoyant stabilization/destabilization and/or unsteady pressure gradient. The reduced model is straightforward and solvable for arbitrary turbulent viscosity variability, and it captures LES results for linearly variable buoyancy and pressure gradient pretty well. The suggested model is thus general and will be useful for elucidating some features of the diurnal cycle, for short-term wind forecast, and in meteorological applications.

¹NSF-PDM under AGS-10266362. Simulations performed at NCAR, and Della server at Princeton University. Cooperative Institute for Climate Science, NOAA-Princeton University under NA08OAR4320752

**KP1.00149 Wake characteristics of a porous square cylinder formed by a multi-scale array of obstacles** , DANIEL J. WISE, Univ of Sheffield, PAULINE AVOUSTIN, MARTIN CASSADOUR, INP ENSEEIHT, WERNHER BREVIS, Univ of Sheffield — The characteristics of the flow developed behind arrays of square cylinders are investigated through Particle Image Velocimetry (PIV) and Acoustic Doppler Velocimetry (ADV) measurements in an open-channel water flume. Four arrangements of cylinders are examined: three are multi-scale arrays of cylinders based on the Sierpinski carpet fractal, and the fourth is a regular aligned array of single length-scale cylinders. The porosity, frontal area and external length scale is the same for each cylinder array, while the internal geometry is changed. The relative effect on the dynamics of the wake of the fractal parameters defining the array geometry, such as lacunarity and succurrallity is quantified. Special focus is given to the effect of parameters on the extension and properties of the separated shear layers and on the low-velocity zone developed downstream the cylinders.

**KP1.00150 Correlation of Near-Wall Turbulence Structures with Heat Transfer in Ribbed-Pipe Flow**¹, HYUNGSU AHN, CHANGWOO KANG, KYUNG-SOO YANG, DOOHYUN PARK, Inha University — Ribbed-pipe flow is one of the most commonly used flow configurations to enhance heat transfer, albeit, at the expense of increased pressure drop. The ribs mounted on the pipe wall differently alter the flow depending on the pitch, the distance between two neighboring ribs. When the pitch is short, fluid is trapped inside grooves, resulting in a low heat-transfer rate. When the pitch is long enough, however, the ribs disturb the flow by shedding vortices, resulting in heat-transfer enhancement. We aim at elucidating the correlation of near-wall turbulence induced by the shed vortices with the increased heat-transfer rate on the ribbed-pipe wall. Our analysis is based on our LES data base obtained for Re=24,000, Pr=0.71, Fr(pitch ratio)=2, 4, 6, 8, 10, 18, Br(blockage ratio)=0.0625. Here, the bulk velocity and the pipe diameter are used as the velocity and length scales, respectively. Our goal is to focus on the near-wall distributions of the higher-order turbulence statistics including but not limited to rms of temperature fluctuation, cross-correlations, rms of vorticity, and turbulent heat fluxes. Octants and JPDF are also presented in order to clarify the prevailing heat-transfer mechanism in the immediate vicinity of the ribbed-pipe wall. This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP) (No. 2015R1A2A2A01002981).

¹This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP) (No. 2015R1A2A2A01002981).

**KP1.00151 A numerical and experimental study of anisotropic turbulence in a converging annular duct** , JUNWOO LIM, JAMES KOPRIVA, GREGORY LASKOWSKI, GE Aviation, SARA ROSTAMI, JONATHON SLEPSKI, GE Global Research Center, JOSHUA SZZUDLAK, ARMAN MIRHASHEMI, SCOTT M M RIS, Department of aerospace and mechanical engineering, University of Notre Dame — Large Eddy Simulations of turbulent flow in a converging annular duct has been conducted in support of turbomachinry applications. The experimental rig at University of Notre Dame utilizes wall slot flows to generate temperature profiles representative of a combustor exit upstream of the contraction inlet. A passive turbulence grid is placed just downstream of the profile generator and upstream of the contraction to generate high levels of anisotropic turbulence which is characterized by measurements. The LES runs include both the profile generator and turbulence screen in the analysis to better understand the impact of flow acceleration on vortex stretching for this high level of anisotropic turbulence. An evaluation of the evolution of the anisotropy tensor based on resolved scale will be discussed and comparisons with data will be provided.

**KP1.00152 Rank-Ordered Multifractal Analysis of Probability Distributions in Fluid Turbulence** , CHENG-CHIN WU, UCLA, TIEN CHANG, MIT — Rank-Ordered Multifractal Analysis (ROMA) was introduced by Chang and Wu (2008) to describe the multifractal characteristic of intermittent events. The procedure provides a natural connection between the rank-ordered spectrum and the idea of one-parameter scaling for monofractals. This technique has successfully been applied to MHD turbulence simulations and turbulence data observed in various space plasmas. In this paper, the technique is applied to the probability distributions in the inertial range of the turbulent fluid flow, as given in the vast Johns Hopkins University (JHU) turbulence database. In addition, a refined method of finding the continuous ROMA spectrum and the scaled probability distribution function (PDF) simultaneously is introduced.

**KP1.00153 Topographic effect on the inclination angle of ramp like structures in rough wall, turbulent channel flow**¹, ANKIT AWASTHI, WILLIAM ANDERSON, UT Dallas — We have studied variation in structural inclination angle of coherent structures responding to a topography with abrupt spanwise heterogeneity. Recent results have shown that such a topography induces a turbulent secondary flow due to spanwise-wall normal heterogeneity of the Reynolds stresses (Anderson et al., 2015: J. Fluid Mech.). The presence of these spanwise alternating Fluid Mech.) are primarily due to the spanwise heterogeneity of the complex roughness under consideration. Results from the present research have been used to explore structural attributes of the hairpin packet paradigm in the presence of a turbulent secondary flow. Vortex visualization in the streamwise-wall normal plane above the crest (high drag) and trough (low drag) demonstrate variation in the inclination angle of coherent structures. The inclination angle of structures above the crest was approximately 45 degrees, much larger than the “canonical” value of 15 degrees. Thus, we present evidence that the hairpin packet concept is preserved - but modified - when a turbulent secondary flow is present.

¹This work was supported by the Air Force Office of Sci. Research, Young Inv. Program (PM: Dr. R. Ponnoppan and Ms. E. Montgomery) under Grant # FA9550-14-1-0394. Computational resources were provided by the Texas Adv. Comp. Center at Univ. of Texas.
VORTEX DYNAMICS AND VORTEX FLOWS —

Unsteady wake of a rotating tire

JEAN-ELOI LOMBARD, DAVE MOXLEY, HUI XU, SPENCER SHERWIN, Imperial College, SHERWIN LAB TEAM — For open wheel race-cars, such as IndyCar and Formula One, the wheels are responsible for 40% of the total drag. For road cars drag associated to the wheels and under-carriage can represent 60% of total drag at highway cruise speeds. Experimental observations have reported two or three pairs of counter rotating vortices, the relative importance of which still remains an open question, that interact to form a complex wake. Traditional RANS based methods are typically not well equipped to deal with such highly unsteady flows which motivates research into more physical, unsteady models. Leveraging a high-fidelity spectral/hp element based method a Large Eddy Simulation is performed to give further insight into unsteady characteristics of the wake. In particular the unsteady nature of both the jetting and top vortex pair is reported as well as the time and length scales associated with the vortex core trajectories. Correlation with experimentally obtained particle image velocimetry is presented.

Simulating external flow using vortex method in two- and three-dimensions

HENRIK JULIUS SPIETZ, MADS MHLOM HEJLESEN, Technical University of Denmark, JENS HONORE WALTHER, Technical University of Denmark and Computational Science and Engineering Laboratory, ETH Zurich, ALLAN LARSEN, COWI Consulting Engineers, COWI CONSULTING ENGINEERS COLLABORATION — Vortex methods are numerical methods for simulating fluid flow. They use a simple formulation where only the trajectories of discrete vortex particles are simulated. In our method we combine a high order particle-mesh based vortex method with an iterative penalization method to simulate external flows around arbitrary geometries such as bridge decks. The method only uses a discretized geometry as input and can easily simulate an arbitrary motion of the geometry. As vorticity is a bounded quantity and the velocity field can easily be calculated for a mixture of free-space- and periodic boundary conditions, the method allows for a minimized domain and hence minimal computational resources. However in an external flow problem, vorticity is produced in the boundary layers and transported downstream microscopically the computed vorticity field has zero growth in time to compensate for the entire vorticity field. We present a method for truncating this domain by supplementing the free-space- and periodic conditions with an outflow condition. The method is conveniently applied within the field of bridge aerodynamics as it can be used for the calculation of the aerodynamic net forces, which depend highly on the geometry and the wake forming behind it. This is demonstrated in 2D and 3D simulations.

Optimal geometry of an axisymmetric wave energy converter

EMMA EDWARDS, DICK K. P. YUE, Massachusetts Institute of Technology, VORTICAL FLOW RESEARCH LABORATORY TEAM — There have been a number of theoretical, experimental and pilot-scale studies on wave energy converters with varying shapes and designs, but due to the complex nature of wave-body hydrodynamics, as yet there is not one single three-dimensional shape that is agreed-upon to be optimal for wave power extraction. Our objective is to determine the optimal geometry to maximize power uptake over a spectrum of incident waves. As an initial investigation, we consider an axisymmetric floating wave power extraction device operating in heave. We assume linear wave conditions. The body geometry is described by smooth polynomial basis functions and is allowed to be completely general, subject to simple constraints. We consider a linear power uptake with a fixed damping coefficient (which could be optimized). For each frequency in the spectrum, hydrodynamic coefficients are calculated using a linear frequency-domain panel method. Then, for a specific incident wave spectrum, maximal extractable power is integrated over the entire spectrum. We will discuss the optimal geometry and associated maximum power for different geometrical constraints and wave conditions.

Langmuir Mixing Effects on Global Climate: WAVEWATCH III in CESM

QING LI, Dept. of Earth, Environmental and Planetary Sciences, Brown University, Providence, Rhode Island, USA, ADREAN WEBB, Dept. of Ocean Technology, Policy, and Environment, The University of Tokyo, Kashiwa, Chiba, Japan, BAYLOR FOX-KEMPER, Dept. of Earth, Environmental and Planetary Sciences, Brown University, Providence, Rhode Island, USA, ANTHONY CRAIG, GOKHAN DANABASOGLU, WILLIAM LARGE, MARIANA VERTENSTEIN, National Center for Atmospheric Research, Boulder, Colorado, USA — Large Eddy Simulations (LES) have shown the effects of ocean surface gravity waves in enhancing the ocean boundary layer mixing through Langmuir turbulence. Neglecting this Langmuir mixing process may contribute to the common shallow bias in mixed layer depth in regions of the Southern Ocean and the Northern Atlantic in most state-of-the-art climate models. A third generation wave model, WAVEWATCH III, has been incorporated as a component of the Community Earth System Model, version 1.2 (CESM1.2). In particular, the wave model is now coupled with the ocean model through a modified version of the K-Profile Parameterization (KPP) to approximate the influence of Langmuir mixing. Unlike past studies, the wind-wave misalignment and the effects of Stokes drift penetration depth are considered through empirical scalings based on the rate of mixing in LES. Wave-Ocean only experiments show substantial improvements in the shallow biases of mixed layer depth in the Southern Ocean. Ventilation is enhanced and low concentration biases of pCFC-11 are reduced in the Southern Hemisphere. A majority of the improvements persist in the presence of other climate feedbacks in the fully coupled experiments.

Biological Derived Nanomotors in a “Domino Fashion”

W. H. MAKSOED, Prodi of Physics UI, Depok 16424- Indonesia — For disproportionation of H2O2, we also considers an electrokinetic mechanism they appear. So far, the more efficient micro/nanoscale motors are derived from biological systems [2003]. Besides, a control experimenting using 3 stripped Au/Pt/Au rods with catalyzed the composition of H2O2, at a similar rate-Walter F Paxton: “Catalytic Nanomotors,” JACS, 2004. We also intended to accomplish the HCCI quotes from Marcin Frackowiak, dissertation, 2009, just in several characters seems as twin of IGNITION through IceCube document project held since Oct 11, 2001 ever concludes as “saw none” so they can be follows the ITER/IFMIF. Refers to S29286 file in UI retrieved: ‘magnetic quantum-dot cellular automata which is nonvolatile & lower power consist of nanomagnets. Since they are magnetically coupled, logic can be performed by switching, on the other hand in a DOMINO fashion...’ [A. Klenm: “Fabrication of Magnetic Tunnel Junction-based Spintronic Devices...” convocation, Aug 11-14, 2010.

GENERAL FLUID DYNAMICS —

Acknowledgments devotes to BB Mandelbrot: “Fractal Geometry: What is it & What Does it do?”

Intends normative description tributes to Richard COURANT to HE. Mr. Prof. Dr.deSoz. GUMILAR RUSLIWA SOMANTRI
KP1.00162 Separation Analysis in a High-Speed Rotating Cylinder for a Binary Gas Mixture. SAHADHEV PRADHAN, VISWANATHAN KUMARAN, Department of Chemical Engineering, Indian Institute of Science, Bangalore-560012, India — The solutions of the species balance equations linked with the generalized Onsager model for the secondary gas flow in a high-speed rotating cylinder are compared with the direct simulation Monte Carlo (DSMC) simulations for a binary gas mixture. The concentration fields are obtained through three different types of driving mechanism. These are: (a) wall thermal forcing, (b) inflow/outflow of gas along the axis, and (c) momentum source/sink inside the flow domain, for the stratification parameter ($A$) in the range (0.707-3.535), and Reynolds number ($Re$) in the range ($10^2$ - $10^4$) with aspect ratio (length / diameter) = 2, 4, 8. Two different types of cases have been considered, (a) no mass difference ($\varepsilon = (2 \cdot m_1 \cdot m_2) = 0$), and (b) with mass difference ($\varepsilon = 0.2$ and 0.5) while calculating the secondary flow field in the analytical solution. Here, the stratification parameter $A = \sqrt{(m_1 \cdot R_2^2)/(2 \cdot k_B \cdot T)}$, and the Reynolds number $Re = \rho \cdot u_\infty \cdot R_2^2/\mu$, where $m$ is the molecular mass, $\Omega$ and $R$ are the angular velocity and radius of the cylinder, $\rho_\infty$ is the wall density, $\mu$ is the gas viscosity and $T$ is the gas temperature. The comparison between numerical and analytical solution reveals that the boundary conditions in the numerical simulations and analytical model have to be matched with care. The commonly used “diffuse reflection” boundary conditions at the solid walls in DSMC simulations result in a non-zero slip velocity as well as a “temperature slip” (gas temperature at the wall is different from wall temperature).

KP1.00163 Splash Dynamics of Watercolors on Dry, Wet, and Cooled Surfaces. DAVID BARON, ASHWIN VAIYDA, HAIYAN SU, Montclair State University — In his classic study in 1908, A.M. Worthington gave a thorough account of splashes and their formation through visualization experiments. In more recent times, there has been renewed interest in this subject, and much of the underlying physics behind Worthington’s experiments has now been clarified. One specific set of such recent studies, which motivates this paper, concerns the fluid dynamics behind Jackson Pollock’s drip paintings. The physical processes and the mathematical structures hidden in his works have received serious attention and made the scientific pursuit of art a compelling area of exploration. Our work explores the interaction of watercolors with watercolor paper. Specifically, we conduct experiments to analyze the settling patterns of droplets of watercolor paint on wet and frozen paper. Variations in paint viscosity, paper temperature, and the height of a released droplet are examined from time of impact, through its transient stages, until its final, dry state. Observable phenomena such as paint splashing, spreading, fingering, branching, rheological deposition, and fractal patterns are studied in detail and classified in terms of the control parameters.

KP1.00164 Premixed Combustion Model for Boron Clouds. MENGZE WANG, WANG HAN, ZHENG CHEN, Peking University — Boron particle is an ideal additive in solid propellants and fuels due to its very high volumetric heat release. In this study, a premixed combustion model for boron clouds is developed based on a previous combustion model for single boron particle. The flame structure is assumed to be composed of three zones: the preheat zone, the ignition zone, and the reaction zone, and analytical solutions are derived from the governing equations. Consequently the influence of the boron clouds’ physical properties on the flame propagation process is investigated.

KP1.00165 Stability of algebraically unstable dispersive flows. KRISTINA KING, PAULA ZARETZKY, STEVEN WEINSTEIN, MICHAEL CRÖMER, NATHANIEL BARLOW, Rochester Institute of Technology — A widely unexplored type of hydrodynamic instability is examined - large-time algebraic growth. Such growth occurs on the threshold of (exponentially) neutral stability. A methodology is provided for predicting the algebraic growth rate of an initial disturbance, when applied to a class of partial differential equations describing wave propagation in dispersive media. There are several morphological differences between algebraically growing disturbances and the exponentially growing wave packets inherent to classical linear stability analysis, and these are elucidated in this study.

KP1.00166 Janus Gel Fabrication Using Liquid Drop Coalescence and Limited Mixing in the Hele-Shaw Geometry. BRITTANY GONZALEZ, Georgia Institute of Technology, ALEXIS MORAN, University of Florida, DONGHEE LEE, SANGJIN RYU, University of Nebraska-Lincoln — Hydrogel substrates of tunable stiffness have been actively pursued for a variety of in vitro cell mechanobiology study. Here we present a new method to fabricate Janus polyacrylamide gel based on limited mixing between liquid drops coalescing in the Hele-Shaw geometry. Two pre-polymer drops with different concentrations were sandwiched and squeezed between two parallel glass surfaces. Once the drops coalesced in the decreased gap between the surfaces, gelation was initiated by UV light exposure with various time delays. AFM nano-indentation was utilized to map the Young’s modulus of obtained gels. Fabricated Janus gels had two regions of different Young’s moduli interfacial by the stiffness gradient zone, and the width of the gradient zone increased with the delay time.

1We acknowledge support from Bioengineering for Human Health grant from UNL and UNMC, and NSF REU grant for UNL.

KP1.00167 Metastable states of rigid bodies in a flow. ASHWIN VAIYDA, DORALIA CASTILLO, MATT CRISTALDI, Montclair State University, BONGJAE CHUNG, George Mason University, KARINA SORIANO, HAIYAN SU, Montclair State University — Symmetric bodies such as cylinders and spheroids, in their terminal stable states, are long known to have their long axes align themselves perpendicular to the direction of flow. This property has been confirmed in sedimentation and horizontal flow setup and the transition to a terminal stable state is believed to coincide with the onset of significant inertial effects in the flow. However, the threshold at which this transition occurs is yet unknown. In this presentation we report a recent experimental study to examine the nature of the transition of prolate spheroids and cylinders of various aspect ratios, from initial to their terminal stable equilibrium. Our experiments reveal that the body reorients itself from its initial orientation, continuing to an angle perpendicular to the flow direction, revealing possible steady intermediate states. Are these “metastable states” real or an artifact of our experiments? Experiments performed on a variety of bodies and using different suspension mechanism show this state to be persistent. A three dimensional numerical simulations performed by us provide strong support for these observations and reveals new findings about the nature of the torque imposed on the body due to the flow at changing Reynolds numbers.

KP1.00168 Mass flow rate of granular material flowing from tilted bins. JAIME KLAPP, Instituto Nacional de Investigaciones Nucleares and ABACUS Departamento de Matematicas, Cinvestav IPN, ABRAMAH MEDINA, SEPI ESIIME Azcapotzalco, IPN; AYAX HEADMAN ON TROPICS VICTORIO, Universidad Politecnica del Valle de Mexico, SALOMON PERALTA LOPEZ, Mexican Petroleum Institute — We report experiments performed to describe the behavior of the experimental mass flow rate of cohesionless granular material, $M_{\beta expt}$, through circular orifices of diameter $D$ made on sidewalks of tilted bins. In such experiments, the influence of the wall thickness of the bin, $w$, and the tilt angle respect to the vertical, $\beta$, were also regarded. The experimental measurements, using beach sand and granulated sugar, yield a linear correlation among $M_{\beta expt}$ and a theoretical piecewise correlation of the mass flow rate, $M_{\beta}$, which is valid for the overall range of values of $\beta$. Numerical simulation will be also a discussed.
The evolution of the liquid surface is computed numerically and compared with the results of simple experiments. The motion of the liquid is confined to a thin layer around the line of intersection whose height increases again as a power law of time and the angle of inclination of the plates and is attained at a point that reaches the line of intersection only after a certain time. At later times, the motion of the liquid is confined to a thin layer around the line of intersection whose height increases again as a power law of time and the exponent of the power law is a function of the angle of inclination. The thickness of the film decreases as the inverse of the power law of time. The evolution of the liquid surface is computed numerically and compared with the results of simple experiments.

Development of Schlieren Imaging for Analysis of Supersonic Complex Multi-stream Rectangular Nozzle

Reduction of aerodynamic load fluctuation on wind turbine blades through active flow control

Investigation of Richtmyer-Meshkov turbulent mixing using front tracking method

The Fluid Mechanics of the Bible: Miracles Explainable by Christian Science?

We would like to acknowledge SBIR Phase 2 with Spectral Energies under direction of Barry Kiel (Program Manager)
KP2.00002 Large-eddy simulation of zero-pressure-gradient turbulent boundary layer with solid particle suspension. MUSTAFA RAHMAN, RAVI SAMTANEY, King Abdullah University of Science and Technology — We present results of solid particle suspension and transport in a fully-developed turbulent boundary layer flow using large-eddy simulation of the incompressible Navier-Stokes equations. We adopt the Eulerian-Eulerian approach to simulating particle laden flow with a large number of particles, in which the particles are characterized by statistical descriptors. For the particulate phase, the direct quadrature method of moments (DQMOM) is chosen in which the weights and abscissas of the quadrature approximation are tracked directly rather than the moments themselves. The underlying approach in modeling the turbulence of fluid phase utilizes the stretched spiral vortex subgrid-scale model and a virtual wall model similar to the work proposed by Inoue & Pullin (J. Fluid Mech. 2011). The solver is verified against simple analytical solutions and the computational results are found to be in a good agreement with these. The capability of the new numerical solver will be exercised to investigate turbulent transport of sand in sandstorms. Finally, the adequacy and limitations of the solver will be discussed.

1Supported by the KAUST Office of Competitive Research Funds under Award No. URF/1/1704-01.

KP2.00003 Physical modeling of the atmospheric boundary layer for wind energy and wind engineering studies. GREGORY TAYLOR-POWER, JOHN TURNER, MARTIN WOSNIK, University of New Hampshire — The Flow Physics Facility (FPF) at UNH has test section dimensions W6.0m, H2.7m, L=72m. It can achieve high Reynolds number boundary layers, enabling turbulent boundary layer, wind energy and wind engineering research with exceptional spatial and temporal instrument resolution. We examined the FPF’s ability to experimentally simulate different types of the atmospheric boundary layer (ABL): the stable, unstable, and neutral ABL. The neutral ABL is characterized by a zero potential temperature gradient, which is readily achieved in the FPF by operating when air and floor temperatures are close to equal. The stable and unstable ABLs have positive and negative vertical temperature gradients, respectively, which are more difficult to simulate without direct control of air or test section floor temperature. The test section floor is a 10 inch thick concrete cement slab and has significant thermal mass. When combined with the diurnal temperature variation of the ambient air, it is possible to achieve vertical temperature gradients in the test section, and produce weakly stable or weakly unstable boundary layer. Achievable Richardson numbers and Obukhov lengths are estimated. The different boundary layer profiles were measured, and compared to theoretical atmospheric models.

1Supported by UNH Hamel Center for Undergraduate Research SURF

KP2.00004 Rayleigh-Bénard convection at high Prandtl numbers in circular and square geometry. STEPHEN R. JOHNSTON, Georgia Institute of Technology, ENRICO FONDA, KATEPALLI R. SREENIVASAN, New York University, DEVESH RANJAN, Georgia Institute of Technology — Experiments using water and simulations have shown that flow structures and turbulent fluctuations in Rayleigh-Bénard convection are affected by the shape of the container. We study the effect of the geometry in both square and cylindrical test cells of aspect ratio of order unity in high Prandtl fluids (up to 104). Flow visualization using a photochromic dye seeded throughout the fluid allows us to uninvassively study the evolution of the large scale structures. We discuss the observations in the two geometries and compare them with previous observations at low Prandtl numbers.

KP2.00005 The Negligible Role of Thermal Inertia in the Marangoni Instability and Evaporation Instability Problems. JOHN SHREFLER, Univ of Florida - Gainesville, GEORG DIETZE, Univ. Paris-Sud, CNRS, Lab. FAST, RANGA NARAYANAN, Univ of Florida - Gainesville — The classical Marangoni instability problem, principally introduced by Pearson in 1958, takes the interface of two fluids heated from below to be non-deflecting and as a consequence of this assumption thermal inertia is responsible for the onset of convective flows. However, in practical problems where interfacial deflections are uncontrollable and necessarily present, we find that thermal inertia is of very little importance for a broad class of fluids. Neglecting the contribution of thermal inertia, we find that the instability persists as in the classical case. This is shown by way of an integral boundary layer method for long wavelength flows and confirmed by detailed calculations using the full equations. We aim to demonstrate that the principal result that thermal inertia is unimportant continues to hold for the evaporation instability problem, provided pure component phase change is considered.

1Supported by NSF 0968313 and Marie Curie IRSES grants.

KP2.00006 First Signs of Flow Reversal Within a Separated Turbulent Boundary Layer. JARED HAMMERTON, AMY LANG, The University of Alabama — A shark’s skin is covered in millions of microscopic scales that have been shown to be bristle in a reversing flow. The motive of this project is to further explore a potential bio-inspired passive separation control mechanism which can reduce drag. To better understand this mechanism, a more complete understanding of flow reversal within the turbulent boundary layer is required. In order to capture this phenomenon, water tunnel testing at The University of Alabama was conducted. Using a long flat plate and a rotating cylinder, a large turbulent boundary layer and adverse pressure gradient were generated. Under our testing conditions the boundary layer had a Reynolds number of 200,000 and a boundary layer height in the testing window of 5.6 cm. The adverse pressure gradient causes the viscous length scale to increase and thus increase the size of the individual components of the turbulent boundary layer. This will make the low speed streaks approximately 1 cm in width and thus large enough to measure. Results will be presented that test our hypothesis that the first signs of flow reversal will occur within the section of lowest momentum located furthest from the wall, or within the low speed streaks.

1This Project was funded by NSF REU Site Award 1358991

KP2.00007 ABSTRACT WITHDRAWN —
KP2.00008 Application of Piezoelectrics to Flapping-Wing MAVs1, ALEX WIDSTRAND, Smith College, J. PAUL HUBNER, The University of Alabama — Micro air vehicles (MAVs) are a class of unmanned aerial vehicles that are size-restricted and operate at low velocities and low Reynolds numbers. An ongoing challenge with MAVs is that their flight-related operations are highly constrained by their size and weight, which limits battery size and, therefore, available power. One type of MAV called an ornithopter flies using flapping wings to create both lift and thrust, much like birds and insects do. Further bio-inspiration from bats led to the design of membrane wings for these vehicles, which provide aerodynamic benefits through passive vibration. In an attempt to capitalize on this vibration, a piezoelectric film, which generates a voltage when stressed, was investigated as the wing surface. Two wing planforms with constant area were designed and fabricated. The goal was to measure the wings’ flight characteristics and output energy in freestream conditions. Complications with the flapper arose which prevented wind tunnel tests from being performed; however, energy data was obtained from table-top shaker tests. Preliminary results indicate that wing shape affects the magnitude of the charge generated, with a quarter-elliptic planform outperforming a rectangular planform.1

1Funding provided by NSF REU Site Award number 1358991.

KP2.00009 Ultra-high speed measurement of a laser-induced underwater shock wave1, KEI SUKE HAYASAKA, YOSHI YUKI TAGAWA, Tokyo Univ of Agri & Tech — We find that a laser-induced underwater shock wave has an interesting character: peak pressure varies in propagation direction while pressure impulse is the same for all directions. In general, a shock wave is often approximated as a single-spherical wave, which seems to contradict with the aforementioned character. In this research, we investigate a structure of a laser-induced underwater shock wave in order to rationalize the character. We utilize an ultra-high speed camera to visualize the shock waves and plasmas. Our measurement results reveal that the shock wave and the plasma consist of multiple spherical-shock waves and multiple plasmas. We here suggest a simple model of multiple-shock waves: a laser-induced shock wave can be interpreted as a collection of spherical shocks originated from multiple plasmas. This model explains both the different peak pressures and the same pressure impulses. In addition, we estimate shock pressure in a two-dimensional field using non-invasive optical methods, which measure a projected density field of a shock wave.1

1JSPS KAKENHI Grant Number26709007

KP2.00010 Three-dimensional blade coating of complex fluid, VACHITAR SINGH, EMMA GRIMALDI, NYU Polytechnic School of Engineering, ALBAN SAURET, SVI, CNRS/Saint-Gobain, EMILIE DRESAIRE, NYU Polytechnic School of Engineering — The application of a layer of non-Newtonian fluid on a solid substrate is an important industrial problem involved in polymer or paint coatings, and an everyday life challenge when it comes to spreading peanut butter on a toast. Most experimental and theoretical work has focused on the two-dimensional situation, i.e. the scraping of a fixed blade on a moving substrate to turn a thick layer of liquid into a thin coat. However, the spreading of a finite volume of non-Newtonian fluid using a blade has received less attention, despite significant practical and fundamental implications. In this study, we investigate experimentally the spreading of a finite volume of a model non-Newtonian fluid, carbopol, initially deposited against the fixed blade. As the substrate is translated at constant speed, we characterize the dynamics of spreading and the final shape of the coated layer. We measure and rationalize the influence of the liquid volume, the height and orientation of the blade, and the speed of the substrate on the spreading.

KP2.00011 FLUID DYNAMICS STUDENT POSTER COMPETITION - CFD –

KP2.00012 ABSTRACT WITHDRAWN –

KP2.00013 Design of a rapid magnetic microfluidic mixer, MATTHEW BALLARD, DREW OWEN, ZACHARY GRANT MILLS, SRINIVAS HANASOGO, PETER HESKETH, ALEXANDER ALEXEEV, Georgia Institute of Technology — Using three-dimensional simulations and experiments, we demonstrate rapid mixing of fluid streams in a microchannel using orbiting magnetic microbeads. We use a lattice Boltzmann model coupled to a Brownian dynamics model to perform numerical simulations that study in depth the effect of system parameters such as channel configuration and fluid and bead velocities. We use our findings to aid the design of an experimental micromixer. Using this experimental device, we demonstrate rapid microfluidic mixing over a compact channel length, and validate our numerical simulation results. Finally, we use numerical simulations to study the physical mechanisms leading to microfluidic mixing in our system. Our findings demonstrate a promising method of rapid microfluidic mixing over a short distance, with applications in lab-on-a-chip sample testing.

KP2.00014 Controlling Hazardous Releases while Protecting Passengers in Civil Infrastructure Systems1, SARA P. RIMER, NIKOLAOS D. KATOPODES, University of Michigan - Ann Arbor — The threat of accidental or deliberate toxic chemicals released into public spaces is a significant concern to public safety, and the real-time detection and mitigation of such hazardous contaminants has the potential to minimize harm and save lives. Furthermore, the safe evacuation of occupants during such a catastrophe is of utmost importance. This research develops a comprehensive means to address such scenarios, through both the sensing and control of contaminants, and the modeling of and potential communication to occupants as they evacuate. A computational fluid dynamics model is developed of a simplified public space characterized by a long conduit (e.g. airport terminal) with unidirectional ambient flow that is capable of detecting and mitigating the hazardous contaminant (via boundary ports) over several time horizons using model predictive control optimization. Additionally, a physical prototype is built to test the real-time feasibility of this computational flow control model. The prototype is a blower wind-tunnel with an elongated test section with the capability of sensing (via digital camera) an injected ‘contaminant’ (propylene glycol smoke), and then mitigating that contaminant using actuators (compressed air operated vacuum nozzles) which are operated by a set of pressure regulators and a programmable controller. Finally, an agent-based model is developed to simulate “agents” (i.e. building occupants) as they evacuate a public space, and is coupled with the computational flow control model such that agents must interact with a dynamic, threatening environment.1

1NSF-CMMI #0856438

KP2.00015 Modifying Airfoils for Low Reynolds Flight, CHRISTOPHER ONG, MARIA-ISABEL CARNASCIALI, University of New Haven — There has been increased interest in Micro Air Vehicles (MAV) by both the private and government sectors. MAVs are miniature classed-UAVs that can operate in tighter spaces in urban or wooded regions. Sizes vary—from that of an insect to that of small bird—depending on intended functionality and usually operate at much lower speeds. Studies have shown that the aerodynamic performance of well-known airfoils can change significantly at low Reynolds numbers. In this work, we examine via parametric CFD analysis tools the behavior of airfoils at low Reynolds values. Furthermore, we investigate the impact of adding bio-inspired features to the airfoils such as humps or dimples. Results will be presented in comparison to established values.
KP2.00016 Controlling Wavebreaking in a Viscous Fluid Conduit, DALTON ANDERSON, MICHELLE MAIDEN, MARK HOFEEF, University of Colorado Boulder — This poster will present a new technique in the experimental investigation of dispersive hydrodynamics. In shallow water flows, internal ocean waves, superfluids, and optical media, wave breaking can be resolved by a dispersive shock wave (DSW). In this work, an experimental method to control the location of DSW formation (gradient catastrophe) is explained. The central idea is to convert an initial value problem (Riemann problem) into an equivalent boundary value problem. The system to which this technique is applied is a fluid conduit resulting from high viscosity contrast between a buoyant interior and heavier exterior fluid. The conduit cross-sectional area is modeled by a nonlinear, conservative, dispersive, third order partial differential equation. Using this model, the aim is to predict the breaking location of a DSW by controlling one boundary condition. An analytical expression for this boundary condition is derived by solving the dispersion relation equation backward in time from the desired step via the method of characteristics. This is used in experiment to generate an injection rate profile for a high precision piston pump. This translates to the desired conduit shape. Varying the jump height and desired breaking location indicates good control of DSW formation. This result can be improved by deriving a conduit profile by numerical simulation of the full model equation. Controlling the breaking location of a DSW allows for the investigation of dynamics independent of the boundary.

Support provided by NSF CAREER DMS-1255422, NSF EXTREEMS

KP2.00017 Modeling Droplet Motion on Liquid-Infused Surface Using Lattice Boltzmann Method, MINGFEI ZHAO, XIN YONG, Binghamton Univ — Understanding self-assembly of nanoparticles driven by the evaporation of the particle-covered sacrificial liquid mass dispensed on a solid substrate is of technological importance for various printing and deposition techniques. Although the convective deposition of suspended nanoparticles (known as the coffee ring effect) has been studied extensively, the self-assembly of nanoparticles directly delivered to the liquid-gas interface remains unexplored. In this work, we develop a hybrid model that combines free-energy multiphase LBM with Lagrangian particle tracking method to reveal the complex interplay between nanoparticles, convective flow in liquid, and the dynamics of three-phase contact line on the substrate. We first verify our computational model using existing experimental and experimental results. We then investigate the evaporation phenomena of a particle-covered droplet with specified nanoparticle distributions and wetting properties. By controlling the boundary conditions, we can implement desired contact angle hysteresis on the substrate to create a substrate that matches experimental observations. This study provides a theoretical framework to explore the dynamics of nanoparticle self-assembly at evaporating liquid-vapor interfaces.

Support provided by KAUST Baseline Research Funds

KP2.00018 A finite volume method for fluctuating hydrodynamics of simple fluids, KIRAN NARAYANAN, RAVI SAMANTANEY, BRIAN MORAN, King Abdullah University of Science and Technology — Fluctuating hydrodynamics accounts for stochastic effects that arise at mesoscopic and macroscopic scales. We present a finite volume method for numerical solutions of the fluctuating compressible Navier Stokes equations. Case studies for simple fluids are demonstrated via the use of two different equations of state (EOS): a perfect gas EOS, and a Lennard-Jones EOS for liquid argon developed by Johnson et al. (Mol. Phys. 1993). We extend the fourth order conservative finite volume scheme originally developed by McCorquodale and Colella (Comm. in App. Math. & Comput. Sci. 2011), to evaluate the deterministic and stochastic fluxes. The expressions for the cell-centered discretizations of the stochastic shear stress and stochastic heat flux are adopted from Espanol, P (Physica A. 1998), where the discretizations were shown to satisfy the fluctuation-dissipation theorem. A third order Runge-Kutta scheme with weights proposed by Delong et. al. (Phy. Rev. E. 2013) is used for the numerical time integration. Accuracy of the proposed scheme will be demonstrated. Comparisons of the numerical solution against theory for a perfect gas as well as liquid argon will be presented. Regularizations of the stochastic fluxes in the limit of zero mesh sizes will be discussed.

KP2.00019 FLUID DYNAMICS STUDENT POSTER COMPETITION - MICROFLOWS, DROPLETS AND BUBBLES —

KP2.00020 Modeling a Bouncing Droplet with a Lubrication Force, MATTHEW CESSNA, Claremont Graduate University — In our laboratory experiments, a shallow bath of silicone oil of a particular viscosity was placed on an electromagnetic shaker where it was driven by a constant frequency just below the threshold of Faraday instability. A small droplet was then deposited on the bath and allowed to evaporate. The dynamic behavior of the droplet was monitored using a high-speed video camera. The droplet velocity and acceleration of the droplets measured below -1 g at the moment when a droplet was being launched back into flight by the oscillating bath. We investigate whether lubrication theory accounts for these measurements and model a bouncing droplet on a vibrating bath of the same viscosity with a lubrication force to reproduce our experimental data using Matlab.

Support provided by NSF CAREER DMS-1255422, NSF EXTREEMS

KP2.00021 Partial Conservation of Liquid Cones with an Electrified Plate, MADELINE ZHANG, CASEY BARTLETT, JAMES BIRD, Boston University — As a liquid drop contacts a surface in the presence of an electric field, charge can promote rapid spreading upon contact. In the moments prior to contact, the drop will often develop a conical tip that is qualitatively similar to that seen in droplet pairs undergoing electrocoalescence. Recently it has been shown that in strong electric fields, droplet pairs can contact and then immediately recoil. Yet, it is unclear whether a similar phenomenon exists for droplets that contact charged solid surfaces. Here, we show that droplets can indeed be repelled from dry rigid surfaces, provided that there is sufficient charge. Our high-speed experiments reveal that when an electric field deforms the contacting drop beyond a critical cone angle, the drop will recoil rather than spread. This critical angle is significantly greater than the critical angle previously observed for the identical drop pair coalescence-recoil transition, but is consistent with surface tension-driven dynamics.

KP2.00022 Experimental study on the motion of a pair of bubbles in quiescent liquids, HIROAKI KUSUNO, TOSHIYUKI SANADA, Shizuoka Univ. — Understanding of the bubble-bubble interaction problem is important step to achieve more accurate bubbly flow simulation. Some theoretical models of bubble-bubble interaction have been proposed. And some numerical results have also been reported. However, the experimental verifications are insufficient. In this study, we experimentally investigated the motion of a pair of bubbles initially positioned in-line configuration in ultrapure water or an aqueous surfactant solution. The bubble motion were observed by two high speed video cameras. The bubbles Reynolds number was ranged from 50 to 300. In ultrapure water, initially the trailing bubble deviated from the vertical line on the leading bubble owing to the wake of the leading bubble. And then, the slight difference of the bubble radius changed the relative motion. When the trailing bubble slightly larger than the leading bubble, the trailing bubble approached to the leading bubble due to it’s buoyancy difference. The bubbles attracted and collided only when the bubbles rising approximately side by side configuration. In addition, we will also discuss the motion of bubbles rising in an aqueous surfactant solution.
KP2.00023 Polymer and protein interfacial competition in a shell production process1, EMMA WILLARD, University of California Davis, GREG RANDALL, General Atomics — We are exploring oil-in-aqueous polymer compound droplet formulations to UV polymerize into shells while in a strong AC electric field (kV/cm, 20 MHz). The electric field drives the drops to adopt a concentric configuration so that a “perfect” spherical shell can be polymerized with a uniform wall thickness. In our previous study of oil-in-water droplet centering, we determined that droplet stretching in the electric field was a problem, which we overcame by using protein additives to strengthen the oil/water interface. However, adding polymer to the shell fluid has been shown to weaken the droplet interface and further complicates T junction droplet generation. In this work, we study the adsorption competition between bovine serum albumin and polyethylene glycol diacylate with the pendant drop method to generate a polymer/protein shell formulation that will resist stretching in the centering electric field. Furthermore, we explore droplet generation of polymer/protein shell formulations in a double T junction and stretching in an electric field.

1Work supported by General Atomics IR&D funds.

KP2.00024 A new device for generating thin jets of highly-viscous liquid1, HAJIME ONUKI, YUTO OI, YOSHIYUKI TAGAWA, Tokyo Univ of Agri & Tech — Thin liquid jets are applied to various devices, such as ink-jet printers. However, it is challenging to generate liquid jets of highly-viscous liquids (~1.000 cSt) using existing methods. To overcome this challenge, we invent a highly-viscous liquid-jet generator. This device has simple structure as follows: a wettable-thin tube is inserted into a liquid filled container. We keep the liquid level inside a thin tube deeper than that outside of the tube. When an impulsive force acts on the bottom of the container, a thin jet is generated. The jet is up to 20 times faster than the initial velocity given by the impulsive force. We successfully generate jets with a wide range of viscosity (1-1.000 cSt). We also propose the physical model based on pressure-impulse approach to rationalize its mechanism. Inside the thin tube, a gradient of pressure impulse is much larger than that outside of the tube. We verify the performance of our device experimentally. We find that the proposed model can describe all experimental results in this research.

1JSPS KAKENHI Grant Number 26709007

KP2.00025 Modeling Electrospay Deposition of Nanoparticle Inks, AO LI, Binghamton Univ, JEFFERSON FIDELES DA SILVA, Murray State University at Murray, KY, XIN YONG, Binghamton Univ — Electrospay of nanoparticle inks is of great importance to the manufacturing of functional materials. In this study, we develop a new three-dimensional multiphysics method to model the electrospay of colloidal suspension to a flat substrate. The Lagrangian Particle Tracking (LPT) transport equation is coupled to mass and heat transfer using convective droplet vaporization model, which allow us to track each particle-laden ink droplets and dry nanoparticles in the electrospay plume and probe the deposit structures. Herein, we consider dilute inks that are experimentally relevant, assuming monodisperse nanoparticles. We characterize the overall statistics of the plume and the dynamics of individual ink droplet or dry nanoparticle. It is shown that the segregation effect affects not only primary and satellite droplets but also dry nanoparticles. We observe nanoparticles deposit structure changing process, in particular time evolution of the density profile along radial direction. Our results show that the region of high nanoparticle density transitioning from only the edge to both the edge and center, which agrees with previous experimental studies.

KP2.00026 Experimental Research on the Capture of Fine Particles in a High-voltage Electric Field, XING JIN, SHUIQING LI, Tsinghua University — Mechanisms for capturing of fine particles through a high-voltage electric field were examined using the electrostatic precipitator (ESP) as an example system. The dimensionless equations governing particle transport were solved and a laboratory-scale ESP was experimentally examined. The analysis indicates that particles in the size range of 0.1-1.0 µm have the lowest electric migration velocity and there is a capture-effective zone in the middle of the ESP for fine particles. Subsequent increase in length had little effect for grade efficiency because of the influence of electrohydrodynamic (EHD) flow. In the particle boundary layer zone, dipole-dipole force and VDW force play crucial roles in capturing fine particles. The packing structure of fine particles on the collecting plate is investigated by digital microscopy technology. The effects of pre-charging, pre-polarization and external electric field on packing morphologies are discussed. It is found that the dipole-dipole force between particles causes the formation of long particle chains and the maximum length of particle dendrites during the packing is dependent on both the density of external field and deposit structure.

KP2.00027 Volume of a laser-induced microjet, SENNOSUKE KAWAMOTO, KEISUKE HAYASAKA, YUTO NOGUCHI, YOSHIYUKI TAGAWA, Tokyo Univ of Agri & Tech — Needle-free injection systems are of great importance for medical treatments. In spite of their great potential, these systems are not commonly used. One of the common problems is strong pain caused by diffusion shape of the jet. To solve this problem, the usage of a high-speed highly-focused microjet as needle-free injection system is expected. It is thus crucial to control important indicators such as ejected volume of the jet for its safe application. We conduct experiments to reveal which parameter influences mostly the ejected volume. In the experiments, we use a glass tube of an inner diameter of 500 micro-meter, which is filled with the liquid. One end is connected to a syringe and the other end is opened. Radiating the pulse laser instantaneously vapors the liquid, followed by the generation of a shockwave. We find that the maximum volume of a laser-induced bubble is approximately proportional to the ejected volume. It is also found that the occurrence of cavitation does not affect the ejected volume while it changes the jet velocity.

KP2.00028 FLUID DYNAMICS STUDENT POSTER COMPETITION - BIOFLUIDS

KP2.00029 A Modification of the Levich Model to Flux at a Rotating Disk in the presence of Planktonic Bacteria, AKHENATON-ANDREW JONES, CULLEN BUIE, Massachusetts Inst of Tech-MIT — The Levich model of flow at a rotating disk describes convective mass transport to a disk when edge effects and wall effects can be neglected. It is used to interpret electrochemical reaction kinetics and electrochemical impedance of flow systems. The solution has been shown to be invalid for high densities (~1%vol/vol) of inert, non-motile nano-sized particles (<0.1 µm) and macro-particles (>1.5 µm), yet little work has been done for motile bacteria and bacterial sized particles. The influence of planktonic bacteria on rotating disk experiments is crucial for the evaluation of electrochemically active biofilms. In this work, we show that the presence of bacteria creates significant deviation from the ideal Levich model not shared by inert particles. We also study the impact of dead (fixed) bacteria on deviation form the Levich model. This work has implications for studies of microbial induced corrosion, microbial adhesion, and antibiotic transport to adhered biofilms preformed in rotating disk systems.
Intraventricular flow alterations due to dyssynchronous wall motion\textsuperscript{1}, AU-DREY M. POPE, Stillwater High School, HONG KUAN LAI, MILAD SAMAE, ARVIND SANTHANAKRISHNAN, Oklahoma State University — Roughly 30% of patients with systolic heart failure suffer from left ventricular dyssynchrony (LVD), which is a common cause of sudden death in young athletes. The myocardium becomes abnormally thick in HCM and deforms the internal geometry of the left ventricle (LV). Previous studies have shown that a vortex is formed during diastolic filling, and further that the dilated LV morphology seen in systolic heart failure results in altering the filling vortex from elliptical to spherical shape. We have also previously shown that increasing LV wall stiffness decreases the filling vortex circulation. However, alterations to intraventricular filling fluid dynamics due to an obstructive LV morphology and locally elevated wall stiffness (in the hypertrophied region) have not been previously examined from a mechanistic standpoint. We conducted an experimental study using an idealized HCM physical model and compared the intraventricular flow fields obtained from 2D PIV to a baseline LV physical model with lower wall stiffness and anatomical geometry. Flow through the LV model was driven using a piston pump, and stepper motors coupled to the above shafts were used to locally perturb the septal walls segments relative to the pump motion. 2D PIV was used to examine the intraventricular flow through the LV physical model. Alterations to SL delay results in a reduction in the kinetic energy (KE) of the flow field compared to synchronous SL motion. The effect of varying SL motion delay from 0% (synchronous) to 100% (out-of-phase) on KE and viscous dissipation will be presented.

\textsuperscript{1}This research was supported by the Oklahoma Center for Advancement of Science and Technology (HR14-022).

Diastolic filling in a physical model of obstructive hypertrophic cardiomyopathy, JOSEPH SCHOVANECE, MILAD SAMAE, HONG KUAN LAI, ARVIND SANTHANAKRISHNAN, Oklahoma State University — Hypertrophic Cardiomyopathy (HCM) is an inherited heart disease that affects as much as one in 500 individuals, and is the most common cause of sudden death in young athletes. The myocardium becomes abnormally thick in HCM and deforms the internal geometry of the left ventricle (LV). Previous studies have shown that a vortex is formed during diastolic filling, and further that the dilated LV morphology seen in systolic heart failure results in altering the filling vortex from elliptical to spherical shape. We have also previously shown that increasing LV wall stiffness decreases the filling vortex circulation. However, alterations to intraventricular filling fluid dynamics due to an obstructive LV morphology and locally elevated wall stiffness (in the hypertrophied region) have not been previously examined from a mechanistic standpoint. We conducted an experimental study using an idealized HCM physical model and compared the intraventricular flow fields obtained from 2D PIV to a baseline LV physical model with lower wall stiffness and anatomical geometry. The obstruction in the HCM model leads to earlier breakdown of the filling vortex as compared to the anatomical LV. Intraventricular filling in both models under increased heart rates will be discussed.

Nonlinear dynamics of flame front instability induced by radiative heat loss: period-doubling bifurcation and chaos, HIKARU KINUGAWA, KAZUHIRO UEDA, Department of Mechanical Engineering, Ritsumeikan University, 1-1-1 Nojihigashi, Kusatsu, Shiga 525-8577, Japan, HIROSHI GOTO, Department of Mechanical Engineering, Tokyo University of Science, 6-3-1 Nijuku, Katsushika-ku, Tokyo 125-8585, Japan — We numerically study the nonlinear dynamics of flame front instability induced by radiative heat loss on the basis of dynamical systems theory. Our previous studies have shown that the radiative heat loss significantly produces the deterministic chaos of flame front temperature fluctuations throughout the period-doubling bifurcation known as Feigenbaum scenario [Gotoda et al., Combust. Theor. Model. 14, 479-493 (2010)], while its short-term behavior can be predicted using a local and global nonlinear predictors [Gotoda et al., Chaos 22, 033106 (2012)]. The present study reports that the similar kind of bifurcation process clearly appears at the fuel concentration, and that the fuel concentration dynamics in the well-developed chaos region is much more complicated than that of the flame front temperature. Recurrence quantification analysis we adopted in the present study can quantify the significant changes in the dynamics in the chaos region that cannot be capture in the bifurcation diagram.

Characterization of degeneration process in thermo-acoustic combustion instability using dynamical systems theory, KENTA HAYASHI, Department of Mechanical Engineering, Ritsumeikan University, HIROSHI GOTO, Department of Mechanical Engineering, Tokyo University of Science, YUTA OKUNO, Department of Mechanical Engineering, Ritsumeikan University, SHIGERU TACHIBANA, Institute of Aeronautical Technology, Japan Aerospace Exploration Agency, TOKYO UNIVERSITY OF SCIENCE COLLABORATION, JAPAN AEROSPACE EXPLOSION AGENCY COLLABORATION — We have experimentally investigated the degeneration process of combustion instability in a lean premixed gas-turbine model combustor on the basis of dynamical systems theory. Our previous study reported that with increasing the equivalence ratio, the dynamical behavior of combustion state close to lean blowout transits from stochastic fluctuations to periodic thermoacoustic combustion oscillations via low-dimensional chaotic oscillations (Gotoda et al., Chaos, 21, 013124 (2011) / Gotoda et al., Chaos, 22, 043128 (2012)). The further increase in the equivalence ratio gives rise to the quasi-periodic oscillations and the subsequent chaotic oscillations with small amplitudes. The route to chaotic oscillations is quantitatively shown by the use of nonlinear time series analysis involving the color recurrence plots, permutation entropy and local predictor.

Stratification effects on laminar premixed-flame response to mixture perturbations, TIERNAN CASEY, JYH-YUAN CHEN, Univ of California - Berkeley — While complete mixing on the molecular level is desirable for ensuring that combustion processes are limited by chemical kinetics rather than mass transport, it is often the case that practical devices operate with some degree of unmixedness. As such, phenomena such as ignition or flame propagation will inevitably occur in regions that exhibit mixture or thermal non-uniformity. Here we present unsteady simulations of laminar premixed flames in the low-Mach limit subject to mixture perturbations of varying wavelength and amplitude, and qualify their effect on the flame behavior. When flames experience variations in mixture the transport processes in the flame zone vary with time and the flame behavior can depend on the burned gas history. Also, the possibility of extending flames beyond their flammability limits so as to maximize the overall mass of fuel burned is explored by exploiting these unsteady effects.

High-Fidelity Simulations of Electrically-Charged Atomizing Diesel-Type Jets, BENNOIT CAILLARD, Ecoles de Saint-Cyr Coëtquidan, MARK OWKES, Montana State University, BRETT VAN POPPEL, West Point Military Academy — Combustion of liquid fuels accounts for over a third of the energy usage today. Improving efficiency of combustion systems is critical to meet the energy needs while limiting environmental impacts. Additionally, a shift away from traditional fossil fuels to bio-derived alternatives requires fuel injection systems that can atomize fuels with a wide range of properties. In this work, the potential benefits of electrically-charged atomization is investigated using numerical simulations. Particularly, the electrostatic forces on the hydrodynamic jet are quantified and the impact of the forces is analyzed by numerical simulations of Diesel-type jets at realistic flow conditions. These studies are performed using a state-of-the-art numerical framework that globally conserves mass, momentum, and the electric charge density even at the gas-liquid interface where discontinuities exist.
Session L1 Porous Media Flows: General  Auditorium - Stefan Llewellyn Smith, UCSD

4:05PM L1.00001 Excess pore water pressure due to ground surface erosion. STEFAN LLEWELLYN SMITH, Department of Mechanical and Aerospace Engineering, UCSD, STEVEN GAGNIER, UCSD — Erosional unloading is the process whereby surface rocks and soil are removed by external processes, resulting in changes to water pressure within the underlying aquifer. We consider a mathematical model of changes in excess pore water pressure as a result of erosional unloading. Neuzil and Pollock (1983) studied this process in the case where the water table initially coincides with the surface. In contrast, we analyze an ideal aquifer which is initially dry. We model the ground surface movement from the ground surface to the water table and solve using Laplace Transform methods in the case of a constant ground surface elevation, with a booster derived by King (1985). The boost operator is used to boost the solution (in the Laplace domain) to a frame of reference moving at constant velocity with respect to the original frame. We use our solution to analyze the evolution of the pressure during erosion of the aquifer itself for small and large erosion rates. We also examine the flux at the upper boundary as a function of time and present a quasi-steady approximation valid for very small erosion rates in the appendix.

4:18PM L1.00002 A strategy for optimising well placement by combining historical well data with a geological model of a porous rock. A.J. EVANS, BP Institute, University of Cambridge, C.P. CAULFIELD, BP Institute & DAMTP, University of Cambridge, ANDREW W. WOODS, BP Institute, University of Cambridge — Flow in porous media is subject to large uncertainties due to sparsity of available data and heterogeneity of reservoir properties over a range of length scales. We investigate the reduction in uncertainty which can be achieved through inversion of flux data between a point source and a point sink. A Monte Carlo simulation with stochastically generated permeabilities conditioned by flux data is used to estimate flux statistics for relocated wells. We demonstrate how the correlation length scale of the permeability influences the reduction in uncertainty for new well positions. Uncertainty is seen to be reduced for well positions within a region around the original well sites. This region scales with the permeability correlation length. Finally, we show that a linearised method for flux estimation shows good agreement to fully non-linear simulations with a considerable reduction in computation time.

4:31PM L1.00003 A volume-balance model for flow on porous media1. CARLOS MALAGA, FRANCISCO MANDUJANO, Physics Department, School of Science, Universidad Nacional Autónoma de México, JULIAN BECERRA, None — Volume-balance models are used by petroleum engineers for simulating multiphase and multicomponent flow phenomena in porous media and the extraction process in oil reservoirs. In these models, mass conservation equations and Darcy's law are supplemented by a balance condition for the pore and fluid volumes. This provides a pressure equation suitable for simulating a compressible flow within a compressible solid matrix. Here we present an alternative interpretation of the volume-balance condition that includes the advective transport within a consolidated porous media. We obtain a modified equation for the time evolution of the pressure field. Numerical tests for phase separation under gravity are presented for multiphase three dimensional flow in heterogeneous porous media.

1 The authors acknowledge funding from Fondo Sectorial CONACYT-SENER grant number 42536 (DGJ-SPI-34-170412-217)

4:44PM L1.00004 The permeability of poly-disperse porous media and effective particle size. B.I. MARKICEVIC, C. PRESTON, Pall Corp., S. OSTERROTH, O. ILIEV, Fraunhofer ITWM and University of Kaiserslautern, M. HURWITZ, Cornell University — The interactions between the fluid and solid phases in porous media account for the openness and length of the flow path that the fluid needs to travel within. The same reasoning applies for both mono- and poly-disperse media, and is reflected in the adoption of the same permeability models. The only difference is that an effective particle size diameter has to be used for the poly-disperse samples. A filtration experiment is used to form a particle layer, filter cake, consisting of particles of different sizes. Both inflow and outflow particle size distribution are measured by particle counting method, and from their difference, the particle size distribution in the cake is determined. In a set of experiments, the filtration history is altered by changing (i) filtration medium; (ii) suspension flow rate; and (iii) particle concentration, where in all cases investigated the cake permeability remains constant. In order to predict the permeability of poly-disperse cake from the analytical models, the particle size distribution moments are calculated, and the permeability is found for each moment. Comparing the experimental to the analytical permeability values the effective particle size is found, where the permeability calculated by using the harmonic mean of the particle size distribution reproduces the permeability experimental value best. Finally, in the parametric study, reducing the cake porosity and/or lowering the particle retention shifts effective particle size used in the permeability model toward higher moments of the particle size distribution function.

4:57PM L1.00005 Momentum transfer at the interface between a porous medium and a pure fluid. HOWARD HU, University of Pennsylvania, SONGPENG ZHANG, Tsinghua University — We examine the flow parallel to the interface between a porous medium and a liquid, focusing on the boundary conditions at the interface. When Darcy's law is used to describe the momentum transport in the porous layer, the classic Beavers-Joseph condition relates the shear rate and the slip velocity at the interface with a slip parameter that depends on the structure of the porous surface. When the Brinkman equation is used, the averaged velocity is continuous at the interface, however the fluid shear stress across the interface commonly experiences a jump. This shear stress jump can be expressed in terms of the slip velocity at the interface divided by a length characterized by the square root of the permeability, and a dimensionless stress jump coefficient. In this work, we study the momentum transfer from the clear fluid onto the solid structure at the interface, and propose a stress partition parameter that characterizes the stress transfer from the clear fluid to the fluid (and solid) phase of the porous medium. Simple models are developed to formulate this stress partition parameter for porous media that are brush-like, long fibers, and random, respectively. Our model predictions are compared with numerical and experimental results in the literature.

5:10PM L1.00006 Topological phase transition in 2D porous media flows. NICOLAS WAIS-BORD, Department of Mechanical Engineering, Tufts University, NORBERT STOOP, Department of Mathematics, MIT, VASILY KANTSLER, Department of Physics, University of Warwick, JEFFREY S. GLASS, Department of Mechanical Engineering, Tufts University, JORN DROST, Department of Mathematics, University of Warwick, NICOLAS WAIS-BORD, Department of Mechanical Engineering, Tufts University, JEFFREY S. GLASS, Department of Mechanical Engineering, Tufts University, JORN DROST — Volume-balance models are used by petroleum engineers for simulating multiphase and multicomponent flow phenomena in porous media and the extraction process in oil reservoirs. In these models, mass conservation equations and Darcy's law are supplemented by a balance condition for the pore and fluid volumes. This provides a pressure equation suitable for simulating a compressible flow within a compressible solid matrix. Here we present an alternative interpretation of the volume-balance condition that includes the advective transport within a consolidated porous media. We obtain a modified equation for the time evolution of the pressure field. Numerical tests for phase separation under gravity are presented for multiphase three dimensional flow in heterogeneous porous media.

Since the establishment of Darcy's law, analysis of porous-media flows has focused primarily on linking macroscopic transport properties, such as mean flow rate and dispersion, to the pore statistics of the material matrix. Despite intense efforts to understand the fluid velocity statistics from the porous-media structure, a qualitative and quantitative connection remains elusive. Here, we combine precisely controlled experiments with theory to quantify how geometric disorder in the matrix affects the flow statistics and transport in a quasi-2D microfluidic channel. Experimentally measured velocity fields for a range of different microstructure configurations are found to be well-excellent agreement with large-scale numerical simulations. By successively increasing the matrix disorder, we study the transition from periodic flow structures to transport networks consisting of extended high-velocity channels. Morse-Smale complex analysis of the flow patterns reveals a topological phase transition that is linked to a qualitative change in the physical transport properties. This work demonstrates that topological flow analysis provides a mathematically well-defined, broadly applicable framework for understanding and quantifying fluid transport in complex geometries.
5:49PM L1.00009 Modeling and simulation of multiphase multicomponent multiphysics porous media flows in the context of chemical enhanced oil recovery\(^1\), SOURAV DUTTA, PRABIR DARIPA, Department of Mathematics, Texas A&M University, College Station, TX - 77843, FLUIDS TEAM — One of the most important methods of chemical enhanced oil recovery (EOR) involves the use of complex flooding schemes comprising of various layers of fluids mixed with suitable amounts of polymer or surfactant or both. The fluid flow is characterized by the spontaneous formation of complex viscous fingering patterns which is considered detrimental to oil recovery. Here we numerically study the physics of such EOR processes using a modern, hybrid method based on a combination of a discontinuous, multiscale finite element formulation and the method of characteristics. We investigate the effect of different types of heterogeneity on the fingering mechanism of these complex multiphase flows and determine the impact on oil recovery. We also study the effect of surfactants on the dynamics of the flow via reduction of capillary forces and increase in relative permeabilities.

\(^1\)Supported by the grant NPRP 08-777-1-141 from the Qatar National Research Fund (a member of The Qatar Foundation).

6:02PM L1.00010 On the stabilizing role of species diffusion in chemical enhanced oil recovery\(^1\), PRABIR DARIPA, CRAIG GIN, Texas A&M University — In this talk, the speaker will discuss a problem on the stability analysis related to the effect of species diffusion on stabilization of fingering in a Hele-Shaw model of chemical enhanced oil recovery. The formulation of the problem is motivated by a specific design principle of the immiscible interfaces in the hope that this will lead to significant stabilization of interfacial instabilities, thereby improving oil recovery in the context of porous media flow. Testing the merits of this hypothesis poses some challenges which will be discussed along with some numerical results based on current formulation of this problem. Several open problems in this context will be discussed. This work is currently under progress.

\(^1\)Supported by the grant NPRP 08-777-1-141 from the Qatar National Research Fund (a member of The Qatar Foundation).

6:15PM L1.00011 Evaporation and Settling in an Idealized Porous Medium\(^1\), DANIEL ANDERSON, MATTHEW GERHART, George Mason University — We investigate a mathematical model of a periodic array of solid blocks supported by squeeze films and separated by vertical fluid-filled channels. Evaporation occurs at the open fluid surface at the top of the vertical channels between the blocks and is coupled to the motion of the blocks through mass conservation and pressure, viscous, surface tension and external forces on the blocks. We derive a simplified mathematical model in the form of coupled ordinary differential equations for the thickness of the squeeze film layer, the height of the fluid in the vertical channels and the contact angle at the free surface. We present numerical solutions of this model that address the coupling between block motion and height of the fluid in the channel in an effort to understand the question of channel dry-out versus wetting and fluid resupply via the underlying squeeze film.

\(^1\)This work was supported in part by the US National Science Foundation, DMS-1107848


4:05PM L2.00001 A simple stochastic quadrant model for the transport and deposition of particles in turbulent boundary layers, MICHAEL REEKS, University of Newcastle UK, CHUNYU JIN, Nottingham University UK, JAN POTTS, University of Newcastle UK — We present a simple stochastic quadrant model for calculating the transport and deposition of heavy particles in a fully developed turbulent boundary layer based on the statistics of wall-normal fluid velocity fluctuations obtained from a fully developed channel flow. Individual particles are tracked through the boundary layer via their interactions with a succession of random eddies found in each of the quadrants of the fluid Reynolds shear stress domain in a homogeneous Markov chain process. Deposition rates for a range of heavy particles predicted by the model compare well with benchmark experimental measurements. In addition deposition rates are compared with those obtained continuous random walk (CRW) models including those based on the Langevin equation for the turbulent fluctuations. In addition, various statistics related to the particle near wall behavior are also presented.

4:18PM L2.00002 Neutral and inertial particle acceleration in non isotropic turbulent flows, ARMAANN GYLFASON, Reykjavik University, MICHEL VAN HINSBERG, Eindhoven University of Technology, CHUNG-MIN LEE, California State University - Long Beach, FEDERICO TOSCHI, Eindhoven University of Technology — Turbulent fluctuations influence the dynamics of particulate matter by accelerating the dispersions and mixing of particles. In several natural and industrial flows turbulent fluctuations are strongly coupled to the presence of intense and anisotropic mean flows. The flows that we study here are homogeneous shear and homogeneous strain turbulence. In these flows the dispersion of particles is strongly influenced by gradients in the mean velocity. A comparison of single particle properties, such as acceleration and velocity variances, and time correlations are presented to illustrate the particle dynamics under such conditions.
Neutral and inertial particle acceleration in strained turbulence

4:31PM L2.00003, CHUNG-MIN LEE, California State University - Long Beach, ARMÁNN GÝLFASON, Reykjavik University, PRASAD PERLEKAR, Tata Institute of Fundamental Research, FEDERICO TOSCHI, Eindhoven University of Technology — Turbulence influences the transport and mixing of particles. We study the dynamics of particles in turbulent flows undergoing asymmetrically expanding straining by means of direct numerical simulations. We investigate the accelerations of tracer and inertial particles. We find a good agreement between tracer acceleration variance and the prediction of rapid distortion theory. Furthermore we study how particle acceleration probability density functions depend on the strain rate, the Stokes number, and the Reynolds number. Acceleration variances of inertial particles are discussed in the context of the formal solution of the equation of particle motion, and we show that in strong straining the acceleration variance of particles with small Stokes numbers can exceed that of tracer particles.

4:44PM L2.00004, MICHEL VAN HINSBERG, HERMAN CLERCX, FEDERICO TOSCHI, Eindhoven University of Technology — Setting of particles in a turbulent flow occurs in various industrial and natural phenomena, examples are clouds and waste water treatment. It is well known that turbulence can enhance the settling velocity of particles. Many studies have been done, numerically and experimentally to investigate this behavior for the case of heavy particles, with particle to fluid density ratios above 100. Here we investigate the case of almost neutrally buoyant particles, i.e. density ratios between 1 and 100. In the case of light particles the Maxey-Riley equations cannot be simplified to only the Stokes drag and gravity force as pressure gradient, added mass and Basset history force are important as well. We investigate the influence of these forces on the settling velocity of particles and show that the extra forces can both increase or decrease the settling velocity, depending on the combination of the Stokes number and gravity applied.

Collision statistics of inertial particles suspended in turbulent flows of low dissipation rates

4:57PM L2.00005, SANDIPAN BANERJEE, University of Delaware, ORLANDO AYALA, Old Dominion University, LIAN-PING WANG, University of Delaware — The collision rate of sedimenting droplets in turbulent flows is of great importance in cloud physics. Parameters like the collision efficiency and collision enhancement are key inputs for the calculation of growth in the size of the cloud droplets due to coalescence. In this presentation we report the collision statistics of particles in turbulent flows of low dissipation rates (in the range of 3 cm$^2$/sec$^3$-100 cm$^2$/sec$^3$) for three different particle-pair sizes. Due to the expensive nature of the simulations, it is a common practice to use the linear interpolation to estimate the collision efficiency enhancement (which is defined as the ratio of the collision efficiency in a turbulent flow to the collision efficiency without the flow). In this study, along with the collision statistics, we also examine the accuracy of the linear interpolation approximation by comparing it to simulation data, at arbitrary dissipation rates, obtained from a hybrid direct numerical simulation. Furthermore, we also report particle pair statistics such as the particle relative velocity and the radial distribution function. A study on the computational cost of the simulations is also included.

A Subgrid Particle Averaged Reynolds Stress Equivalent (SPARSE) model for Eulerian-Lagrangian particle-laden-flow simulation

5:10PM L2.00006, SEAN DAVIS, GUSTAAF JACOBS, san diego state university — The direct Eulerian-Lagrangian simulation of turbulent, particle-laden flow through the Navier-Stokes equations combined with the tracing of a large number of particles is computationally expensive for large-scale problems. To reduce computational cost, small scale turbulence is often modeled and groups of physical particles are amalgamated into clouds, whose average location is tracked. Typical Lagrangian models (such as Particle-Source-In-Cell and Cloud-In-Cell models) assume that the average motion of the cloud is governed by the average interphase momentum difference between the carrier and disperse phases, neglecting subscale perturbations. We present a new Lagrangian particle model for the tracing of clouds of particles in particle-laden flows. By expanding the particle drag correction factor to include fluctuating terms and Reynolds averaging the full particle momentum equation, the so-called SPARSE model accounts for the effect of subgrid turbulence and particle perturbations. A priori results demonstrate the efficacy of the SPARSE model in 1D velocity fields and 3D decaying isotropic turbulence computations.

Accelerated Stochastic Vortex Structure Method for Transport of Interacting Particles in Turbulent Flow

5:23PM L2.00007, JEFFREY MARSHALL, KYLIE SALA, FARZAD DIZAJI, The University of Vermont — Turbulent particle transport with RANS or LES methods typically requires an additional model for effect of subgrid-scale eddies on the particles. For non-interacting particles stochastic Lagrangian methods are widely used for this purpose, but these models yield poor results for interacting particles due to lack of spatial correlation in the random forcing terms. Traditional synthetic turbulence methods used for LES initial conditions are often too slow to be useful for particle transport, and they usually lack the vortex structures which are important for generation of particle clustering. In the current work, an accelerated stochastic vortex structure (SVS) method is proposed for generation of synthetic turbulence for transport of interacting particles. The SVS model is shown to yield flow measures, such as energy spectrum and velocity, acceleration and vorticity pdfs, in good agreement with DNS results and with relevant theory. When coupled to a discrete-element method (DEM) code for particle transport, the SVS model is observed to yield very accurate results for particle collision rate and other measures of particle interaction.

Euler-Euler anisotropic Gaussian mesoscale direct numerical simulation of homogeneous and wall-bounded cluster-induce gas-particle turbulence

5:36PM L2.00008, BO KONG, Department of Chemical and Biological Engineering, Iowa State University, Ames, IA, USA, HENG FENG, Department of Chemical and Biological Engineering, Iowa State University, Ames, IA, USA — In our previous works, the exact Reynolds-averaged equations for the particle phase were derived to develop a new multiphase turbulence model with a rigorous conceptual foundation, and detailed Euler-Lagrangian(EL) particle simulations of cluster-induced turbulence (CIT) were performed to aid its development. However, sophisticated filtering techniques have to be used to extract Eulerian particle-phase statistics from the EL simulations, which can be directly provided by Euler-Euler approaches. In this work, a novel Euler-Euler anisotropic Gaussian (AG) approach was used to perform mesoscale DNS of the CIT cases. A three-dimension Hermite Quadrature formulation is used to calculate finite-volume kinetic flux for ten velocity moments. Bhatnagar-Gross-Krook model is applied to account for the inelastic particle collisions. Detailed comparisons with EL simulations demonstrate that the AG particle velocity assumption is valid and this novel method can be used to perform mesoscale DNS for gas-particle flows with high fidelity.

1We gratefully acknowledge support from AFOSR.

1Funded by National Science Foundation project CBET- 1332472.
4:18PM L3.00002 Existence and Smoothness of solution of Navier-Stokes equation on $\mathbb{R}^3$. OGNJEN VUKOVIC, University of Liechtenstein — Navier-Stokes equation has for a long time been considered as one of the greatest unsolved problems in three dimensions. This paper proposes a solution to the aforementioned equation on $\mathbb{R}^3$. It proves the existence and uniqueness of smooth solution. Firstly, the concept of turbulent solution is defined. It is proved that turbulent solutions become strong solutions after some time in Navier-Stokes set of equations. However in order to define the turbulent solution, the decay or blow-up time of solution must be examined. Differential inequality was defined and it was proved that solution of Navier-Stokes equation exists in a finite time although it exhibits blow-up solutions. The equation is introduced that establishes the distance between the strong solutions of Navier-Stokes equation and heat equation. As it is demonstrated, as the time goes to infinity, the solution of heat equation is identical to the solution of Navier-Stokes equation. As the solution of heat equation is defined in the heat-sphere, after its analysis, it is proved that as the time goes to infinity, solution converges to the stationary state. The solution has a finite time and it exists when that implies that it exists and it is periodic. The aforementioned statement proves the existence and smoothness of solution of Navier-Stokes on $\mathbb{R}^3$.1

4:31PM L3.00003 Kaluza’s kinetic theory description of the classical Hall effect in a single component dilute gas within the Chapman-Enskog approximation. A. SANDOVAL-VILLALBAZO, Departamento de Física y Matemáticas, Universidad Iberoamericana, A.L. GARCÍA-PERCIANTE, Departamento de Matemáticas Aplicadas y Sistemas, Universidad Autónoma Metropolitana-Cuajimalpa, A.R. SÁGACETA-MEJÍA, Departamento de Física y Matemáticas, Universidad Iberoamericana — Kinetic theory is used to establish the explicit form of the particle flux associated to the Hall effect for the case of a dilute single component charged gas, using the Chapman-Enskog method and the BGK approximation for the collision Kernel. It is shown that when the system evolves towards mechanical equilibrium, the standard treatment using the concept of external force fails to describe the Hall effect. It is also shown that the use of a five-dimensional curved space-time in the description of the dynamics of the charged particle in the kinetic treatment (Kaluza’s theory) formally solves the problem. The implications of this result are briefly discussed.1

1This project was funded by NSF project NSF-DMS 1318161.

1This study was supported by DOE PSAAP2 Program.
Navier-Stokes equations have more than one complex solution even for low (real) $Re$ enhanced using a generalized Padé approximant technique. This method also predicts complex solution branches, and identifies bifurcation needs to be solved, providing a series representation of the solution for all $Re$. This formalism corresponds to an extension of the result obtained for the case of the direct effect between the particle flux and the electric field within Kaluza’s MHD (A. Sandoval-Villalbazo, A. R. Sagaceta-Mejía, A. L. García-Perciante; Journal of Non-Equilibrium Thermodynamics, 2015, Vol. 40, pp. 93-101.)

The authors acknowledge support from CONACyT through grant CB2011/167563.

Does relativistic kinetic theory predict a viscous analog of the non-equilibrium generalization of Tolman’s law? J.H. Monda-Ramon-Suarez, D. Brun-Battistini, A. Sandoval-Villalbazo, Departamento de Física y Matemáticas, Universidad Iberoamericana, Ciudad de México, A.L. García-Perciante, Departamento de Matemáticas Aplicadas y Sistemas, Universidad Autónoma Metropolitana-Cuajimalpa — When the temperature of a fluid is increased its out of equilibrium behavior is significantly modified. In particular kinetic theory predicts that the heat flux is not solely driven by a temperature gradient but can also be coupled to other thermodynamic vector forces. We explore the nature of heat conduction in a single component charged fluid in special relativity, where the electromagnetic field is introduced as an external force. We obtain an electrothermal effect, similar to the mixture’s cross-effect, which is not present in the non-relativistic simple fluid. The general lines of the corresponding calculation will be shown, emphasizing the importance of reference frame invariance and the origin of the extra heat sources, in particular the difference in fluid’s and molecules’ proper time. The constitutive equation for the heat flux obtained using Chapman-Enskog’s expansion in Marle’s approximation will be analyzed together with the corresponding transport coefficients. The impact of this effect in the overall dynamics of the system here considered will be briefly discussed.

The authors acknowledge support from CONACyT through grant CB2011/167563.

Heat dissipation in relativistic single charged fluids A. L. García-Perciante, Universidad Autónoma Metropolitana - Cuajimalpa, A. Sandoval-Villalbazo, D. Brun-Battistini, Universidad Iberoamericana — When the temperature of a fluid is increased its out of equilibrium behavior is significantly modified. In particular kinetic theory predicts that the heat flux is not solely driven by a temperature gradient but can also be coupled to other thermodynamic vector forces. We explore the nature of heat conduction in a single component charged fluid in special relativity, where the electromagnetic field is introduced as an external force. We obtain an electrothermal effect, similar to the mixture’s cross-effect, which is not present in the non-relativistic simple fluid. The general lines of the corresponding calculation will be shown, emphasizing the importance of reference frame invariance and the origin of the extra heat sources, in particular the difference in fluid’s and molecules’ proper time. The constitutive equation for the heat flux obtained using Chapman-Enskog’s expansion in Marle’s approximation will be analyzed together with the corresponding transport coefficients. The impact of this effect in the overall dynamics of the system here considered will be briefly discussed.

The authors acknowledge support from CONACyT through grant CB2011/167563.

Dynamical density functional theory for arbitrary-shape colloidal fluids including inertia and hydrodynamic interactions M. G. Duran-Olivencia, Imperial College London, UK, Ben Goddard, University of Edinburgh, UK, Serafim Kalliadasis, Imperial College London, UK — Over the last few decades the classical density-functional theory (DFT) and its dynamic extensions (DDFTs) have become a remarkably powerful tool in the study of colloidal fluids. Recently there has been extensive research to generalise all previous DDFTs finally yielding a general DDFT equation (for spherical particles) which takes into account both inertia and hydrodynamic interactions (HI) which strongly influence non-equilibrium properties. The present work will be devoted to a further generalisation of such a framework to systems of anisotropic particles. To this end, the kinetic equation for the Brownian particle distribution function is derived starting from the Liouville equation and making use of Zwanzig’s projection-operator techniques. By averaging over all but one particle, a DDFT equation is finally obtained with some similarities to that for the kinetic equation for the Brownian particle distribution function, it is possible to obtain the well-known expressions that relate the heat flux with the electric field in a dilute gas, without resorting to the steady state approximation. This formalism corresponds to an extension of the result obtained for the case of the direct effect between the particle flux and the electric field within Kaluza’s MHD (A. Sandoval-Villalbazo, A. R. Sagaceta-Mejía, A. L. García-Perciante; Journal of Non-Equilibrium Thermodynamics, 2015, Vol. 40, pp. 93-101.)

The authors acknowledge support from CONACyT through grant CB2011/167563.

Establishment of the thermoelectric effect in Kaluza’s MHD through the kinetic theory A.R. Sagaceta-Mejía, Departamento de Física y Matemáticas, Universidad Iberoamericana, A.L. García-Perciante, Departamento de Matemáticas Aplicadas y Sistemas, Universidad Autónoma Metropolitana-Cuajimalpa, A. Sandoval-Villalbazo, Departamento de Física y Matemáticas, Universidad Iberoamericana — The study of the behavior of charged gases in curved space-times is an active research area in which cross effects, such as thermoelectricity, have not been studied in depth. In our kinetic description of transport through the electric charge is introduced into the fifth component of the particle velocity, following the idea first proposed by Kaluza in 1919. Using Chapman-Enskog’s method, the first order in the gradients correction to the gas distribution function is established, noticing that some of the thermodynamic forces present in the system are associated with the space-time curvature. It is shown that with this distribution function, it is possible to obtain the well-known expressions that relate the heat flux with the electric field in a dilute gas, without resorting to the steady state approximation. This formalism corresponds to an extension of the result obtained for the case of the direct effect between the particle flux and the electric field within Kaluza’s MHD (A. Sandoval-Villalbazo, A. R. Sagaceta-Mejía, A. L. García-Perciante; Journal of Non-Equilibrium Thermodynamics, 2015, Vol. 40, pp. 93-101.)

The authors acknowledge support from CONACyT through grant CB2011/167563.

Solutions to the Navier-Stokes Equation in the complex plane. Jonathan Mestel, Florencia Boshier, Imperial College London — A Stokes series is a theoretically attractive approach to solving the Navier-Stokes equations. Essentially the solution is expressed as a power series in the Reynolds number, $Re$. At each order, a linear problem needs to be solved, providing a series representation of the solution for all $Re$. This method was pioneered by Van Dyke in the 1970s. However, typically this series has a finite radius of convergence, and the solution has singularities at complex values of $Re$. The behaviour of the series can be enhanced using a generalised Padé approximant technique. This method also predicts complex solution branches, and identifies bifurcation points to multiple solutions. Solutions branches for complex $Re$ can be followed back onto the real $Re$-axis. It is shown that in general the Navier-Stokes equations have more than one complex solution even for low (real) $Re$. The intricate structure of complex solutions is followed in detail for Dean flows, and new branches are presented.

Experimental Confirmation of a Causal, Covariant, Relativistic Theory of Dissipative Fluid Flow. Dillon Scofield, Dept. Physics, Oklahoma State University, Pablo Huq, College of Earth, Ocean & Environment, University of Delaware — Using newtonian viscous dissipation stress in covariant, relativistic fluid flow theories leads to a violation of the second law of thermodynamics and to acausality of their predictions. E.g., the Landau & Lifshitz theory, a Lorentz covariant formulation, suffers from these defects. These problems effectively limit such theories to time-independent flow regimes. Thus, these theories are of little fundamental interest to astrophysical, geophysical, or thermonuclear flow modeling. We discuss experimental confirmation of the new geometrodynamical theory of fluids solving these problems (GTF, Fluid Dynamics Research, 46, 055513,055514 (2014), Submitted 2015). This theory is derived from recent results of geometrodynamics showing current conservation implies gauge field creation; the vortex field lemma (Phys. Lett. A 374 3476–82 (2010)).
Evangelista Torricelli claimed that the outflow velocity from a vessel is equal to the terminal speed of a body falling freely from the filling level.

Most mathematics and engineering textbooks describe the process of "subtracting off" the steady state of a linear parabolic partial differential equation as a technique for obtaining a boundary-value problem with homogeneous boundary conditions that can be solved by separation of variables (i.e., eigenfunction expansions). While this method produces the correct solution for the start-up of the flow of, e.g., a Newtonian fluid between parallel plates, it can lead to erroneous solutions to the corresponding problem for a class of non-Newtonian fluids. We show that the reason for this is the non-rigorous enforcement of the start-up condition in the textbook approach, which leads to a violation of the principle of causality. Nevertheless, these boundary-value problems can be solved correctly using eigenfunction expansions, and we present the formulation that makes this possible (in essence, an application of Duhamel's principle). The solutions obtained by this new approach are shown to agree identically with those obtained by using the Laplace transform in time only, a technique that enforces the proper start-up condition implicitly (hence, the same error cannot be committed).

Program.

6:15PM L3.00011 Description of the non-equilibrium extension of Tolman’s law in terms of kinetic theory: suppression of the acceleration term and the use of the geodesic in the treatment of Boltzmann’s equation.

6:28PM L3.00012 The influence of inertia on the efflux velocity: From Daniel Bernoulli to a contemporary theory.

Torricelli’s law is obtained, when inertia is neglected and the cross section of the opening is small compared to the vessel’s cross section. To efflux would start with the highest velocity directly from the initiation of motion which contradicts the inertia principle. In 1738 Daniel Bernoulli derived a much more sophisticated and instantaneous flow theory basing on the conservation of potential and kinetic energy. As a special case Torricelli’s law is obtained, when inertia is neglected and the cross section of the opening is small compared to the vessel’s cross section. To the Authors knowledge, this theory was never applied or even mentioned in text books although it is superior to the Torricelli theory in many aspects. In this paper Bernoulli’s forgotten theory will be presented. Deriving this theory using the state of the arts hydrodynamics results in a new formula \( v = \sqrt{2gh} \). Although this formula contradicts Torricelli’s principle, it is confirmed by all kind of experiments stating that a discharge coefficient of about \( \beta = 0.7 \) is needed in Torricelli’s formula \( v = \beta \sqrt{2gh} \).

Monday, November 23, 2015 4:05PM - 6:41PM –
Session L4 Stratified Boundary Layers

4:05PM L4.00001 Spots and stripes: Isolating the building blocks of intermittent stratified turbulence in plane Couette flow.

4:18PM L4.00002 Coupling of Interfacial Mixing Events in Stratified Taylor–Couette Flow.

The authors acknowledge support from CONCyT through grant CB2011/167563.

1Supported, in part, by NSF Grant DMS-1104047 and the U.S. DOE (Contract No. DE-AC52-06NA25396) through the LANL/LDRD Program.

1Research supported by EPSRC Programme Grant EP/K034529/1 entitled ‘Mathematical Underpinnings of Stratified Turbulence’.
4:31PM L4.00003 Instability onset of the boundary layer on a rotating cylinder in a stratified fluid

JAN-BERT FLOR, LEGI, LIONEL HIRSCHBERG, BART OOSTENRIJK, GERT-Jan VAN HEIJST, TUE Eindhoven, MEIGE TEAM — We consider the instability of the laminar shear layer on a circular cylinder that is impulsively set into rotation about its vertical axis with angular speed $\Omega$. The outer wall of this large gap Taylor-Couette flow is at a radial distance of about 10 times the inner cylinder radius, and the gap is either filled with a homogeneous or linearly stratified fluid. In a homogeneous fluid, the thickness of the boundary layer on the cylinder, $d$, grows until it becomes centrifugally unstable with a wavelength that is determined by the boundary layer thickness $d$. In a linearly stratified fluid with stratification $N$, the flow instability is set by the Froude number $F = \frac{U}{\sqrt{g d} \sqrt{N}}$. For $F \gg 1$ the onset of the centrifugal instability is well predicted by the Taylor-Görtler number and theory for homogeneous fluids. When $F \ll 1$ the onset of a relatively higher Reynolds number, and bifurcates from a vortex regime to a wave regime with a pure inertial wave in the boundary layer. The mechanism of instability is determined by parametric resonance and the generation of waves with subharmonic frequencies typical for Parametric Subharmonic Instability. The results are discussed in view of former results on stratified TC flow.

1Supported by LabEx Oseg@2020 (Investissements davenir ANR10LABX56)

4:44PM L4.00004 DNS of stably stratified Ekman flow with surface cooling

S. M. IMAN GOHARI, PhD Student, SUTANU SARKAR, Professor — Direct numerical simulations of stably stratified Ekman flow are performed to study turbulence in an atmospheric boundary layer under surface cooling. Stability, classified by the normalized Monin-Obukhov (MO) length scale, is varied by imposing a range of cooling fluxes at the surface to mimic ground radiative cooling. The subsequent flow stability, measured by the MO length scale and bulk Richardson number, changes significantly as the flow evolves. We find considerable qualitative differences when a neutrally stratified Ekman flow is exposed to a constant surface cooling rather than a constant temperature, i.e. changes in the veering angle, super-geostrophic velocity, surface shear velocity and the boundary layer height. Under strongly stable condition, the transient evolution shows the presence of intermittent turbulent patches. These patches contain small-scale, inclined hairpin structures that are organized into near-surface streaks. A low-level jet forms at steady state and the high-shear region between the surface and the low level jet is found to play a vital role in promoting turbulence. Our simplified setup is sufficient to observe turbulence collapse, intermittency and the low-level jet formation, indicating the applicability of this model to atmospheric problems.

4:57PM L4.00005 A Multilevel Kinematic Simulation for the Stratified Surface Layer

ADITYA GHATE, SANJIVA LELE, Stanford University — For problems involving a wide range of spatially disparate scales, Kinematic Simulations (KS) offer a low-cost alternative to LES. This is especially true when the phenomena of interest (Ex. Wind turbine fatigue, pollution dispersion, etc.) are those that are primarily affected by the statistical properties of turbulence. In the proposed KS, isotropic turbulence is first "rapidly distorted" using an effective mean shear and density gradient (RDT). The temporal advancement of the stochastic fields can then be done using two different models. The first model idealizes the inter-scale interactions as exclusively those due to "sweeping" of smaller eddies by the larger eddies. In the second formulation, the inter-scale "straining" is accounted for using an RDT-like formulation wherein the Gabor transform is used to explicitly enforce separation of scales between successive resolution levels. Both models produce non-Gaussian turbulent fields for the velocity and temperature fluctuations. The KS will be appraised by comparison of cross spectra, space-time correlations, higher order statistics and other attributes of near wall turbulence using higher fidelity results obtained from LES.

1This research is supported as an Exploratory Research Project by the TomKat Center for Sustainable Energy at Stanford University

5:10PM L4.00006 High Reynolds number effects on a localized stratified turbulent flow

QI ZHOU, University of Cambridge, PETER DIAMESSIS, Cornell University — We report large-eddy simulations (LES) of the turbulent flow behind a sphere of diameter $D$ translating at speed $U$ in a linearly stratified Boussinesq fluid with buoyancy frequency $N$. These simulations are performed using a spectral-multidomain-incomparable Navier-Stokes solver, at Reynolds numbers $Re = U D / \nu \in \{5 \times 10^4, 10^5, 4 \times 10^5\}$ and Froude numbers $Fr \equiv 2U / (ND) \in \{4, 16, 64\}$. An increasingly richer turbulent fine-structure is observed within the larger-scale quasi-horizontal vortices at later times. Turbulent transport of momentum is examined during the non-equilibrium (NEQ) regime of the turbulent life cycle, with an emphasis on the vertical transport that occurs after the establishment of local buoyancy control. The turbulent viscosities in both horizontal and vertical directions are estimated through the LES data; possible parameterization of the vertical turbulent viscosity with the buoyancy Reynolds number $Re_B = \epsilon / (\nu N^2)$ (or its easy-to-obtain surrogates) is discussed. The dynamical role of the buoyancy Reynolds number in choosing the turbulent length scale is also investigated.

1ONR grant N00014-13-1-0665 (managed by Dr. R. Joslin); HPCMP Frontier Project FP-CFD-FY14-007 (P.I.: Dr. S. de Bruyn Kops)

5:23PM L4.00007 Internal length scales in rotating and stratified Boussinesq flows

SUSAN KURIEN, New Mexico Consortium and Los Alamos National Laboratory, X.M. ZHAI, P.K. YEUNG, Georgia Tech — We study the characteristic length scales of the propagating (wave) and non-propagating (vortical) modes, in a suite of simulations of forced, rotating, stably stratified Boussinesq flows. We employ a pseudo-spectral code, periodic boundary conditions and grid resolutions ranging from $512^2$ to $2048^3$ on Blue Gene/Q (Argonne) under DOE’s INCITE program. The relative strength of rotation to stratification frequencies is given by the Burger number $Bu$. Integral length scales in the vertical and horizontal directions are chosen as the characteristic scales and their ratio defines an internal aspect ratio. Nominally quasi-geostrophic (QG) scaling of $Bu^{1/2}$ is recovered for the vortical scale aspect ratio in the stratification-dominated regime $Bu \gg 4$. Much weaker scaling in $Bu$ emerges for the vortical mode in the rotation-dominated regime $Bu \ll 4$. The aspect ratio of the wave modes in both regimes are only weakly dependent on $Bu$. Turbulence affects the wave modes in the strongly rotating case by increasing the aspect ratio systematically but has no impact on the weak $Bu$ dependence. It appears that for unit aspect ratio domains, QG scaling of the vortical mode holds only for stratification-dominated flows irrespective of the strength of rotation.

5:36PM L4.00008 Laboratory experiments of an atmospheric/oceanic turbulence

ADRIEN THACKER, OLIVIER EIFF, Institut de Mécanique des Fluides de Toulouse, WAVES, TURBULENCE, ENVIRONMENT TEAM — Atmospheric or oceanic turbulence is strongly influenced by the effects of stratification leading to the emergence of quasi-horizontal layers often described as “pancake” structures. The mechanisms of this layering and the selection of the vertical length scale of pancake structures is discussed for one decade whereas it is of a major importance to elucidate the energetic cascade that leads to viscous dissipation. In this present work, we analyze a new series of decaying grid turbulence experiments under the effects of stratification aiming to identify and observe the strongly stratified turbulence regime. The experiments have been performed in a large water towing tank with salt stratification and measurements have been carried out using a scanning correlation imaging velocimetry technique providing instantaneous 3DSC velocity fields along the decaying turbulence. Self similar power laws of the decaying grid turbulence have been assessed and allow the definition of empirical critical time giving transitions to the strongly stratified turbulence regime. A first experimental evidence of overturning process between layers of pancake vortices has been obtained through vorticity fields observation. This observation support the existence of a downscale energy cascade.
5:49PM L4.00009 Characteristics of the residual stress tensor as a function of length scale in simulations of stably stratified turbulence. FELIPE AUGUSTO DE BRAGANCA ALVES, STEPHEN DE BRUYN KOPS, University of Massachusetts Amherst — A priori analysis of the relationships between the deviatoric residual stress tensor \( \tau \) and kinematic tensors is made for stably stratified Boussinesq turbulence. Two data sets from direct numerical simulation are used for the analyses: the decaying Taylor-Green simulations of Riley and de Bruyn Kops(2003), and the forced homogeneous stratified turbulence simulations of Almalkie and de Bruyn Kops(2012) resolved on up to 8192 \( \times \) 8192 \( \times \) 4096 grid points. The data sets are filtered using a Gaussian kernel with filter widths up to the buoyancy scale. Through tensor decomposition theorems described in Thompson et al.(2010) the relationship between the strain rate tensor and the residual stress is quantified for each filter width and case. This is also done for the tensor formed by the Lie product between the strain rate and rate of rotation tensors. The role of each tensor, seen as a part of the residual stress tensor, is analyzed, in particular with respect to filtered kinetic energy budget equation.

The authors acknowledge the support from CAPES grant BEX 13649/13-2, DoD HPCMP Frontier Project FPCFD-FY14-007 and ONR grant N00014-15-1-2248

6:02PM L4.00010 Energy transfer in stably stratified turbulence. YOSHIFUMI KINUMA, Nagoya University, JACKSON HERRING, NCAR — Energy transfer in forced stably stratified turbulence is investigated using pseudo-spectral DNS of the Navier-Stokes equations under the Boussinesq approximation with 1024\(^3\) grid points. Making use of the Craya-Herring decomposition, the velocity field is decomposed into vortex (\( \Phi_1 \)) and wave (\( \Phi_2 \)) modes. To understand the anisotropy of stably stratified turbulence, the energy fluxes in terms of the spherical, the horizontal and the vertical wave numbers, are investigated for the total kinetic, \( \Phi_1 \), \( \Phi_2 \) energies, respectively. Among the three fluxes, the spherical and the horizontal look similar for strong stratification, and \( k^5/3 \) for the spherical and \( k^3/5 \) for the horizontal cases. In contrast to these, the vertical energy fluxes show completely different features. We have observed the saturation spectrum \( E(k_z) \sim C N^2 k_z^{-3} \) for strong stratification as before\(^1\), but the mechanism to produce this spectrum seems different from the Kolmogorov picture.

1Support from the US National Science Foundation and the Gulf of Mexico Research Initiative is gratefully acknowledged

6:15PM L4.00011 LES of oscillating boundary layers under neutrally stratified and unstably stratified conditions. MARIO JUHA, JIE ZHANG, ANDRES TEJADA-MARTINEZ, University of South Florida — Results are presented from LES of open channel flow driven by an oscillating pressure gradient with zero surface shear stress. The flow is representative of an oscillating tidal boundary layer. Under neutrally stratified conditions, during certain phases of the oscillating pressure gradient, the flow develops large scale secondary structures, characterized by full-depth regions (or limbs) of negative and positive wall-normal velocity fluctuations. These structures are similar but less coherent than the classical Couette cells found in Couette flow driven by parallel no-slip plates moving in opposite direction. Unstable stratification will be imposed by a constant cooling flux at the surface and an adiabatic bottom wall. The effect of the surface cooling on the large scale secondary structures and the overall turbulence statistics will be investigated. The analysis will be performed in terms of the Rayleigh number (Ra), representative of the importance of surface buoyancy relative to shear, and the Rossby number (Ro), representative of the importance of the turbulence throughout the water column. For example, in unstratified conditions, if Ro is relatively small, turbulence stress is expected to be important only near the bottom of the boundary layer.

6:28PM L4.00012 Turbulent flows over a modeled steep topography in a thermally-stratified boundary layer. WEI ZHANG, Mechanical Engineering, Cleveland State University, COREY MARKFORT, Civil and Environmental Engineering, University of Iowa, FERNANDO PORTE-AGEL, Ecole polytechnique fédérale de Lausanne, Switzerland — Turbulent flows with features of separation and reattachment, induced by topography of steep slopes, have been very challenging to predict using numerical models. The geometry of the topography, surface roughness and temperature along with the inflow characteristics (velocity, turbulence level, and atmospheric thermal stability) play important roles in determining onset of separation, reattachment location and momentum and heat flux distribution. To address the coupled effects of steep slope and thermal stability on turbulent flows over topography, series of wind-tunnel measurements of velocity fields, temperature and heat fluxes will be presented. Results will demonstrate the turbulent flow properties, including the size of the separation bubble, separated shear layers and the boundary layer recovery in different cases. Focus will be placed on correlation of momentum and heat flux distribution in the wake and turbulent kinetic energy transport.

Monday, November 23, 2015 4:05PM - 6:41PM — Session L5 Jets III: General 104 - Sofia Larsson, Lulea University of Technology, Sweden

4:05PM L5.00001 Jet mixing in a down-scaled model of a rotary kiln. SOFIA LARSSON, SIMON JOHANSSON, Luleå University of Technology, Sweden — Rotary kilns are large, cylindrical, rotating ovens with a burner in one end that are used in various industrial processes to heat up materials to high temperatures. Kiln burners are characterized by long diffusion flames where the combustion process is largely controlled by turbulent diffusion mixing between the burner fuel jet and the surrounding combustion air. The combustion air flow patterns have a significant effect on the mixing and hence the combustion efficiency and flame shape, motivating a systematic study of the kiln aerodynamics and the mixing characteristics. In this work, a downscaled, isothermal model of a rotary kiln is investigated experimentally through simultaneous particle image velocimetry and planar laser-induced fluorescence measurements. The kiln is modeled as a cylinder with three inlets in one end; two semicircular-shaped inlets for which is called the secondary fluid divided by a wall in between, called the back plate, where the burner nozzle is located. Three momentum flux ratios of the secondary fluid are investigated, and the interaction with the burner jet is scrutinized. It is found that the burner jet characteristics, its mixing with the secondary fluid and the resulting flow field surrounding the jet are dependent on the momentum flux ratio.

4:18PM L5.00002 Stretched Inertial Jets. ELISABETH CHABACHE, ARNAUD ANTKOWIAK, THOMAS SEON, Institut Jean Le Rond d’Alembert, UPMC & CNRS, Paris, EMMANUEL VILLERMAUX, IRPHE, Aix Marseille Université — Liquid jets often arise as short-lived bursting liquid flows. Cavitation or impact-driven jets, bursting champagne bubbles, shaped-charge jets, ballistospores or drop-on-demand inkjet printing are a few examples where liquid jets are suddenly released. The trademark of all these discharge jets is the property of being stretched, due to the quenching injection.the present theoretical and experimental investigation, the structure of the jet flow field will be unraveled experimentally for a few emblematic occurrences of discharge jets. Though the injection markedly depends on each flow configuration, the jet velocity field will be shown to be systematically and rapidly attracted to the universal stretching flow \( z/t \). The emergence of this inertial attractor actually only relies on simple kinematic ingredients, and as such is fairly generic. The universality of the jet velocity structure will be discussed.
4:31PM L5.00003 On the local acceleration and flow trajectory of jet flows from circular and semi-circular pipes via 3D particle tracking velocimetry, JIN-TAE KIM, Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, USA, ALEX LIBERZON, Mechanical Engineering, Tel Aviv University, Israel, LEONARDO P. CHAMORRO, Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, USA — The distinctive differences between two jet flows that share the same hydraulic diameter $d_h = 0.01$ m and $Re \approx 6000$, but different (nozzle) shape are explored via 3D Particle Tracking Velocimetry using OpenPTV (http://www.openptv.net). The two jets are formed from circular and semi-circular pipes and released in a quiescent water tank of 40 $d_h$ height, 40 $d_h$ wide, and 200 $d_h$ long. The recirculating system is seeded with 100 $\mu$m particles, where flow measurements are performed in the intermediate flow field (14.5 $<<$Re$<<$18.5) at 550Hz for a total of $\approx$ 30,000 frames. Analysis is focused on the spatial distribution of the local flow acceleration and curvature of the Lagrangian trajectories. The velocity and acceleration of particles are estimated by low-pass filtering their position with a moving cubic spline fitting, while the curvature is obtained from the Frenet-Serret equations. Probability density functions (p.d.f.) of these quantities are obtained at various sub-volumes containing a given streamwise velocity range, and compared between the two cases to evaluate the memory effects in the intermediate flow field.

4:44PM L5.00004 Experimental study of global oscillations in low density rectangular jets, RATHWIK N, VINOTH B. R, Indian Institute of Space Science and Technology, India — The global oscillations of helium jets from rectangular nozzles are studied experimentally. The objectives of the work are: (i) study the effect of nozzle aspect ratio on global oscillations, (ii) study the effect of velocity profile which has continuously varying $Re_d$ and $\frac{c}{d}$ at the nozzle exit, on global oscillations, and (iii) control of global oscillations. The effect of aspect ratio (AR) of rectangular nozzle on the global oscillations is studied using three nozzles of AR = 06, 12 and 20. The behavior of nozzles with AR = 12 and AR = 20 are found to be similar. Nozzles which have a varying height at the exit plane, are used to generate exit velocity profiles with continuously varying $Re_d$ and $\frac{c}{d}$ to study the role of $Re_d$ and $\frac{c}{d}$ on global oscillations of rectangular jets. It is found that the jets with the above velocity profile oscillates with single frequency. Circular wires are used to control the global oscillation of rectangular jets. The most sensitive region for rectangular jets with respect to suppression of global oscillations is found to be near the nozzle exit. But, the placement of wire at nozzle exit does not suppress the global oscillations.

4:57PM L5.00005 High fidelity simulation of liquid jet in an excited crossflow, XIAOYI LI, Staff Research Engineer, United Technologies Research Center, MARIOS SOTERIOU, Fellow, United Technologies Research Center — Dynamic excitation of liquid jet in crossflow by externally oscillating the air stream has attracted much attention mainly due to its relevance to thermoacoustic instability mitigation. In this work, first principle high fidelity simulations of liquid jet atomization in an excited gaseous crossflow are performed using a dual-fluid Combined Level-Set and Volume Of Fluid (CLSVOF) interface capturing approach enhanced by a ghost fluid sharp interface treatment. Adaptive mesh refinement and Lagrangian algorithm for the smallest, spherical droplets are used to reduce the simulation cost. The simulations are validated against recently published experimental results. Mean features such as average jet penetration, volume flux and droplet size distribution and dynamic evolution of these quantities are studied. Proper Orthogonal Decomposition (POD) analyses of liquid surface features as well as detailed visualizations of the gaseous flow in the vicinity of the liquid column are performed. Detailed discussions of the impact of excitation on the physics of atomization is presented and the mechanisms by which excitation modifies the spray are identified.

5:10PM L5.00006 Strain Rates and Scalar Dissipation Rates in Gaseous Transverse Jets, TAKESHI SHOJI, LEVON GEVORKYAN, ANDREA BESNARD, ANN KARAGOZIAN, University of California, Los Angeles — This experimental study quantifies local strain rates and scalar dissipation rates for the non-reactive gaseous jet in crossflow (JICF) using simultaneous acetone planar laser-induced fluorescence (PLIF) imaging and stereo particle image velocimetry (PIV). Flush nozzle and flush pipe injectors are used to create jets consisting of mixtures of He and N2, with varying exit velocity profiles, jet-to-crossflow momentum flux ratios $J$, and density ratios $S$. Strain rates in the vicinity of windward and lee-side jet shear layers are quantified based both on scalar dissipation rates extracted from PLIF measurements within locally 1D layer-like structures and on vector fields extracted from PIV measurements. Strain rates from the simulations measurements are in very good agreement with one another in windward and lee sides, and are also consistent with flame ignition locations in comparable reactive JICF experiments. Quantitative differences in strain fields are most pronounced at lower $J$ values, corresponding to absolutely unstable shear layers and high local strain fields, although these differences are affected by the PLIF spatial resolution for a range of flow conditions. Extraction of dominant mode structures via POD will also be presented.

1Supported by NSF grant CBET-1437014 & AFOSR grant FA9550-15-1-0261 (A004376801)

5:23PM L5.00007 Rapid mixing of viscous liquids by electrical coiling, TIANTIAN KONG, JINGMEI LI, ZHOU LIU, LIQUW WANG, HO CHEUNG SHUM, University of Hong Kong — We study the coiling of viscous liquid jets under an axial electric field. As a viscous jet accelerated by the electric field encounters a solid substrate, it is forced to decelerate, leading a compressive force that sets the jet to coil. We show that the coiling characteristics are significantly influenced by the applied electric force. Based on a balance between the electric and viscous torque, we deduce a scaling law to predict the coiling frequency from the relevant physical parameters, including the viscosity, dielectric constant, volumetric flow rate of the liquid and the applied electric field intensity. Moreover, we exploit this balance between the electric and viscous torque, we deduce a scaling law to predict the coiling frequency from the relevant physical parameters, force that sets the jet to coil. Linear stability analysis is performed to study the stability characteristics of the jet.

5:36PM L5.00008 ABSTRACT MOVED TO KP2.036 —

5:49PM L5.00009 Numerical and experimental study of the dynamics of a superheated jet, AVICK SINHA, SHIVASUBRAMANIAN GOPALAKRISHNAN, SRIDHAR BALASUBRAMANIAN, Department of Mechanical Engineering, Indian Institute of Technology Bombay, India — Flash-boiling is a phenomenon where a liquid experiences low pressures in a system resulting in it getting superheated. The sudden drop in pressures results in accelerated expansion and violent vapour formation. Understanding the physics behind the jet disintegration and flash-boiling phenomenon is still an open problem, with applications in automotive and aerospace combustors. The behaviour of a flash-boiling jet is highly dependent on the input parameters, inlet temperature and pressure. In the present study, the external (outside nozzle) and the internal (inside nozzle) flow characteristics of the two-phase flow has been studied numerically and experimentally. The phase change from liquid to vapour takes place over a finite period of time, modeled sing Homogeneous Relaxation Model (HRM). In order to validate the numerical results, controlled experiments were performed. Optical diagnostic techniques such as Particle Image Velocimetry (PIV) and Shadowgraphy were used to study the flow characteristics. Spray angle, penetration depth, droplet spectra were obtained which provides a better understanding of the break-up mechanism. Linear stability analysis is performed to study the stability characteristics of the jet.
6:02PM L5.00010 Effects of Annular and Rectangular Confinement on the Hydrodynamics of Reacting, Swirling Jets, BENJAMIN EMERSON, TIM LIEUJVEN, Georgia Tech — In gas turbine combustors, flames are stabilized in the shear layers of swirling jets. In such devices, the flame's dynamics and its unsteady heat release are strongly governed by the fluid dynamics of the swirling jet flow. This unsteady heat release can couple with an acoustic mode of the combustor to cause a troublesome self-excited oscillation known as combustion instability. This coupling often occurs through the fluid dynamics, where the flame is dynamically wrinkled by acoustically excited vortical structures. This study uses linear stability analysis to study the effects of confinement on the fluid dynamics of reacting, swirling jets. Previous studies have explored confinement effects of an outer cylindrical wall. This study investigates other types of confinement. The analysis compares the classical arrangement, with flow through an outer cylindrical wall, to two other arrangements: flows through annular or rectangular confinements. The analysis shows that these confinement changes can have significant impacts on the instability growth rates, frequencies, and mode shapes. For example, changing a cylindrical confinement to a rectangular confinement tends to alter the hydrodynamic mode shape by straightening the nodal lines in the hydrodynamic velocity field.

6:15PM L5.00011 Experimental observations of a complex, supersonic nozzle concept, ANDREW MAGSTADT, MATTHEW BERRY, MARK GLAUSER, Syracuse University, CORY STACK, DATTA GAITONDE, The Ohio State University, MARK GLAUSER, Syracuse University, SYRACUSE UNIVERSITY TEAM, SPECTRAL ENERGIES, LLC, BARRY KIEL, Air Force Research Laboratory, WPAFB, SKYTOP TURBULENCE LABS, SYRACUSE UNIVERSITY TEAM, THE Ohio STATE UNIVERSITY TEAM — Advanced supersonic nozzle concepts are currently under investigation, utilizing multiple bypass streams and airframe integration to bolster performance and efficiency. This work focuses on the parametric study of a supersonic, multi-stream jet with a single plane of symmetry, additional shear layer (referred to as a wall jet) and an aft deck representative of airframe integration. Operating near a Reynolds number of $3 \times 10^6$, the nozzle architecture creates an intricate flow field comprised of high turbulence levels, shocks, shear & boundary layers, and powerful corner vortices. Current data suggest that the wall jet, which is an order of magnitude less energetic than the core, has significant control authority over the acoustic power through some non-linear process. As sound is a direct product of turbulence, experimental and analytical efforts further explore this interesting phenomenon associated with the turbulent flow.

6:28PM L5.00012 Advanced Supersonic Nozzle Concepts: Experimental Flow Visualization Results Paired With LES, MATTHEW BERRY, ANDREW MAGSTADT, Syracuse University, CORY STACK, DATTA GAITONDE, The Ohio State University, MARK GLAUSER, Syracuse University, SYRACUSE UNIVERSITY TEAM, THE Ohio STATE UNIVERSITY TEAM — Advanced supersonic nozzle concepts are currently under investigation, utilizing multiple bypass streams and airframe integration to bolster performance and efficiency. This work focuses on the parametric study of a supersonic, multi-stream jet with a single plane of symmetry, rectangular nozzle, displays very complex and unique flow characteristics. Flow visualization techniques in the form of PIV and schlieren capture flow features at various deck lengths and Mach numbers. LES is compared to the experimental results to both validate the computational model and identify limitations of the simulation. By comparing experimental results to LES, this study will help create a foundation of knowledge for advanced nozzle designs in future aircraft.

Monday, November 23, 2015 4:05PM - 6:28PM – Session L6 CFD: Applications I

4:05PM L6.00001 How does network design constrain optimal operation of intermittent water supply?, ANNA LIEB, UC Berkeley, JON WILKENING, UC Berkeley Mathematics, CHRIS RYCROFT, Harvard School of Engineering and Applied Sciences — Urban water distribution systems do not always supply water continuously or reliably. As pipes fill and empty, pressure transients may contribute to degraded infrastructure and poor water quality. To help understand and manage this undesirable side effect of intermittent water supply—a phenomenon affecting hundreds of millions of people in cities around the world—we study the relative contributions of fixed versus dynamic properties of the network. Using a dynamical model of unsteady transition pipe flow, we study how different elements of network design, such as network geometry, pipe material, and pipe slope, contribute to undesirable pressure transients. Using an optimization framework, we then investigate to what extent network operation decisions such as supply timing and inflow rate may mitigate these effects. We characterize some aspects of network design that make them more or less amenable to operational optimization.

4:18PM L6.00002 Defining boundary conditions for RANS predictions of urban flows using mesoscale simulations, CLARA GARCIA SANCHEZ, von Karman Institute for Fluid Dynamics, CATHERINE GORLE, Columbia University, JEROEN VAN BEECK, von Karman Institute for Fluid Dynamics — Pollutant dispersion and wind flows in urban canopies are major concerns for human health and energy, and the complex nature of the flow and transport processes remains a challenge when using Computational Fluid Dynamics (CFD) to predict wind flows. The definition of the inflow boundary condition in Reynolds-Averaged Navier-Stokes simulations (RANS) is one of the uncertainties that will strongly influence the prediction of the flow field, and thus, the dispersion pattern. The goal of the work presented is to define a methodology that improves the level of realism in the inflow condition for RANS simulations by accounting for larger mesoscale effects. The Weather Research and Forecasting model (WRF) is used to forecast mesoscale flow patterns, and two different approaches are used to define inflow conditions for the RANS simulations performed with OpenFOAM: 1) WRF variables such as local velocity magnitude, ABL height and friction velocity are directly interpolated onto the boundaries of the CFD domain; 2) WRF predictions for the geostrophic wind and friction velocity are applied as a forcing boundary condition. Simulations of the Joint Urban 2003 experimental campaign in Oklahoma City have been performed using both approaches and a comparison of the results will be presented.
4:31PM L6.00003 Numerical Study of Wind Shielding Impacts on Water Quality in a Tropical Urban Lake1, HAIYAN MIAO, Institute of High Performance Computing, A*STAR, Singapore, ZIKUN XING, Nanyang Technological University, Singapore, LLOYD CHUA, Deakin University, Australia — In this study, we investigate the impact of wind shielding effect on hydrodynamics and water quality in Marina reservoir, a tropical lake located in downtown Singapore. This kind of urban lakes are usually smaller and shallower comparing with naturally formed ones and therefore, subject to a higher degree of interaction with wind. To establish wind field over the lake surface, Computational Fluid Dynamics (CFD) modeling was conducted to simulate seasonal impacts of two dominant monsoon seasons, in which the prevailing wind patterns interact very differently with urban landscape. The CFD model results were then used as input to a 3D lake hydrodynamics and water quality model to study the impacts to the hydrodynamics and water quality in the lake. By comparing simulations using uniform and spatial variable wind field, this study demonstrates that wind forcing variability in urban reservoirs that arise from shielding effects can have significant impacts on the thermal stratification and mixing, and phytoplankton distribution in both vertical and horizontal directions. There exist significant seasonal differences in wind field, hydrodynamics and water quality between the northeast and southwest monsoon seasons.

1This work is supported by the Singapore National Research Foundation (project 1002-IRIS-09).

4:44PM L6.00004 SimVascular 2.0: an Integrated Open Source Pipeline for Image-Based Cardiovascular Modeling and Simulation, HONGZHI LAN, Stanford University, JAMESON MERKOW, University of California, San Diego, ADAM UPDEGROVE, University of California, Berkeley, DANIELE SCHIAVAZZI, Stanford University, NATHAN WILSON, Open Source Medical Software Corporation, SHAWN SHADDEN, University of California, Berkeley, ALISON MARSDEN, Stanford University — SimVascular (www.simvascular.org) is currently the only fully open source software package that provides a complete pipeline from medical image based modeling to patient specific blood flow simulation and analysis. It was initially released in 2007 and has contributed to numerous advances in fundamental hemodynamics research, surgical planning, and medical device design. However, early versions had several major barriers preventing wider adoption by new users, large-scale application in clinical and research studies, and educational access. In the past years, SimVascular 2.0 has made significant progress by integrating open source alternatives for the expensive commercial libraries previously required for anatomic modeling, mesh generation and the linear solver. In addition, it simplified the across-platform compilation process, improved the graphical user interface and launched a comprehensive documentation website. Many enhancements and new features have been incorporated for the whole pipeline, such as 3-D segmentation, Boolean operation for discrete triangulated surfaces, and multi-scale coupling for closed loop boundary conditions. In this presentation we will briefly overview the modeling/simulation pipeline and advances of the new SimVascular 2.0.

4:57PM L6.00005 Flapping dynamics of an inverted flag in a uniform flow, JAEHA RYU, SUNG GOON PARK, BOYOUNG KIM, HYUNG JIN SUNG, KAIST — Much research in recent years has focused on the flow dynamics of flexible structures in a uniform flow and particularly on the flow dynamics related to energy harvesting systems. An energy harvesting system comprising piezoelectric patches attached to the surface of a flexible structure can convert the energy stored in solid deformations into an electric current that powers a purely resistive output circuit. Recently, an inverted flag which has the freely moving leading edge and the clamped trailing edge was suggested. The inverted flag improved the amount of strain energy that was converted into the flag deformations from the surrounding fluid. In this study, the flapping dynamics of an inverted flag in a uniform flow were simulated using the immersed boundary method. The flapping dynamics of and vertical structures around the inverted flag were examined in terms of the bending rigidity and the Reynolds number. The strain energy of the inverted flag and the proportion of the strain energy of the inverted flag to the kinetic energy of the flow were considered as an indicator of the energy harvesting system efficiency.

5:10PM L6.00006 ABSTRACT WITHDRAWN —

5:23PM L6.00007 Numerical simulation of artificial and natural rough surfaces, ROBIN ANDERSSON, Luleå University of Technology, PATRIK ANDREASSON, Vattenfall Research and Development, Luleå University of Technology, GUNNAR HELLMART LANDERS ANDERSSON, Luleå University of Technology — Flow in hydraulic tunnels is characterized by high velocities, non-uniform and fluctuating distributions of pressure and velocity. Pressure fluctuations may in the long run lead to unwanted effects such as erosion and in extreme cases the complete collapse of a tunnel. Natural rough surfaces, such as in hydropower tunnels, typically have the property of self-similarity. When observing the surfaces from a far, a large scale roughness is visible, but when observed from a very short distance a similar pattern of surface roughness can be observed. One procedure when evaluating flow over rough surfaces is to account for the large scale roughness and replace the small scale roughness with numerical wall functions, the self-similarity is excluded and is replaced by a numerical shear stress. The objective for this work is to gain a deeper understanding the impact that the small scale roughness has on the flow. In this work simulations where performed on a rough surface from a laser scanned hydropower tunnel, the simulation was then compared with an artificially generated rough surface. The reported results include evaluation and comparison of the friction velocity and Reynolds-stresses. Also, a characteristic roughness length scale Ld is evaluated from the logarithmic law of the wall.

5:36PM L6.00008 Numerical modelling of microdroplet self-propelled jumping on micro-textured surface1, S. M. REZA ATTARZADEH, ALI DOLATABADI, Concordia Univ, KYUNG CHUN KIM, Pusan National University — Understanding various stages of single and multiple droplet impact on a super-hydrophobic surface is of interest for many industrial applications such as aerospace industry. In this study, the phenomenon of coalescence induced droplets self-propelled jumping on a micro-textured super-hydrophobic surface is numerically simulated using Volume of Fluid (VOF) method. This model mimics the scenario of coalescing cloud-sized particles over the surface structure of an aircraft. The VOF coupled with a dynamic contact angle model is used to simulate the coalescence of two equal size droplets, that are initially placed very close to each other with their interface overlapping with each other’s which triggers the incipience of their coalescence. The textured surface is modeled as a series of equally spaced squared pillars, with 111° as the intrinsic contact angle all over the solid contact area. It is shown that the radial velocity of coalescing liquid bridge is reverted to upward direction due to the counter action of the surface to the basal area of droplet in contact. The presence of air beneath the droplet inside micropores which aimed at repelling water droplet is also captured in this model. The simulated results are found in good agreement with experimental observations.

1The authors gratefully acknowledge the financial support from Natural Sciences and Engineering Research Council of Canada (NSERC), Consortium de Recherche et d’innovation en Aerospatiale au Quebec (CRIAQ), Bombardier Aerospace, Pratt Whitney Canada.
indoor environments. The instantaneous and mean concentration profiles compared. The results from this study will provide insight into canine detection of vapour in floor of the room and the resulting vapour dispersion has been modelled. Sources with different surface areas have been included and their play a key part in their detection process. A low Reynolds number (Re less than 5000 at the inlet) benchmark test case for indoor airflow that are considerably higher than the time averaged values. As dogs can sample the air at 5 Hz it is possible that these unsteady fluctuations play a key part in their detection process. A low Reynolds number (Re less than 5000 at the inlet) benchmark test case for indoor airflow has been studied using large-eddy simulation computational fluid dynamics. Fixed concentration vapour sources have been included on the floor of the room and the resulting vapour dispersion has been modelled. Sources with different surface areas have been included and their instantaneous and mean concentration profiles compared. The results from this study will provide insight into canine detection of vapour in indoor environments.

6:15PM L6.00011 ABSTRACT WITHDRAWN —

Monday, November 23, 2015 4:05PM - 6:41PM — Session L7 CFD: Immersed Boundary Method 107 - Tim Colonius, Caltech

4:05PM L7.00001 A fast lattice Green’s function method for solving viscous incompressible flows on unbounded domains , SEBASTIAN LISKA, TIM COLONIUS, California Institute of Technology — A novel, parallel, computationally efficient immersed boundary method for solving three-dimensional, viscous, incompressible flows on unbounded domains is presented. The method formally discretizes the incompressible Navier-Stokes equations on an infinite staggered Cartesian grid. Operations are limited to a finite computational domain through a lattice Green’s function technique. This technique obtains solutions to inhomogeneous difference equations through the discrete convolution of source terms with the fundamental solutions of the discrete operators. The differential algebraic equations describing the temporal evolution of the discrete momentum equation, incompressibility constraint, and the Greens’ function are solved by a semi-implicit scheme for the viscous term and a half-explicit Runge-Kutta scheme for the convective term. A nested projection that exploits the mimetic and commutativity properties of the discrete operators is used to efficiently solve the system of equations that arises in each stage of the time integration scheme. Linear complexity, fast computation rate, and parallel scalability are achieved using recently developed fast multipole methods for difference equations. Results for three-dimensional test problems are presented, and the performance and scaling of the present implementation are discussed.

4:18PM L7.00002 Computational Analysis of Flow Field Inside Coral Colony , MD MONIR HOSSAIN, ANNE STAPLES, Virginia Polytechnic Institute and State University — Development of the flow field inside coral colonies is a key issue for understanding coral natural uptake, photosynthesis and wave dissipation capabilities. But most of the computations and experiments conducted earlier, measured the flow outside the coral reef canopies. Experimental studies are also constrained due to the limitation of measurement techniques and limited environmental conditions. Numerical simulations can be an answer to overcome these shortcomings. In this work, a detailed, three-dimensional simulation of flow around a single coral colony was developed to examine the interaction between coral geometry, hydrodynamics. The new computational cost, Immersed Boundary method (IBM) was implemented. The implementation of IBM involves identification of the interface between the solid body and the fluid, establishment of the grid/interface relation and identification of the forcing points on the grid and distribution of the forcing function on the corresponding points. LES was chosen as the framework to capture the turbulent flow field without requiring extensive modeling. The results presented will give insight into internal coral colony flow fields and the interaction between coral and surrounding ocean hydrodynamics.

4:31PM L7.00003 A cut-cell immersed boundary technique for fire dynamics simulation , MARCOS VANELLA, The George Washington University, RANDALL MCDERMOTT, GLENN FORNEY, National Institute of Standards and Technology — Fire simulation around complex geometry is gaining increasing attention in performance based design of fire protection systems, fire-structure interaction and pollutant transport in complex terrains, among others. This presentation will focus on our present effort in implementing the capability of FDS (Fire Dynamics Simulator, https://github.com/firemodels/fds-smv) to represent fire scenarios around complex bodies. Velocities in the vicinity of the bodies are reconstructed using a classical immersed boundary scheme (Fadlun and co-workers, J. Comput. Phys., 161:35-60, 2000). Also, a conservative treatment of scalar transport equations (i.e. for chemical species) will be presented. In our method, discrete conservation and no penetration of species across solid boundaries are enforced using a cut-cell finite volume scheme. The small cell problem inherent to the method is tackled using explicit-implicit domain decomposition for scalar, within the FDS time integration scheme. Some details on the derivation, implementation and numerical tests of this numerical scheme will be discussed.

4:44PM L7.00004 An Adaptive and Implicit Immersed Boundary Method for Cardiovascular Device Modeling , AMNEET PAL S. BHALLA, BOYCE E. GRIFFITH, University of North Carolina at Chapel Hill — Computer models and numerical simulations are playing an increasingly important role in understanding the mechanics of fluid-structure interactions (FSI) in cardiovascular devices. To model cardiac devices realistically, there is a need to solve the classical fluid-structure interaction equations efficiently. Perkins explicit immersed boundary method is one such approach to model FSI equations for elastic structures efficiently. However, in the presence of rigid structures the IB method faces a severe timestep restriction. To overcome this limitation, we are developing an implicit version of immersed boundary method on adaptive Cartesian grids. Higher grid resolution is employed in spatial regions occupying the structure while relatively coarser discretization is used elsewhere. The resulting discrete system is solved using geometric multigrid solver for the combined Stokes and elasticity operators. We use a rediscritization approach for standard finite difference approximations to the divergence, gradient, and viscous stress. In contrast, coarse grid versions of the Eulerian elasticity operator are constructed via a Galerkin approach. The implicit IB method is tested for a pulse duplicator cardiac device system that consists of both rigid mountings and elastic membrane.
A high-order Immersed Boundary method for solving fluid problems on arbitrary smooth domains. — We present a robust, flexible, and high-order Immersed Boundary method for solving the equations of fluid motion on domains with smooth boundaries using FFT-based spectral methods. The solution to the PDE is coupled with an equation for a smooth extension of the unknown solution; high-order accuracy is a natural consequence of this additional global regularity. The method retains much of the simplicity of the original Immersed Boundary method, and enables the use of simple implicit and implicit/explicit timestepping schemes to be used to solve a wide range of problems. We show results for the Stokes, Navier-Stokes, and Oldroyd-B equations.

Implicit solution of Navier-Stokes equations on staggered curvilinear grids using a Newton-Krylov method with a novel analytical Jacobian. — IMAN BORAZJANI, HAFEZ ASGHARZADEH, The State University of New York at Buffalo (SUNY) — Flow simulations involving complex geometries and moving boundaries suffer from time-step size restriction and low convergence rates with explicit and semi-implicit schemes. Implicit schemes can be used to overcome these restrictions. However, implementing implicit solver for nonlinear equations including Navier-Stokes is not straightforward. Newton-Krylov subspace methods (NKMs) are one of the most advanced iterative methods to solve non-linear equations such as implicit discretization of the Navier-Stokes equation. The efficiency of NKMs massively depends on the Jacobian formation method, e.g., automatic differentiation is very expensive, and matrix-free methods slow down as the mesh is refined. Analytical Jacobian is inexpensive method, but derivation of analytical Jacobian for Navier-Stokes equation on staggered grid is challenging. The NKM with a novel analytical Jacobian was developed and validated against Taylor-Green vortex and pulsatile flow in a 90 degree bend. The developed method successfully handled the complex geometries such as an intracranial aneurysm with multiple overset grids, and immersed boundaries. It is shown that the NKM with an analytical Jacobian is 3 to 25 times faster than the fixed-point implicit Runge-Kutta method, and more than 100 times faster than automatic differentiation depending on the grid (size) and the flow problem. The developed methods are fully parallelized with parallel efficiency of 80-90% on the problems tested.

An efficient immersed boundary projection method for flow around moving bodies. — WEI-XI HUANG, RU-YANG LI, CHUN-MEI XIE, CHUN-XIAO XU, Tsinghua University, TURBULENCE RESEARCH TEAM — An immersed boundary method based on the projection approach is proposed for simulation of flow over moving bodies. In this framework, the momentum forcing added to the incompressible Navier-Stokes equations acts as a Lagrangian multiplier to satisfy the no-slip condition on the immersed boundary, as the role of the pressure on enforcing the divergence-free constraint. The fractional step method with a fully implicit time-advancement scheme is adopted to compute the system, thus eliminating the CFL limitation. Based on the approximate block LU decomposition, velocity-pressure-momentum forcing decoupling is achieved. Moreover, decoupling of the intermediate velocity components and further decoupling of the three directions of the Cartesian coordinates for each velocity component is also performed. As a result, tridiagonal matrix systems for the intermediate velocity, the pressure Poisson equation, and a linear system for the momentum forcing which is fully implicit time-advancement scheme is adopted to compute the system, thus eliminating the CFL limitation. Based on the approximate block LU decomposition, velocity-pressure-momentum forcing decoupling is achieved. Moreover, decoupling of the intermediate velocity components and further decoupling of the three directions of the Cartesian coordinates for each velocity component is also performed. As a result, tridiagonal matrix systems for the intermediate velocity, the pressure Poisson equation, and a linear system for the momentum forcing which is one-order lower than the fluid dimensions, are solved, resulting in a significant saving of the computation cost. Both the temporal and spatial accuracies of the proposed method are tested. For validation, several benchmark numerical examples are presented, including flow over 2D and 3D moving bodies.

A discrete-forcing immersed boundary method with a semi-implicit predictor for weakly-coupled fluid-structure interaction. — WOOJIN KIM, INJAE LEE, HAECHEON CHOI, Seoul National University — We present a weak coupling approach for the fluid-structure interaction using a discrete-forcing immersed boundary method. The incompressible Navier-Stokes equations and the motion of a solid body are based on the Eulerian and Lagrangian coordinates, respectively. A semi-implicit Euler method is applied to the governing equation of a solid body for obtaining provisional position and velocity of a solid body prior to implicitly solving each governing equation. Then, both equations are implicitly solved to obtain a sufficiently large computational time step size. The present weak-coupling approach shows a second-order temporal accuracy and stable solutions for problems with a low density ratio (fluid to solid) without requiring an iterative method. With the present method, we simulate several fluid-structure interaction problems including the flows around a freely vibrating circular cylinder, a flexible beam attached to a circular cylinder, a flapping flag, a flexible plate, and an elastic vocal fold. The results obtained agree well with those from previous studies. All the simulations are conducted at maximum CFL numbers of 1.0-1.5.

Numerical simulation of Shallow water wave propagation around arrays of emerged bodies. — AMIR ZAINALI, ROBERT WEISS, Virginia Tech — Flow around the fixed groups of localized bodies are often encountered in the context of environmental fluid mechanics. Common examples include surface wave propagation through vegetation and flow around the offshore and onshore structures. In the literature, shallow water equations (SWE) are frequently used to model environmental flows. Due to conservative and shock-capturing properties they provide us with good approximations of the wave breaking and runup. In addition, various studies have shown that the inclusion of dispersive effects before the wave breakup can be of a crucial importance. To model the interaction of the wave with the emerged bodies, the exact geometry of the emerged bodies can be considered in the model. However, this approach can be computationally very expensive, particularly if we want to model the interaction of arrays of bodies with complicated geometries. Alternatively, an immersed boundary method can be used. This approach provides us with a significant improvement in numerical efficiency with a negligible numerical accuracy loss. In this study, we use the fully nonlinear and weakly dispersive Green-Naghdi model, coupled with Brinkman penalization technique to simulate the interaction between fluid flow and emerged bodies.

Level set immersed boundary method for gas-liquid-solid interactions. — SHIZHAO WANG, ELIAS BALARAS, The George Washington University — We will discuss an approach to simulate the interaction between free surfaces and deformable structures. In our formulation the Navier-Stokes equations are solved on a block-structured grid with adaptive mesh refinement, and the pressure jumps across the interface between different phases, which is tracked using a level set approach, are sharply defined. Deformable structures are simulated with a solid mechanics solver utilizing a finite element method. The overall approach is tailored to problems with large displacement/deformations. The boundary conditions on a solid body are imposed using a direct forcing, immersed boundary method (Vanella & Balaras, J. Comput. Physics, 228(18), 6617-6628, 2009). The flow and structural solvers are coupled by a predictor-corrector, strong-coupling scheme. The consistency between the Eulerian field based level set method for fluid-fluid interface and Lagrangian marker based immersed boundary method for fluid-structure interface is ensured by reconstructing the flow field around the three phase intersections. A variety of 2D and 3D problems ranging from water impact of wedges, entry and exit of cylinders and flexible plates interacting with a free surfaces, are presented to demonstrate the accuracy of the proposed approach.

Supported by ONR N000141110588 monitored by Dr. Thomas Fu.
6:15PM L7.00011 A 2D domain decomposition, a customized immersed boundary method and a zest of numerical dissipation: a successful cocktail to tackle turbulence on HPC systems, SYLVAIN LAIZET, Imperial College London, ERIC LAMBALLAIS, P’ Institute, Poitiers, J. CHRISTOS VASSILICOS, Imperial College London — Incompact3d is a high-order flow solver dedicated to Direct and Large Eddy Simulations (DNS/LES) using High Performance Computing (HPC) systems which is devoted to turbulent flows at the interface between academic research and upstream industrial R&D. It is originating from the University of Poitiers (France) and was developed there as well as, more recently, in the Turbulence, Mixing and Flow Control Group at Imperial College London (UK). This high-order flow solver can reconcile accuracy, efficiency, versatility and scalability using a simple Cartesian mesh and up to one million computational cores. The three key ingredients of this successful cocktail to tackle turbulence on HPC systems will be given in this talk followed by various applications such as fractal-generated turbulence, gravity currents in an open basin, impinging jets on a heated plate and a micro-jet device to control a turbulent jet.

Monday, November 23, 2015 4:05PM - 6:41PM — Session L8 CFD: High Order and Discontinuous Galerkin Methods

4:05PM L8.00001 Combined Immersed-Boundary / High-Order Finite Difference Methods For Simulations of Acoustic Scattering1, WALTER ARIAS-RAMIREZ, UNICAMP-Univ de Campinas, BRITTON OLSON, Lawrence Livermore National Laboratory, WILLIAM WOLF, UNICAMP-Univ de Campinas, LAWRENCE LIVERMORE NATIONAL LABORATORY TEAM, UNIVERSITY OF CAMPINAS TEAM — The suitability of a continuing forcing immersed boundary method (IBM) combined with a high-order finite difference method is examined on several acoustic scattering problems. A suite of two-dimensional numerical simulations of canonical cases are conducted with the aim of analyzing the error behavior associated with the IBM, through wave reflection, wave diffraction, and the shock-boundary layer interaction phenomena. The compressible Navier-Stokes equations are solved using the Miranda code developed at Lawrence Livermore National Laboratory. Comparison of analytical solution against numerical results is shown for different flow parameters. Preliminary results indicate that the continuing forcing approach has the largest error in wave reflection compared to analytical solution.

This work was performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract No. DE-AC52-07NA27344.

4:18PM L8.00002 A high-order solver for unsteady incompressible Navier-Stokes equations using the flux reconstruction method on unstructured grids with implicit dual time stepping1, CHRISTOPHER COX, CHUNLEI LIANG, MICHAEL PLESNIAK, George Washington Univ — This paper reports development of a high-order compact method for solving unsteady incompressible flow on unstructured grids with implicit time stepping. The method falls under the class of methods now referred to as flux reconstruction/correction procedure via reconstruction. The governing equations employ the classical artificial compressibility treatment, where dual time stepping is needed to solve unsteady flow problems. An implicit non-linear lower-upper symmetric Gauss-Seidel scheme with backward Euler discretization is used to efficiently march the solution in pseudo time, while a second-order backward Euler discretization is used to march in physical time. We verify and validate implementation of the high-order method coupled with our implicit time-stepping scheme. Three-dimensional results computed on many processing elements will be presented. The high-order method is proven effective in parallel computing and can easily be extended to moving and deforming grids. The current implicit time stepping scheme is very effective in satisfying the divergence-free constraint on the velocity field in the artificial compressibility formulation within the context of the high-order flux reconstruction method.

1This work was performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract No. DE-AC52-07NA27344.

4:31PM L8.00003 High-order boundary layer analysis using B-splines on hybrid unstructured meshes, ALVIN ZHANG, ONKAR SAHNI, RPI — Boundary layer flows are present in many engineering applications. In such flows, boundary layers span only a fraction of the characteristic length of the problem near the walls and possess large velocity gradients in the wall normal direction. This mandates use of a layered and graded mesh with a dense anisotropic h-resolution near the walls in order to accurately resolve the boundary layer. To account for complex geometries, a hybrid unstructured mesh approach is adopted. In this approach, the mesh is decomposed into wall normal and wall parallel directions. An alternative to an anisotropic h-resolution is to use a similar setting for both h- and p-resolution possibly with greater smoothness. For this purpose a mixed B-spline basis becomes attractive, where B-splines are used in the wall-normal direction and a C0 basis in the wall-parallel directions as well as the fully unstructured region of the mesh. A mixed B-spline basis offers several advantages over the traditional C0 basis utilized in finite element methods, which include greater accuracy per degree-of-freedom, ease of p-refinement as well as potential for k-refinement. In this study we demonstrate that the mixed B-spline basis, defined for the hybrid unstructured mesh, accurately models the boundary layer behavior.

1Financial support provided under the GW Presidential Merit Fellowship
4:44PM L8.00004 The direct Discontinuous Galerkin method for the compressible Navier-Stokes equations on arbitrary grids, XIAOQUAN YANG, JIAN CHENG, North Carolina State University, TIEGANG LIU, Beihang University, HONG LUO, North Carolina State University — The direct discontinuous Galerkin (DDG) method based on a traditional discontinuous Galerkin (DG) formulation is extended and implemented for solving the compressible Navier-Stokes equations on arbitrary grids. Compared to the widely used second Bassi-Rebay (BR2) scheme for the discretization of diffusive fluxes, the DDG method has two attractive features: first, it is simple to implement as it is directly based on the weak form, and therefore there is no need for any local or global lifting operator; second, it can deliver comparable results, if not better than BR2 scheme, in a more efficient way with much less CPU time. Two approaches to perform the DDG flux for the Navier–Stokes equations are presented in this work, one is based on conservative variables, the other is based on primitive variables. In the implementation of the DDG method for arbitrary grid, the definition of mesh size plays a critical role as the formation of viscous flux explicitly depends on the geometry. A variety of test cases are presented to demonstrate the accuracy and efficiency of the DDG method for discretizing the viscous fluxes in the compressible Navier-Stokes equations on arbitrary grids.

4:57PM L8.00005 A New Reconstructed Discontinuous Galerkin Method for Compressible Flows on Unstructured Grids, JIAN CHENG, North Carolina State University, TIEGANG LIU, Beihang University, HONG LUO, North Carolina State University — A reconstructed discontinuous Galerkin method (rDG) has been developed for solving the compressible Euler equations on unstructured grids. The rDG method is designed not only to enhance the accuracy of the discontinuous Galerkin method, but also to avoid non-physical oscillations in the vicinity of discontinuities. In this work, a new hybrid least-squares reconstruction scheme is developed for the reconstructed discontinuous Galerkin method rDG(P1P2) for compressible flows on unstructured grids. The new hybrid least-squares reconstruction can be regarded as a combination of least-squares recovery method and least-square reconstruction method. Compared to Green-Gauss reconstruction and original least-squares reconstruction, the new hybrid least-squares reconstruction method can strictly satisfy 2-exact property when obtain a quadratic polynomial representation of the underlying discontinuous Galerkin linear polynomial solution on each cell. The numerical experiments for a variety of flow problems demonstrate that this new hybrid reconstruction method is more accurate than the Green-Gauss and the original least-squares reconstruction method, and is able to achieve the designed third-order of accuracy for both inviscid and viscous flow problems.

5:10PM L8.00006 Boundary treatment for the Recovery discontinuous Galerkin method with application to the Navier-Stokes equations, PHILIP JOHNSON, ERIC JOHNSEN, Univer of Michigan - Ann Arbor — The Recovery discontinuous Galerkin (DG) method is a highly accurate approach to computing diffusion problems, which achieves up to 3p+2 convergence rates on Cartesian cells, where p is the order of the polynomial basis. Based on the construction of a unique and differentiable solution across cell interfaces, Recovery DG has mostly been investigated on periodic domains. However, whether such accuracy can be sustained for Dirichlet and Neumann boundary conditions has not been thoroughly explored. We present boundary treatments for Recovery DG on 2D Cartesian geometry that exhibit up to 3p+2 convergence rates and are stable. We demonstrate the efficiency of Recovery DG in context with other commonly used approaches using scalar shear diffusion problems and apply it to the compressible Navier-Stokes equations. The extension of the method to perturbed quadrilateral cells, rather than Cartesian, will also be discussed.

5:23PM L8.00007 Quantifying numerical dissipation rate for discontinuous Galerkin methods, JULIAN DOMARADZKI, GIACOMO CASTIGLIONI, USC, FELIX SCHRANNER, TU Munich, NICOLAS KRAIS, ANDREA BECK, CLAUS-DIETER MUNZ, Uni Stuttgart — The numerical dissipation quite often can be large for typical Finite Volume and Finite Difference schemes. In LES approaches it is an important feature as it enables a preservation of turbulence. For DG methods, the numerical dissipation is typically of second order, but there are cases where the strong algebraic dissipation becomes significant. Because of that there is an increasing interest in CFD using the discontinuous Galerkin (DG) methods because they are of high order and have the ability to handle complex domains. We present comparison between numerical dissipation rates computed for the DG method and for standard FV methods. The numerical dissipation is estimated following Schranner et al. (2015), allowing to compute the numerical dissipation rate for arbitrary sub-domains in a self-consistent way, using only information provided by the code in question. The specific flow considered is a 3D Taylor-Green vortex flow which is simulated with 643 degrees of freedom and for different divisions of the computational domain into elements with polynomial orders inside elements varying from 3 to 31. We find that for low polynomial orders the numerical dissipation of the DG method is comparable to what is observed for the FV codes at the same resolution but it decreases by an order of magnitude for the polynomials of the highest order used.

5:36PM L8.00008 Fully-Implicit Reconstructed Discontinuous Galerkin Method for Stiff Multiphysics Problems, ROBERT NOURGALIEV, Lawrence Livermore National Laboratory — A new reconstructed Discontinuous Galerkin (rDG) method, based on orthogonal basis/test functions, is developed for fluid flows on unstructured meshes. Orthogonality of basis functions is essential for enabling robust and efficient fully-implicit Newton-Krylov based time integration. The method is designed for generic partial differential equations, including transient, hyperbolic, parabolic or elliptic operators, which are attributed to many multiphysics problems. We demonstrate the method’s capabilities for solving compressible fluid-solid systems (in the low Mach number limit), with phase change (melting/solidification), as motivated by applications in Additive Manufacturing. We focus on the method’s accuracy (in both space and time), as well as robustness and solvability of the system of linear equations involved in the linearization steps of Newton-based methods. The performance of the developed method is investigated for highly-stiff problems with melting/solidification, emphasizing the advantages from tight coupling of mass, momentum and energy conservation equations, as well as orthogonality of basis functions, which leads to better conditioning of the underlying (approximate) Jacobian matrices, and rapid convergence of the Krylov-based linear solver.

5:49PM L8.00009 A new hybrid RANS/LES technique based on Reynolds stress reconstruction, MICHELE NINI, ANTONELLA ABBÁ, Politecnico di Milano, MASSIMO GERMANO, Duke University, MARCO RESTELLI, Max Planck Institut — A new hybrid RANS/LES technique, based on the hybrid RANS/LES filter, has been studied. The novelty herein introduced is represented by the reconstruction of the Reynolds stress tensor. As a consequence, no explicit RANS model is needed. The model is implemented in a numerical code based on a high order Discontinuous Galerkin (DG) finite element formulation. The test case considered for numerical simulations is the turbulent channel flow at Mach = 0.2 and the simulations have been carried out for two friction Reynolds number, 180 and 395. The results show that, for coarse grid, the technique can give benefits with respect to the pure LES, confirming that the methodology herein proposed represents a promising approach to the numerical simulation of turbulent flows.
Simulation (LES). The results will be compared to the available experimental data and the flow physics will be discussed. The flowfield of the direction, thus yielding a negative thrust. The interaction of the freestream with the strong reverse flow from the propeller creates massive velocity gradient. The model is successfully tested for canonical turbulent flows (isotropic turbulence, turbulent channel flow, mixing layer).

The resulting relaxation parameter depends on the invariants of the discrete relaxation model. This balance is imposed at the discrete level; here using a second-order finite-volume discretization. Notice that the relaxation parameter is then determined such that the production of all box-fitting scales is counterbalanced by the dissipation associated with all scales. To verify that the scales of motion are truncated properly by the LES-model an explicit box filter is introduced. The approach can in principle be applied to any discretization method. The funding from NSF CAREER award is greatly acknowledged.

The author would like to acknowledge the support of funding from Stanford Graduate Fellowship. This work is supported by the Office of Naval Research.

6:15PM L8.00011 An Adaptive De-Aliasing Strategy for Discontinuous Galerkin methods, ANDREA BECK, DAVID FLAD, HANNES FRANK, CLAUS-DIETER MUNZ, Univ Stuttgart — Discontinuous Galerkin methods combine the accuracy of a local polynomial representation with the geometrical flexibility of an element-based discretization. In combination with their excellent parallel scalability, these methods are currently of great interest for DNS and LES. For high order schemes, the dissipation error approaches a cut-off behavior, which allows an efficient wave resolution per degree of freedom, but also reduces robustness against numerical errors. One important source of numerical error is the inconsistent discretization of the non-linear convective terms, which results in aliasing of kinetic energy and solver instability. Consistent evaluation of the inner products prevents this form of error, but is computationally very expensive. In this talk, we discuss the need for a consistent de-aliasing to achieve a neutrally stable scheme, and present a novel strategy for recovering a part of the incurred computational costs. By implementing the de-aliasing operation through a cell-local projection filter, we can perform adaptive de-aliasing in space and time, based on physically motivated indicators. We will present results for a homogeneous isotropic turbulence and the Taylor-Green vortex flow, and discuss implementation details, accuracy and efficiency.

6:02PM L8.00010 Implicit LES using the Embedded Discontinuous Galerkin method, JOHN MOORE, FlowEDG — High order methods have been gaining greater traction in the CFD community recently, and are believed to be especially well-suited to vortex-dominated flows and Large Eddy Simulation (LES). However, realizing the theoretical performance of these methods has been difficult, in part due to the time step restrictions of explicit methods and the large number of coupled degrees of freedom arising from implicit high order schemes. In this presentation, the development and efficient implementation of an implicit high-order solver based on the Embedded Discontinuous Galerkin (EDG) method, which requires less coupled degrees of freedom than standard DG, is detailed. Results are presented for several external flow cases and validated against experimental results.

The author would like to acknowledge the support of funding from Stanford Graduate Fellowship.

4:44PM L9.00004 LES of propelled bodies in crashback, PRAVEEN KUMAR, KRISHNAN MAHESH, University of Minnesota — Crashback is an off-design operating condition to quickly stop a propelled vehicle by rotating the propeller in reverse direction, thus yielding a negative thrust. The interaction of the freestream with the strong reverse flow from the propeller creates massive unsteadiness and flow separation. This talk will discuss our work towards simulation of crashback flow over an entire hull using Large-Eddy Simulation (LES). The results will be compared to the available experimental data and the flow physics will be discussed. The flowfield of the hull-attached propeller in crashback will be analyzed using dynamic mode decomposition to understand the mechanism of the unsteady loads.

The author would like to acknowledge the support of funding from Stanford Graduate Fellowship.
Large eddy simulation on unstructured meshes using Lagrangian subgrid-scale model for complex turbulent flows. STEVEN TRAN, ONKAR SAINI, RPI, RPI TEAM -

Large eddy simulations (LES) provide high fidelity in which the large-scale turbulent structures are resolved while their interactions with the subgrid scales are modeled. In a Smagorinsky-based LES approach, the unresolved stresses are modeled using an eddy viscosity which in turn involves a model parameter that is unknown a priori and varies in space and time for complex problems. Therefore, dynamic procedures are employed to determine this parameter where averaging is applied to make the procedure robust. When applicable, spatial averaging is applied across homogeneous directions. However, for complex flows the Lagrangian subgrid-scale model employing averaging over pathlines becomes attractive. In contrast to the dynamic Smagorinsky model, variational multiscale (VMS) models have also been developed for LES. In this study, we investigate dynamic mixed models for LES based on the combinations of the Lagrangian subgrid-scale model and the residual-based VMS (RBVMS) approach to study complex, inhomogeneous turbulent flows on unstructured meshes. Applications range from flow through a channel to flow over an airfoil at a moderate angle of attack. Experimental and DNS data are used to make comparisons.

Large eddy simulation on deforming unstructured meshes using Lagrangian subgrid-scale model. REED CUMMINGS, ONKAR SAINI, STEVEN TRAN, RPI -

Large eddy simulation (LES) provides a high fidelity alternative to direct numerical simulation (DNS) in which computational costs are significantly reduced. LES resolves the large-scale turbulent structures present in the flow while modeling the effect of the unresolved or subgrid scales on resolved scales. The Smagorinsky-based LES approach uses an eddy viscosity to model the unresolved stresses. These models rely on a parameter to calculate the eddy viscosity. For complex flows this parameter varies with space and time, thus dynamic procedures are required to determine the parameter. Additionally, many problems of interest involve moving geometries or deforming domains for which appropriate dynamic procedure is required. This work employs the Lagrangian subgrid-scale model within an arbitrary Lagrangian Eulerian (ALE) formulation involving deforming unstructured meshes. Two cases will be studied including flow through a channel and flow over an airfoil.

Numerical investigation of the convective heat transfer coefficient with longitudinal pitch variation in a staggered tube bank. ASHRAF ALFANDI, Jordan Atomic Energy Commission, JUHYEON YOON, Korea Atomic Energy Research Institute, KHALIFEH ABUSALEEM, University of Jordan, MOHAMMAD ALBATI, SALIH KHAFAJI, Jordan Atomic Energy Commission —

In this study, the effect on a shell-side heat transfer coefficient is investigated using the CFD code FLUENT with a variation in longitudinal pitch to diameter ratio. The benchmark purposes with the available empirical correlation, typical thermal-hydraulic conditions for the Zukauskas correlation are assumed. Many sensitivity calculations for different mesh sizes and turbulent models are performed to check the accuracy of the numerical solution. A realistic-κ-ε turbulence model was found to be in good agreement with results of the Zukauskas correlation among the other models employed, at least for the staggered tube bank. It was found that the average heat transfer coefficient of a crossflow over a staggered tube bank calculated using FLUENT is in good agreement with the Zukauskas correlation-calculated heat transfer coefficient in the range of 1.15 – 2.6. For a staggered tube bank, using the Zukauskas correlation seems to be valid down to SL = 1.15.

Implicit Large Eddy Simulation of a wingtip vortex at Re = 1.2 x 10^5. JEAN-ELOI LOMBARD, DAVE MOXEY, SPENCER SHERWIN, Imperial College London, SHERWINLAB TEAM —

We present recent developments in numerical methods for performing a Large Eddy Simulation (LES) of the formation and evolution of a wingtip vortex. The development of these vortices in the near wake, in combination with the large Reynolds numbers present in these cases, make these types of test cases particularly challenging to investigate numerically. To demonstrate the method’s viability, we present results from numerical simulations of flow over a NACA 0012 profile wingtip at Re = 1.2 x 10^5 and compare them against experimental data, which is to date the highest Reynolds number achieved for a LES that has been correlated with experiments for this test case. Our model correlates favorably with experiment, both for the characteristic jetting in the primary vortex and pressure distribution on the wing surface. The proposed method is of general interest for the modeling of transitioning vortex dominated flows over complex geometries.

Large-eddy simulations of a propelled submarine model. ANTONIO POSA, ELIAS BALARAS, The George Washington University —

The influence of the propeller on the wake as well as the evolution of the turbulent boundary layers over an appended notional submarine geometry (DARPA SUBOFF) is reported. The present approach utilizes a Lagrangian subgrid-scale model within an arbitrary Lagrangian Eulerian (ALE) formulation involving deforming unstructured meshes. Two cases will be studied including flow through a channel and flow over an airfoil.

Numerical Analysis of the Acoustic Field of Tip-Clearance Flow. S.M. ALAVI MOGHADAM, Research Scientist, M. MEINKE TEAM, W. SCHRÖDER TEAM —

Numerical simulations of the acoustic field generated by a shrouded axial fan are studied by a hybrid fluid-dynamics-acoustics method. In a first step, large-eddy simulations are performed to investigate the dynamics of tip clearance flow for various tip gap sizes and to determine the acoustic sources. The simulations are performed for a single blade out of five blades with periodic boundary conditions in the circumferential direction on a multi-block structured mesh with 1.4 x 10^5 grid points. The turbulent flow is simulated at a Reynolds number of 9.36 x 10^5 at undisturbed inflow conditions and the results are compared with experimental data. The diameter and strength of the tip vortex increase with the tip gap size, while simultaneously the efficiency of the fan decreases. In a second step, the acoustic field on the near field is determined by solving the acoustic perturbation equations (APE) on a mesh for a single blade consisting of approx. 9.8 x 10^5 grid points. The overall agreement of the pressure spectrum and its directivity with measurements confirm the correct identification of the sound sources and accurate prediction of the acoustic duct propagation. The results show that the longer the tip gap size the higher the broadband noise level.
6:15PM L9.00011 New approaches to the design optimization of hydrofoils. POORIYA BEYHAGHI, GIANLUCA MENEGHELLO, THOMAS BEWLEY, University of California San Diego — Two simulation-based approaches are developed to optimize the design of hydrofoils for foiling catamarans, with the objective of maximizing efficiency (lift/drag). In the first, a simple hydrofoil model based on the vortex-lattice method is coupled with a hybrid global and local optimization algorithm that combines our Delaunay-based optimization algorithm with a Generalized Pattern Search. This optimization procedure is compared with the classical Newton and conjugate gradient method. The accuracy of the vortex-lattice simulations is compared with a more accurate and computationally expensive LES-based simulation. In the second approach, the (expensive) LES model of the flow is used directly during the optimization. A modified Delaunay-based optimization algorithm is used to maximize the efficiency of the optimization, which measures a finite-time averaged approximation of the infinite-time averaged value of an ergodic and stationary process. Since the optimization algorithm takes into account the uncertainty of the finite-time averaged approximation of the infinite-time averaged statistic of interest, the total computational time of the optimization algorithm is significantly reduced. Results from the two different approaches are compared.

Monday, November 23, 2015 4:05PM - 6:41PM –
Session L10 Granular Flows: General 110 - Anthony Rosato, NJIT

4:05PM L10.00001 Dynamics of a Tapped Granular Column, ANTHONY ROSATO, Granular Science Lab, New Jersey Institute of Technology, DENIS BLACKMORE, Mathematical Sciences, New Jersey Institute of Technology, LUO ZUO, Granular Science Lab, New Jersey Institute of Technology, WU HAO, DAVID HORTNROP, Mathematical Sciences, New Jersey Institute of Technology — We consider the behavior of a column of spheres subjected to a time-dependent vertical taps. Of interest are various dynamical properties, such as the motion of its mass center, its response to taps of different intensities and forms, and the effect of system size and material properties. The interplay between different time and length scales are the key contributors to the column’s evolving dynamics. Soft sphere discrete element simulations were conducted over a very wide parameter space to obtain a portrait of column behavior as embodied by the collective dynamics of the mass center motion. Results compared favorably with a derived reduced-order paradigm of the mass center motion (surprisingly analogous to that for a single bouncing ball on an oscillating plate) with respect to dynamical regimes and their transitions. A continuum model obtained from a system of Newtonian equations, as a locally averaged limit in the transport mode along trajectories is described, and a numerical solution protocol for a one-dimensional system is outlined. Typical trajectories and density evolution profiles are shown. We conclude with a discussion of our investigations to relate predictions of the continuum and reduced dynamical systems models with discrete simulations.

4:18PM L10.00002 Spreading granular material with a blade, EMMIE DRESSAIRE, VACHITAR SINGH, NYU Polytechnic School of Engineering, ALBAN SAURET, SVI, CNRS/Saint-Gobain — The spreading of a complex fluid with a blade is encountered in applications that range from the building of granular material in construction projects to the coating of substrates with fluids in industrial applications. This spreading process is also present in everyday life, when we use a knife to turn a lump of peanut butter into a thin layer over our morning toast. In this study, we rely on granular media in a model experiment to describe the three-dimensional spreading of the material. Our experimental set-up allows tracking the spreading of a sandpile on a translating flat surface as the blade remains fixed. We characterize the spreading dynamics and the shape of the spread fluid layer when varying the tilt of the blade, its spacing with the surface and its speed. Our findings suggest that it is possible to tune the spreading parameters to optimize the coating.

4:31PM L10.00003 Granular media in transformation: dynamics and structure, AYMERIC MERCERON, PIERRE JOP, ALBAN SAURET, SVI, CNRS/Saint-Gobain, SVI, CNRS/SAIN-GOIBAN TEAM — Sintering, glas melting and other industrially relevant processes turn batches of grains into continuous end products. Such processes involve complex and mostly misunderstood chemical and physical transformations of the granular packing. Affecting the contact network, physicochemical reactions entail mechanical rearrangements. But such reorganizations may also trigger new potential reactions. Granular reactive systems are strongly coupled and need investigations for achieving industrial optimizations. This study is focused on how transformations appearing on its components affect the response of the granular packing. Inert brass disks and grains undergoing well-known transformations like volume decrease are mixed and then confined in a vertical 2D cell. While the system reacts, the granular packing is regularly photographed with a high-resolution camera. Events largely distributed both spatially and temporally occur around reactive grains. Thanks to image processing, this reorganization process is then analyzed. Spatial and temporal amplitudes of events are quantified as well as their local and global impacts on the granular structure.

4:44PM L10.00004 DEM simulations of shear flow of spherical particles mixed with long granular rods, OLEH BARAN, CD-adapco — Using Discrete Element Method (DEM) I investigate the effect of adding rigid rod-shape particles to the granular flow of spherical particles inside ring shear tester. The simulated geometry includes an annulus, bounded by two concentric cylindrical walls resting on a stationary bottom disk and covered with a top lid. Both the top lid and the bottom have protruding vanes oriented radially and uniformly spaced around the annulus, to prevent slipping of the bulk solid, see image at this link. The top lid rotates with a controlled angular speed and applies a constant normal load to the tested material. I analyze the results for shear stress on the top lid as a function of time for the mixture of spheres and rods and compare these results with ones obtained for the same amount of spherical particles without rods. I also present the analysis of the orientation of granular rods in a shear flow and discuss the results in terms of new time-scale related to the mobility of rods.

4:57PM L10.00005 Novel Discrete Element Method for 3D non-spherical granular particles, LUUK SEelen, JOHAN PADDING, HANS KUIPERS, Eindhoven University of Technology — Granular materials are common in many industries and nature. The different properties from solid behavior to fluid like behavior are well known but less well understood. The main aim of our work is to develop a discrete element method (DEM) to simulate non-spherical granular particles. The non-spherical shape of particles is important, as it controls the behavior of the granular materials in many situations, such as static systems of packed particles. In such systems the packing fraction is determined by the particle shape. We developed a novel 3D discrete element method that simulates the particle-particle interactions for a wide variety of shapes. The model can simulate quadratic shapes such as spheres, ellipsoids, cylinders. More importantly, any convex polyhedron can be used as a granular particle shape. These polyhedrons are very well suited to represent non-rounded sand particles. The main difficulty of any non-spherical DEM is the determination of particle-particle overlap. Our model uses two iterative geometric algorithms to determine the overlap. The algorithms are robust and can also determine multiple contact points which can occur for these shapes. With this method we are able to study different applications such as the discharging of a hopper or silo. Another application the creation of a random close packing, to determine the solid volume fraction as a function of the particle shape.
5:10PM L10.00006 Stability and Structure of Star-Shape Granules

We acknowledge supports from W.M.Keck Foundation and Research Triangle MRSEC

5:23PM L10.00007 Simulating flow and segregation of cylindrical particles

5:36PM L10.00008 Cohesion of wet grains at high liquid content

1Currently at the Universidad Adolfo Ibanez (Chile)

5:49PM L10.00009 ABSTRACT WITHDRAWN —

6:02PM L10.00010 Flowing layer kinematics for constant dimension flowing layers with variable erosion velocities

6:15PM L10.00011 Models for grains and gas ejection dynamics from a silo

6:28PM L10.00012 Sound of silo's: An experimental investigation into sound emissions from granular flows in a vertical tube
Monday, November 23, 2015 4:05PM - 6:28PM –
Session L11 Convection and Buoyancy-Driven Flows: Turbulence 111 - Najmeh Foroozani,
The Abdus Salam ICTP

4:05PM L11.00001 Re-orientations of the large scale flow in turbulent convection with cubic confinement 1  . NAJMEH FOROOZANI, JOSEPH NIEVELA, The Abdus Salam ICTP, VINCENZO ARMEMIO, University of Trieste, KATEPALLI SREENIVASAN, New York University — Large-eddy simulations (LES) of turbulent Rayleigh-Bénard convection were conducted with a fluid of Prandtl number $Pr = 0.7$ in a fully three dimensional cubic confinement of characteristic width-to-height aspect ratio unity for $Ra = 10^6$ and $10^8$. The model solves the unsteady Navier-Stokes equations under the Boussinesq approximation, using a dynamic Smagorinsky model with a Lagrangian averaging technique for the subgrid scale terms. Under fully developed conditions the flow topology is characterized by a large scale circulation (LSC) or mean wind developing in a plane containing one of the diagonals of the cell, while two counter-rotating vortices develop in the other diagonal plane, resulting in inflow at the midplane. This flow structure is not stable in time, undergoing non-periodic re-orientation or switching between the two diagonal planes. The time-interval over which the flow maintains a particular orientation is not constant. We contrast the three-dimensional time-averaged flow structures with single point measurements (time-series) to shed light on the dynamics of the re-orientations. We observe that as $Ra$ increases the LSC becomes more robust and attains a more squarish-like shape.

4:18PM L11.00002 Azimuthal diffusion of the large-scale circulation of turbulent Rayleigh-Bénard convection 1 . XIAOZHOU HE, DENNIS P. M. VAN GILS, EBERHARD BODENSCHATZ, Max Planck Institute for Dynamics and Self-Organization, Göttingen, Germany, GUENTER AHLERS, Department of Physics, University of California Santa Barbara, USA — We present measurements of the large-scale circulation (LSC) of turbulent Rayleigh-Bénard convection. The sample was a cylinder with height and diameter equal to 1.12 m. We used compressed SF$_6$ gas at pressures up to 19 bars as the fluid. The measurements covered the Rayleigh-number range $10^{12} \leq Ra \leq 10^{14}$ at a Prandtl number $Pr \approx 0.80$. We found that the preferred orientation of the LSC upflow was aligned to the West, consistent with Earth’s Coriolis force. The LSC azimuthal dynamics was diffusive, driven by the small scale turbulent fluctuations. For $Ra \leq 10^{12}$ the Reynolds number $Re^b$ based on the azimuthal diffusivity had a $Ra$ dependence similar to that seen for $10^6 \leq Ra \leq 10^{13}$ and $Pr = 4.38$. The $Pr$ dependence $Re^b \propto Pr^\alpha$ with $\alpha \approx -1.2$ was the same as that found for the Reynolds number based on the mean-square fluctuation velocity in the interior bulk flow. For $Ra = Ra^* \approx 2 \times 10^{13}$ $Re^b$ showed the ultimate-state transition and for $Ra \geq Ra^*_2 \approx 8 \times 10^{13}$ it had a $Ra$ dependence with an exponent of $0.40 \pm 0.02$.

1 Supported by the Max Planck Society, the Volkswagenstiftung, the DFD Sonderforschungsbereich SFB963, and NSF Grant DMR11-58514.

4:31PM L11.00003 Fluctuations of entropy production rate in turbulent thermal convection . FRANCESCO ZONTA, Dept. of Elec., Manag., and Mechanical Engineering, University of Udine, Udine, Italy, SERGIO CHIBBARO, UPMC Univ Paris 06, UMR 7190, Institut Jean Le Rond d’Alembert, F-75005, Paris, France — We use Direct Numerical Simulation (DNS) to compute the entropy production rate ($\dot{\sigma}$) in turbulent thermal convection. The overall entropy production rate is measured injecting a large number of pointwise Lagrangian tracers within the flow and simultaneously sampling velocity and temperature fields along the tracers trajectory. For an isolated system, classical thermodynamics prescribes that $\dot{\sigma}$ always increases until equilibrium. However, we show here that the entropy production rate is characterized by large fluctuations and becomes often negative. We also discuss the fluctuations of $\sigma$ averaged over a time lag $\tau$ in the framework of the fluctuation theorem.

4:44PM L11.00004 Quantifying Rayleigh-Bénard convection via a symmetry approach . HONG-YUE ZOU, Peking University, XI CHEN, Texas Tech University, YUN BAO, Sun Yat-sen University, FAZLE HUSSAIN, Texas Tech University, ZHEN-SU SHE, Peking University — We apply our recent symmetry-based theory of wall bounded turbulent flow - WBT - (i.e. channel, pipe and TBL) to study turbulent Rayleigh-Bénard convection (RBC), which yields a multi-layer description of both mean velocity and temperature profile in the vertical direction. Close analogy to the WBT is developed in terms of two order functions, i.e. a momentum stress length function and a thermal diffusion function. Using the multi-layer formulas, the predictions are in quantitative agreement with DNS and experimental data for the Rayleigh-number ($Ra$) covering seven decades. In particular, a thermal buffer layer is predicted in accordance with previously postulated mixing zone which follows a $Ra^{\alpha/7}$ scaling. Recently observed logarithmic profile of the mean temperature is reproduced, and the $Ra$-dependence of the log profile is explained. The non-homogeneous effects in the horizontal direction of the RBC cell are also characterized by slight variations of the multi-layer parameters (i.e. layer thicknesses), influenced by the plumes and corner vortex in the flow. Thus, the turbulent RBC shares a similar multi-layer structure with the canonical wall-bounded flows whose mean profiles are quantified here for the first time.

4:57PM L11.00005 Forced Convection from Square Cylinder Placed Near a Wall Using Variable Resolution Turbulence Modelling . PRITANSHU RANJAN, ANUPAM DEWAN, Indian Institute of Technology New Delhi — The effect of wall proximity on flow and heat transfer around a square cylinder placed inside a channel is numerically investigated. This flow configuration is a fundamental problem and is widely encountered in several engineering applications. The presence of wall close to the cylinder can alter the shedding process and this in turn can affect the thermal transport in the wake region. Many researchers have studied this phenomenon experimentally but the heat transfer characteristics around a square cylinder placed inside a channel still remain an open question. We present here an insight into this problem. The simulations were carried out for a Reynolds number of 37,000 (based on cylinder diameter, D) and as a function of gap height, $G/D$, at different blockage ratios. A variable resolution modelling approach (PANS SST k-ω model) was used to study turbulence structures. The results are presented in terms of pressure coefficient, drag coefficient, thermal fluctuations and local and average Nusselt number (Nu). The results obtained showed that, for $G/D < 0.5$ very weak shedding process at random time intervals occurs suggesting the suppression of vortex shedding due to wall. Thus, the local and average Nu decrease as the cylinder is moved towards wall at all blockage ratios.
5:10PM L11.00006 An analysis of spatially varying turbulent Prandtl number in a flow with local acceleration and deceleration. EUNBUM JUNG, WOOK LEE, SEONGWON KANG. Sagam Univ., Korea, GIANLUCA IACCARINO, Stanford Univ., USA — The turbulent Prandtl number (Pr) is an important parameter in turbulent flows used in many engineering models for heat transfer. In the present study, spatial variation of Pr is investigated using DNS. We derived a form of Pr applicable to a general flow configuration, using the least-square method in a manner consistent with the turbulent viscosity model in LES. For a flow subject to local acceleration and deceleration induced by the wall geometry, we performed a parametric study for the Reynolds number, Prandtl number and a geometric factor using DNS. A comparison of the data from DNS and RANS with a constant Pr indicates the potential of improved RANS predictions using the present variable Pr subject to the local flow field. Also, it is observed that the local pressure gradient has an important effect on the Pr field. From the flow statistics, a few flow variables showing higher correlations with Pr are identified. An elementary model for Pr is devised, and used for RANS prediction producing a more accurate prediction of the heat transfer rate.

1Corresponding author

5:23PM L11.00007 Effects of Natural Convection on the Near-Wall Turbulence in Unstably Stratified Turbulent Channel Flows. SAMIR SID, VINCENT TERRAPON, Dept. of Aerospace and Mechanical Engineering, University of Liege, Liege, Belgium, YVES DUBIEF, School of Engineering, University of Vermont, Burlington VT — Results of direct numerical simulation of turbulent channel flows under unstable stratification are reported. Two Reynolds number are considered: Re = 180,395 and the Rayleigh number ranges between Ra = [10^6 – 10^9]. The Prandtl number is set to 1. The channel is periodic in both streamwise and spanwise directions and non-slip/isothermal boundary conditions are imposed at the walls. The temperature difference between the walls is set so that the stratification is unstable and the coupling between temperature and momentum is achieved using the Boussinesq approximation. The dependency of the typical large scale convective structures on both Reynolds and Rayleigh numbers are investigated through cross flow sectional statistics and instantaneous flow field visualizations. Moreover, the effects of the natural convection on the coherent structures associated to the cycle of wall-bounded turbulence (Jimenez, et al. JFM 1999), namely velocity streaks and streamwise vortices, are examined. Finally, macroscopic quantities such as friction coefficient and Nusselt number are reported as a function of the Rayleigh number and are compared for both Reynolds numbers.

1The Belgian Team acknowledges computational resources from CéCII (F.R.S.-FNRS grant No.2.5020.11) and the PRACE infrastructure. YD acknowledges the support of NSF and DOE under grant NSF/DOE 1258697.

5:36PM L11.00008 Thermal Convection From a Minimal Flow Unit to a Wide Fluid Layer. CURTIS HAMMAN, PARVIZ MOIN, Center for Turbulence Research, Stanford University — The computations of the “minimal channel” by Jiménez & Moin (1991, JFM) provided a conceptual framework and building block from which to study the structure of near-wall turbulence driven by mean shear. Mean buoyancy, on the other hand, can sustain a vigorous field of very large eddies whose horizontal extent can extend across many full channel heights. We examine the extent to which a flow model consisting of a finite periodic array of such structures can successfully predict certain turbulence statistics in thermal convection with and without a mean flow in very large-aspect ratio channels.

5:49PM L11.00009 Large-eddy simulation of bubble-driven plume in stably stratified flow. DI YANG, University of Houston, BICHENG CHEN, Penn State University, SCOTT SOCOLOFSKY, Texas A&M University, MARCELO CHAMECKI, Penn State University, CHARLES MENEVEAU, Johns Hopkins University — The interaction between a bubble-driven plume and stratified water column plays a vital role in many environmental and engineering applications. As the bubbles are released from a localized source, they induce a positive buoyancy flux that generates an upward plume. As the plume rises, it entrains ambient water, and when the plume rises to a higher elevation where the stratification-induced negative buoyancy is sufficient, a considerable fraction of the entrained fluid detrains, or peels, to form a downward outer plume and a lateral intrusion layer. In the case of multiphase plumes, the intrusion layer may also trap weakly buoyant particles (e.g., oil droplets in the case of a subsea accidental blowout). In this study, the complex plume dynamics is studied using large-eddy simulation (LES), with the flow field simulated by hybrid pseudospectral/finite-difference scheme, and the bubble and dye concentration fields simulated by finite-volume scheme. The spatial and temporal characteristics of the buoyant plume are studied, with a focus on the effects of different bubble buoyancy levels. The LES data provide useful mean plume statistics for evaluating the accuracy of 1-D engineering models for entrainment and peeling fluxes. Based on the insights learned from the LES, a new continuous peeling model is developed and tested.

1Study supported by the Gulf of Mexico Research Initiative (GoMRI).

6:02PM L11.00010 Energy spectrum of stably-stratified and convective turbulent flows. MAHENDRA VERMA, ABHISHEK KUMAR, IIT Kanpur, India — In the inertial range of fluid turbulence, the energy flux is constant, while the energy spectrum scales as k^{-5/3} (k=wavenumber). The buoyancy however could change the phenomenology dramatically. Bolgiano and Obukhov (1959) had conjectured that stably stratified flows (as in atmosphere) exhibits a decrease in the energy flux as k^{-4/5} due to the conversion of kinetic energy to the potential energy, consequently, the energy spectrum scales as k^{-11/5}. We show using detailed numerical analysis that the stably stratified flows indeed exhibit k^{-11/5} energy spectrum for Froude numbers Fr near unity. The flow becomes anisotropic for small Froude numbers. For weaker buoyancy (large Fr), the kinetic energy follows Kolmogorov’s spectrum with a constant energy flux. However, in convective turbulence, the energy flux is a nondecreasing function of wavenumber since the buoyancy feeds positively into the kinetic energy. Hence, the kinetic energy spectrum is Kolmogorov-like (k^{-5/3}) or shallower. We also demonstrate the above scaling using a shell model of buoyancy-driven turbulence.

1A. Kumar, A. G. Chatterjee, and M. K. Verma, PRE, 90, 023016 (2014)

6:15PM L11.00011 ABSTRACT WITHDRAWN —

Monday, November 23, 2015 4:05PM - 6:41PM —
Session L12 Wind Turbines: Wind Farms II 200 - Claire VerHulst, United States Military Academy
4:05PM L12.00001 Enhancing kinetic energy entrainment in LES of large wind farms by unconventional forcing at the turbine rotors

This work was supported by NSF grant 1243182 (the WINDINSPIRE project).

4:18PM L12.00002 Effect of topography on wind turbine power and load fluctuations

Numerical simulations were performed on XSEDE TACC, Grant CT5070066. This work was supported by the NSF, grant IIA-1243482 (WINDINSPIRE).

4:31PM L12.00003 Coupling the Weather Research and Forecasting (WRF) model and Large Eddy Simulations with Actuator Disk Model: predictions of wind farm power production

Numerical simulations were performed at TACC (Grant CT5070066). This work was supported by NSF, (grant IIA-1243482 WINDINSPIRE).

4:44PM L12.00004 Optimal coordinated control of energy extraction in LES of wind farms: effect of turbine arrangement patterns

The authors acknowledge support from the European Research Council (FP7-Ideas, grant no. 306471). Simulations were performed on the infrastructure of the Flemish Supercomputer Center, funded by the Hercules Foundation and the Flemish Government.

4:57PM L12.00005 Low-order representations of a wind turbine array boundary layer via double POD

The truncated basis of sub-modes represents a total reduction to 0.015% of the original degrees of freedom in the wake. Low-order description of the stress tensor is corrected to account for energy excluded from the truncated basis. Root-mean-square error associated with low-order statistics is less than 15% for normal stresses and 3% for shear stresses.
5:10PM L12.00006 A simplified model for average kinetic energy flux within large wind turbine arrays. COREY MARKFORT, IHR-Hydroscience and Engineering, University of Iowa, WEI ZHANG, Cleveland State University, FERNANDO PORTE-AGEL, EPFL — We investigate the kinetic energy distribution within an array of wind turbines using a 1-D model for the interactions between large-scale wind farms and the atmospheric boundary layer (ABL). Obstructed shear flow scaling is used to predict the development length of the wind farm flow as well as vertical momentum flux. Within the region of flow development, momentum and energy are advected into the wind farm and wake turbulence draws excess momentum in from between turbines. This is characterized by large dispersive fluxes. Once the flow within the farm is developed, the area - averaged velocity profile exhibits an inflection point, characteristic of obstructed sheared flows. The inflected velocity profile is responsible for a characteristic turbulence eddy scale, which may be responsible for a significant amount of the vertical momentum and energy flux. Prediction of this scale is useful for determining the amount of available power for harvesting. The model result for kinetic energy flux is compared to wind tunnel measurements. The model is useful for optimizing wind turbine spacing and layout, and for assessing the impacts of wind farms on nearby wind resources and the environment.

5:23PM L12.00007 A simple and complete two-interface model for spatially developing flow in rigid and flexible canopies. SAMANEH SADRI, PAOLO LUZZATTO-FEGIZ, UC Santa Barbara — At the front of a canopy, flow deceleration is associated with strong vertical fluxes of mass and momentum. Accurately describing this region is important in many applications, including terrestrial and aquatic vegetation, as well as large wind farms. Simple models can provide a framework to analyze these flows, thereby guiding and complementing more refined and computationally intensive tools. Jerram et al. (2003) introduced a linearised model that describes the flow field through sparse canopies, albeit at the cost of solving a PDE. A simpler approach involves vertically integrating the governing equations across the canopy, yielding scalings that relate key variables (e.g. Chen & Nepf 2013), which in turn can be used to construct empirical fits. We build a simple and complete model, by separating the flow in three horizontal layers. These comprise the canopy, the overlying boundary layer, and the outer flow, such that exchanges of mass and momentum occur at two interfaces. We parameterize turbulent exchanges by means of the entrainment hypothesis; this is a closure that has been used extensively in other problems in geophysical fluid dynamics. We neglect pressure gradients inside the canopy, but account for upstream pressure variations and retain nonlinear terms. Our two-interface model quantitatively describes the flow velocities and boundary layer heights in developing canopy flows, and successfully accounts for the effect of ambient stratification. Finally, we discuss developments accounting for the effects of flexibility in vegetation canopies.

5:36PM L12.00008 Physical Model Study of the Fully Developed Wind Turbine Array Boundary Layer in the UNH Flow Physics Facility. JOHN TURNER, MARTIN WOSNIK, University of New Hampshire — Results from an experimental study of an array of up to 100 model wind turbines with 0.25 m diameter are reported. The study was conducted in the UNH Flow Physics Facility (FFP), which has test section dimensions of 6.0 m wide, 2.7 m high and 72.0 m long. For a given configuration (spacing, initial conditions, etc.), the model wind farm reaches a fully developed condition, in which turbulence statistics remain the same from one row to the next within and above the wind turbine array. Of interest is the transport of kinetic energy within the wind turbine array boundary layer (WTABL). Model wind farms of up to 20 rows are possible in the FFP at the wind turbine scale used. The present studies in the FFP are able to achieve the fully developed WTABL condition, which can provide valuable insight to the optimization of wind farm energy production. The FFP can achieve a boundary layer height on the order of 1 m at the beginning of the wind turbine array. The wind turbine array was constructed of porous disks, which where drag (thrust) matched to wind turbines at typical operating conditions and therefore act as momentum sinks similar to wind turbines. The flow in the WTABL was measured with constant temperature anemometry using an X-wire.

5:49PM L12.00009 Assessing the Impacts of Low Level Jets over Wind Turbines. WALTER GUTIERREZ RODRIGUEZ, Department of Mechanical Engineering, Texas Tech University, Lubbock, TX, GUILLERMO ARAYA, Mechanical Engineering Department, University of Puerto Rico, Mayaguez, ARQUIMEDES RUIZ-COLUMBIE, National Wind Institute, Texas Tech University, Lubbock, TX, MURAT TUTKUN1, Institute for Energy Technology (IFE), Kjeller, Norway, LUCIANO CASTILLO, Department of Mechanical Engineering, Texas Tech University, Lubbock, TX — Low Level Jets (LLJs) are defined as regions of relatively strong winds in the lower 1-2 km of the atmosphere. They are a common feature over the Great Plains in the United States. This paper is focused on the determination of the static/dynamic impacts that real LLJs in West Texas have over wind turbines and wind farms. High-frequency (50Hz) observational data from the 200-m meteorological tower (Reese, Texas) have been input as inflow conditions into the NREL FAST code in order to evaluate the front of a canopy, flow deceleration is associated with strong vertical fluxes of mass and momentum. Accurately describing this region is important in many applications, including terrestrial and aquatic vegetation, as well as large wind farms. Simple models can provide a framework to analyze these flows, thereby guiding and complementing more refined and computationally intensive tools. Jerram et al. (2003) introduced a linearised model that describes the flow field through sparse canopies, albeit at the cost of solving a PDE. A simpler approach involves vertically integrating the governing equations across the canopy, yielding scalings that relate key variables (e.g. Chen & Nepf 2013), which in turn can be used to construct empirical fits. We build a simple and complete model, by separating the flow in three horizontal layers. These comprise the canopy, the overlying boundary layer, and the outer flow, such that exchanges of mass and momentum occur at two interfaces. We parameterize turbulent exchanges by means of the entrainment hypothesis; this is a closure that has been used extensively in other problems in geophysical fluid dynamics. We neglect pressure gradients inside the canopy, but account for upstream pressure variations and retain nonlinear terms. Our two-interface model quantitatively describes the flow velocities and boundary layer heights in developing canopy flows, and successfully accounts for the effect of ambient stratification. Finally, we discuss developments accounting for the effects of flexibility in vegetation canopies.

6:02PM L12.00010 Properties of wind turbine wakes under various atmospheric stability conditions. SHENGBAI XIE, CRISTINA ARCHER, University of Delaware — Large-eddy simulations (LES) are performed to study the properties of wind turbine wakes under various atmospheric stability conditions. The Wind Turbine and Turbulence Simulator (WTITTS), a 4th-order finite-difference LES code is used for stable, neutral, and unstable conditions. The Coriolis forcing is also considered. Three cases are studied: isolated turbine, finite-size turbine array, and infinite wind farm. The results show strong correlations with stability. For the stable condition, the power extraction by an isolated turbine is highest, but the wake is also longest, thus the relative performance inside the array is lowest. In contrast, although the single-turbine power extraction is low for the unstable condition, the performance of downstream turbines is improved due to faster wake recovery. The wake shape is distorted by the stability-related wind veering. Therefore, the self-similar Gaussian wake deficit is not accurate. Here, a new wake model is proposed for correction. The infinite wind-farm case shows that the temperature near the ground is warmed by about 1 K for the stable condition, but the influence is almost negligible for the unstable and neutral conditions. For all conditions, the near-ground shear stress is reduced.

1Grants: NSF-CBET #1157246, NSF-CMMI #1100948, NSF-PIRE # NSF-OISE-1243482.
2University of Oslo, Department of Mathematics, Oslo, Norway
Real wind turbines, however, operate in the atmospheric boundary layer. They are subjected to mean shear and turbulence, both have been observed to improve wake recovery. Similarity considerations are extended to place a turbulent axisymmetric wake with rotation in a boundary layer flow, and the scaling implications are examined. Corresponding experiments were carried out in the UNH Flow Physics Facility, using model wind turbines of various sizes as swirling wake generators.

1 Supported by NSF CBET grant 1150797

4:18PM L13.00002 Similarity considerations for a turbulent axisymmetric wake with rotation subjected to a boundary layer flow1. MARTIN WOSNIK, University of New Hampshire — Recently an analytical and experimental investigation of the turbulent axisymmetric wake with rotation found a new asymptotic scaling function for the mean swirl, $W_{max} \propto U_{ref}^{3/2} \propto x^{-1}$ (Dufresne and Wosnik, Mar Technol Soc J, 47, no 4, 193-205, 2013). An equilibrium similarity theory derived scaling functions from the conditions for the existence of similarity directly from the equations of motion. Axial and azimuthal (swirl) velocities were measured in the wake of a single 3-bladed wind turbine in a free stream up to 20 diameters downstream, and the data were found to support the theoretical results. The scaling implies that the mean swirl decays faster, with $x^{-1}$, than the mean velocity deficit, with $x^{-2/3}$. Real wind turbines, however, operate in the atmospheric boundary layer. They are subjected to mean shear and turbulence, both have been observed to improve wake recovery. Similarity considerations are extended to place a turbulent axisymmetric wake with rotation in a boundary layer flow, and the scaling implications are examined. Corresponding experiments were carried out in the UNH Flow Physics Facility, using model wind turbines of various sizes as swirling wake generators.

1We gratefully acknowledge support from NSF grant IIA-12124382 (WINDINSPIRE).
4:44PM L13.00004 Optimal control of wind farms for power tracking using simplified one-dimensional convection-diffusion equation¹. CARL SHAPIRO, Johns Hopkins University, PIETER BAUWERAERTS, JOHAN MEYERS, Katholieke Universiteit Leuven, CHARLES MENEVEAU, DENNICE GAYME, Johns Hopkins University — Coordinated control of wind turbines within a wind farm, accounting for wake interactions and associated flow phenomena, has the potential to provide a number of important services to the power grid. In this work we develop a simple time-dependent extension of a standard steady-state wake model that is used to obtain an optimal control strategy for tracking a time-varying power signal. First, we introduce a one-dimensional convection-diffusion equation for wind turbine wakes that is based on the Jensen wake model and the actuator disk model. This equation is tested during wind farm start up by comparing to large-eddy simulations of wind farms with both aligned and staggered turbine arrangements. Second, we investigate optimal control for power tracking applications, where turbines are controlled via the local thrust coefficient. The control strategy is designed to minimize the squared difference between the modeled farm power and a given power reference signal. Finally, the control strategies obtained are tested using large-eddy simulations.

¹CS, CM, and DG are supported by NSF (SEP-1230788 and IIA-1243482, the WINDINSPIRE project). PB and JM are supported by ERC (ActiveWindFarms, grant no: 306471).

4:57PM L13.00005 Investigation on the near-wake flow structures of a utility-scale wind turbine using snowflake based flow visualization¹. TEJA DASARI, MOSTAFA TOLONI, MICHELE GUALA, JIARONG HONG, University of Minnesota — Super-large-scale particle image velocimetry and flow visualization techniques using natural snow particles have been shown as an effective tool to probe the structure of the flow around full-scale wind turbines (Hong et al. *Nature Comm.* 2014). Here we present a follow-up study based on the data collected during a deployment around the 2.5 MW wind turbine at EOLOS Wind Energy Research Station on April, 4th, 2014. The dataset includes the snow visualization of flow fields from different perspectives in the near wake of the turbine. The motions of the dominant coherent structures including tip, blade root, hub and tower vortices, represented by the snow voids, are examined with the objective of quantifying and correlating their behavior with the meteorological and turbine operating conditions. Some preliminary studies on flow-structure interaction are also performed by correlating the data from strain gauges, accelerometers mounted on the turbine blades, with the flow measurements. The statistical analysis of the motions of blade induced vortices shows a clear impact of atmospheric turbulence and vortex interaction on flow development in the near wake. The result further indicates a strong connection between near-wake vorticity field, turbine operation and structure deformations.

¹The work was supported by National Science Foundation (NSF-CBET-1454259) and the research infrastructure was supported by Department of Energy.

5:10PM L13.00006 Studying Wake Deflection of Wind Turbines in Yaw using Drag Disk Experiments and Actuator Disk Modeling in LES¹. MICHAEL HOWLAND, The Johns Hopkins University, JULIAAN BOSSUYT, JOHAN MEYERS, KU Leuven, CHARLES MENEVEAU, The Johns Hopkins University — Recently, there has been a push towards the optimization in the power output of entire large wind farms through the control of individual turbines, as opposed to operating each turbine in a maximum power point tracking manner. In this vane, the wake deflection by wind turbines in yawed conditions has generated considerable interest in recent years. In order to effectively study the wake deflection according to classical actuator disk momentum theory, a 3D printed drag disk model with a coefficient of thrust of approximately 0.75 – 0.85 and a diameter of 3 cm is used, studied under uniform inflow in a wind tunnel with test section of 1 m by 1.3 m, operating with a negligible inlet turbulence level at an inflow velocity of 10 m/s. Mean velocity profile measurements are performed using Pitot probes. Different yaw angles are considered, including 10, 20, and 30 degrees. We confirm earlier results that (e.g.) a 30 degree yaw angle deflects the center of the wake around 1/2 of a rotor diameter when it impinges on a downstream turbine. Detailed comparisons between the experiments and Large Eddy Simulations using actuator disk model for the wind turbines are carried out in order to help validate the CFD model.

¹Work supported by NSF (grants CBET-113380 and IIA-1243482, the WINDINSPIRE project) and by ERC (ActiveWindFarms, grant no. 306471)

5:23PM L13.00009 A Stereo PIV Study on the Wake Characteristics behind Dual-Rotor Wind Turbines¹. HUI HU, ZHENYU WANG, WEI TIAN, Iowa State university — We report an experimental study to investigate the aeromechanics and wake characteristics of dual-rotor wind turbines (DRWTs) with co- and counter-rotating configurations, in comparison to those of a conventional single-rotor wind turbine (SRWT). The experiments were performed in a large-scale Aerodynamic/Atmospheric Boundary Layer (AABL) wind tunnel under neutral stability conditions. In addition to measuring the power outputs and dynamic wind loads acting on the SRWT and DRWT systems, a stereoscopic PIV was used for detailed wake flow field measurements (free-run and phase-locked) to quantify the characteristics of the turbulent turbine wake flow and to reveal visualize the evolution of the unsteady vortex structures in the wakes of DRWTs, in comparison with those behind a conventional SRWT systems. The detailed flow field measurements are correlated with the dynamic wind loads and power output measurements to elucidate underlying physics for higher total power yield and better durability of the wind turbines.

¹The funding support from the Iowa Energy Center with Grant No. 14-008-OG and National Science Foundation (NSF) with Grant Numbers of CBET-1133751 and CBET-1438099 is gratefully acknowledged.
5:36PM L13.00008 Effect of nacelle on wake meandering in a laboratory scale wind turbine using LES

Daniel Foti, Xiaolei Yang, Michele Guala, Fotis Sotiropoulos, St. Anthony Falls Laboratory, University of Minnesota — Wake meandering, large scale motion in the wind turbine wakes, has considerable effects on the velocity deficit and turbulence intensity in the turbine wake from the laboratory scale to utility scale wind turbines. In the dynamic wake meandering model, the wake meandering is assumed to be caused by large-scale atmospheric turbulence. On the other hand, Kang et al. (J. Fluid Mech., 2014) demonstrated that the nacelle geometry has a significant effect on the wake meandering of a hydrokinetic turbine, through the interaction of the inner wake of the nacelle vortex with the outer wake of the tip vortices. In this work, the significance of the nacelle on the wake meandering of turbine wind turbine previously used in experiments (Howard et al., Phys. Fluid, 2015) is demonstrated with large eddies simulations (LES) using immersed boundary method with fine enough grids to resolve the turbine geometric characteristics. The three dimensionality of the wake meandering is analyzed in detail through turbulent spectra and meander reconstruction. The computed flow fields exhibit wake dynamics similar to those observed in the wind tunnel experiments and are analyzed to shed new light into the role of the energetic nacelle vortex on wake meandering.

This work was supported by Department of Energy DOE (DE-EE0002980, DE-EE0005482 and DE-AC04-94AL85000), and Sandia National Laboratories. Computational resources were provided by Sandia National Laboratories and the University of Minnesota Supercomputing Institute.

5:49PM L13.00009 Dynamic Gaussian wake meandering in a restricted nonlinear simulation framework

Joel Bretheim, Johns Hopkins University, Fernando Porte-Agel, Ecole Polytechnique Federale de Lausanne, Dennice Gayme, Charles Meneveau, Johns Hopkins University — Wake meandering can significantly impact the performance of large-scale wind farms. Simplified wake expansion (e.g., Jensen/PARK) models, which are commonly used in industry, lead to accurate predictions of certain wind farm performance characteristics (e.g., time- and row-averaged total power output). However, they are unable to capture certain temporal phenomena such as wake meandering, which can have profound effects on both power output and turbine loading. We explore a dynamic wake modeling framework based on the approach proposed by Larsen et al. (Wind Energy 11, 2008) whereby turbine “wake elements” are treated as passive tracers and advected by an averaged streamwise flow. Our wake elements are treated as Gaussian velocity deficit profiles (Bastankhah and Porte-Agel, Renew. Energy 70, 2014). A restricted nonlinear (RNL) model is used to capture the turbulent velocity fluctuations that are critical to the wake meandering phenomenon. The RNL system, which has been used in prior wall-turbulence studies, provides a computationally affordable way to model atmospheric turbulence, making it more reasonable for use in engineering models than the more accurate but computationally intensive approaches like large-eddy simulation.

This work is supported by NSF (IGERT 0801471, SEP-1230788, and IIA-1243482, the WINDSPIRE project).

6:02PM L13.00010 An Immersed Boundary - Adaptive Mesh Refinement solver (IB-AMR) for high fidelity fully resolved wind turbine simulations

Dionysis Angelidis, Fotis Sotiropoulos, University of Minnesota — The geometrical details of wind turbines determine the structure of the turbulence in the near and far wake and should be taken in account when performing high fidelity calculations. Multi-resolution simulations coupled with an immersed boundary method constitutes a powerful framework for high-fidelity calculations past wind farms located over complex terrains. We develop a 3D Immersed-Boundary Adaptive Mesh Refinement flow solver (IB-AMR) which enables turbine-resolving LES of wind turbines. The idea of using a hybrid staggered/non-staggered grid layout adopted in the Curvilinear Immersed Boundary Method (CURVIB) has been successfully incorporated on unstructured meshes and the fractional step method has been employed. The overall performance and robustness of the second order accurate, parallel, unstructured solver is evaluated by comparing the numerical simulations against conforming grid calculations and experimental measurements of laminar and turbulent flows over complex geometries. We also present turbine-resolving multi-scale LES considering all the details affecting the induced flow field; including the geometry of the tower, the nacelle and especially the rotor blades of a wind tunnel scale turbine.

This material is based upon work supported by the Department of Energy under Award Number DE-EE0005482 and the Sandia National Laboratories.

6:15PM L13.00011 Wind tunnel measurements of the power output variability and unsteady loading in a micro wind farm model

Juliaan Bossuyt, KU Leuven, Michael Howland, Charles Meneveau, Johns Hopkins University, Johan Meyers, KU Leuven — To optimize wind farm layouts for a maximum power output and wind turbine lifetime, mean power output measurements in wind tunnel studies are not sufficient. Instead, detailed temporal information about the power output and unsteady loading from every single wind turbine in the wind farm is needed. A very small porous disc model with a realistic thrust coefficient of 0.75 - 0.85, was designed. The model is instrumented with a strain gage, allowing measurements of the thrust force, incoming velocity and power output with a frequency response up to the natural frequency of the model. This is shown by reproducing the −5/3 spectrum from the incoming flow. Thanks to its small size and compact instrumentation, the model allows wind tunnel studies of large wind turbine arrays with detailed temporal information from every wind turbine. Translating to field conditions with a length-scale ratio of 1:3,000 the frequencies studied from the data reach from 10^{-4} Hz up to about 6 \times 10^{-2} Hz. The model’s capabilities are demonstrated with a large wind farm measurement consisting of close to 100 instrumented models. A high correlation is found between the power outputs of stream wise aligned wind turbines, which is in good agreement with results from prior LES simulations.

Work supported by ERC (ActiveWindFarms, grant no. 306471) and by NSF (grants CBET-113380 and IIA-1243482, the WINDSPIRE project).

Monday, November 23, 2015 4:05PM - 6:41PM —
Session L14 Aerodynamics: Fluid-Structure Interaction II 202 - Behrouz Karami, George Washington University

4:05PM L14.00001 Flag flapping in a channel

Silas Alben, University of Michigan, Kourosh Shoefle, Rajat Mittal, Johns Hopkins Univ., Sourabh Jha, Ari Glezer, Georgia Tech — We study the flapping of a flag in an inviscid channel flow. We focus especially on how quantities vary with channel spacing. As the channel walls move inwards towards the flag, heavier flags become more unstable, while light flags’ stability is less affected. We use a vortex sheet model to compute large-amplitude flapping, and find that the flag undergoes a series of jumps to higher flapping modes as the channel walls are moved towards the flag. Meanwhile, the drag on the flag and the energy lost to the wake first rise as the walls become closer, then drop sharply as the flag moves to a higher flapping mode.
4:18PM L14.00002 Fluidic harvesters in free stream turbulence undergoing flow-induced vibrations or flutter, JOAN GOMEZ, VAHID AZADEH RANJBAR, OLEG GOUSHCHA, YIANNIS ANDREOPoulos, NIELL ELVIN, The City College of New York. In the present experimental work we investigate the performance of fluidic harvesters consisting of cylindrical body mounted at the tip of a flexible beam in the presence of nearly homogeneous and isotropic turbulence. Circular, semi-circular and square shapes have been tested. It was found that turbulence interferes with resonance conditions between the flow and the structure in the case of vortex induced vibrations and has absolutely no effect in flutter dominated case. As a result, turbulence increases the power output of non-linear harvesters subjected to vortex induces vibration and it has no effect in harvester under flutter conditions.

1Supported by NSF Grant: CBET #1033117.

4:31PM L14.00003 Experimental investigation of the effects of high-frequency electroactive morphing on the shear-layer, JOHANNES SCHELLER, KARL-JOSEPH RIZZO, GURVAN JODIN, ERIC DUHAYON, JEAN-FRANCOIS ROUCHON, LAPLACE, Laboratoire Plasma et Conversion d’Energie, UMR CNRS-INPT-UPS No 5213, 2 Rue Charles Camichel, F-31071 Toulouse, France, JULIAN HUNT, Department of Earth Sciences, University College London, London WC1E 6BT, UK, MARIANNA BRAZA, IMFT, Institut de M´ecanique des Fluides de Toulouse, UMR CNRS-INPT-UPS No 5502, All´ee du Prof. Camille Soulou, F-31400 Toulouse, France — Time-resolved PIV measurements are conducted at a Reynolds number of 270,000 downstream of the trailing edge of a NACA4412 airfoil equipped with trailing-edge piezoelectric tab actuators to investigate the high-frequency low-amplitude actuation’s effect on the shear-layer. A comparison of the time-averaged Reynolds stress tensor components at different actuation frequency reveals a significant effect on the vortex dynamics in the wake as well as the heat transfer efficiency. A proper orthogonal decomposition analysis is conducted in order to investigate the actuation’s impact on the vortex breakdown. It will be shown that a specific low-amplitude actuation frequency enables a reduction of the predominant shear-layer frequencies.

4:44PM L14.00004 Self-sustained oscillations of a sinusoidally-deformed plate, DIEGO F MURIEL, EDWIN A COWEN, Cornell University. Motivated by energy harvesting, the oscillatory motion of a deformed elastic material with aspect ratio Length/Width=2 immersed in an incompressible flow is studied experimentally. To induce the wave-like deformation a polycarbonate sheet is placed under longitudinal compression with external forcing provided by equispaced tension lines anchored in a frame. No additional constrains are placed in the material. Based on quantitative image-based edge detection, ADV, and PIV measurements, we document the existence of three natural states of motion. Bellow a critical velocity, a stable state presents a sinusoidal-like deformation with weak small perturbations. Above a critical velocity, instability appears in the form of a traveling wave with predictable dominant frequency accompanied by higher-order harmonics. As the flow velocity increases the instability converges faster to its limit cycle in the phase plane (e.g., vertical velocity and position), until the stable oscillatory mode transitions to chaos showing a broad energy spectrum and unstable limit cycle. The underlying objective is to induce the onset of the instability at lower critical velocities for higher bending rigidities, promoting possible energy extraction and increasing the range at which stable oscillations appear.

5:01PM L14.00005 Streamwise vortex-induced and galloping-like vibrations of a rotating cylinder, REMI BOURGUET, DAVID LO JACONO, IMFT / CNRS — The flow-induced vibrations of an elastically mounted circular cylinder, free to oscillate in the direction parallel to the current and subjected to a forced rotation about its axis, are investigated numerically at a Reynolds number equal to 100. The cylinder is found to oscillate up to a rotation rate close to 2 (first vibration region), then the body and the flow are steady until a rotation rate close to 2.7, where a second vibration region begins. Each vibration region is characterized by a specific regime of response. In the first region, the oscillation amplitude follows a bell-shaped evolution as a function of the reduced velocity (inverse of the natural frequency) and the vibration develops under a condition of wake-body synchronization: such behavior resembles the vortex-induced vibrations previously described in the absence of rotation. In the second region, the vibration amplitude increases unboundedly with the reduced velocity and may become very large, higher than 2.5 body diameters in the present parameter space. Such galloping-like responses were not observed when the body was restrained to oscillate in the cross-flow direction. They cannot be predicted through quasi-steady analysis and it is found that body oscillation and flow unsteadiness remain synchronized.

5:10PM L14.00006 One- versus two-degree-of-freedom vortex-induced vibrations of a circular cylinder at Re=3900, SIMON GSELL, REMI BOURGUET, MARIANNA BRAZA, Institut de M´ecanique des Fluides de Toulouse, UMR 5502 CNRS-INPT-UPS — The response of an elastically-mounted circular cylinder, immersed in a current and free to move either in the streamwise or cross-flow direction, or in both directions, is predicted by means of direct numerical simulation. The Reynolds number based on the inflow velocity and the cylinder diameter is kept equal to 3900. Each configuration is studied over a range of the reduced velocity (inverse of the oscillator natural frequency) encompassing the entire region of lock-in, i.e. where body motion and flow unsteadiness are synchronized. The impact of an additional degree of freedom on the body response and fluid loading is analyzed. Particular attention is paid to the synchronization between the streamwise and cross-flow oscillations, their frequency ratio and phase difference, and to the frequency content of the fluid forces, including the occurrence of large higher harmonic contributions. The reciprocal influence of flow patterns and body motion is investigated on the basis of three-dimensional visualizations of the wake; the formation and modulation of the shear layer vortices is explored in this context.

5:23PM L14.00007 Flow-Induced Vibration of a Reed in a Channel: Effect of Reed Shape on Convective Heat Transfer with Application to Electronic Cooling, AARON RIPS, KOUSHRO SHOELE, Johns Hopkins University, ARI GLEZER, Georgia Institute of Technology, RAJAT MITTAL, Johns Hopkins University — Flow-induced vibration of a reed (a thin plate or flag) in a channel can improve heat transfer efficiency in forced convection applications, allowing for more heat transfer for the same fan power. Such systems have wide ranging applications in electronic and power cooling. We investigate the effect of 3D reed shape on heat transfer enhancement. To study 3D effects, we first use 2D fluid-structure interaction (FSI) simulations of an optimized reed (in terms of mass and stiffness) to generate a prescribed reed motion. We then apply that motion to a pseudo 3D reed (i.e. infinitely stiff in the spanwise direction) and study the heat transfer enhancement in a 3D channel. This method allows us to explore a large parameter space exhaustively, and using this method, we examine the effect of several parameters, such as reed planform and spanwise gap, on the heat transfer enhancements for forced convection in a channel. Simulations indicate that these geometrical feature have a significant effect on the vortex dynamics in the wake as well as the heat transfer efficiency.

1This work was supported by grants from AFOSR, EPRI and NSF.
5:36PM L14.00008 Generalized “thick” strip modelling for vortex-induced vibration of long flexible cylinders1. YAN BAO, RAFAEL PALOCIOS, SPENCER SHERWIN, Imperial College London, NEKTAR++ COLLABORATION — We propose a generalized strip modelling method that is computationally efficient for the VIV prediction of long flexible cylinders in three-dimensional incompressible flow. In order to overcome the shortcomings of conventional strip theory-based 2D models, the fluid domain is divided into “thick” strips, which are sufficiently thick to locally resolve the small scale turbulence effects and three dimensionality of the flow around the cylinder. An attractive feature of the model is that we independently construct a three-dimensional scale resolving model for individual strips, which have local spanwise scale along the cylinder’s axial direction and are only coupled through the structural model of the cylinder. Therefore, this model is able to cover the situations of fully resolved 3D model and 2D strip theory model. The connection between these strips is achieved through the calculation of a tensioned beam equation, which is used to represent the dynamics of the flexible body. In the limit, however, a single “thick” strip would request the full 3D domain. A parallel Fourier spectral/hp element method is employed to solve the 3D flow dynamics in the strip-domain, and then the VIV response prediction is achieved through the strip-structure interactions.

1This work is supported by EPSRC grant EP/K037536/1. Acknowledge UK Turbulence Consortium(UKTC) for ARCHER time under EPSRC grant EP/L000261/1.

5:49PM L14.00009 Large-eddy simulations of a flexible cylinder in axial flow. BEHROUZ KARAMI, ELIAS BALARAS, PHILIPPE BARDET, The George Washington University — A slender cylinder immersed in axial flow shows different behavior for different flow and material properties. Several studies have pointed to the importance of the dimensionless velocity, \(U = \left( \frac{\rho A E I}{\mu} \right)^{0.5} U_o D\), relating the fluid and structural inertia. However, it is not clear how this behavior changes for different Reynolds numbers and flow regimes, while keeping \(U\) constant. In this study a slender cylinder immersed in axial flow is considered as an one-dimensional beam. The fluid-structure interaction is simulated using an immersed-boundary method for a series of Re numbers. A non-linear Euler-Bernoulli hypothesis is utilized to account for the three-dimensional flexural vibration of the cylinder. It is observed that for small dimensionless velocities the cylinder oscillates with small amplitude around its axis. Increasing \(U\) results in buckling of the cylinder. For higher \(U\) beam looses its quasi steady buckled state and flutters. It is investigated that how this behavior changes for different Re and different flow regimes (laminar vs turbulent boundary layers). Overall buckling occurs at higher \(U\) at laminar flow conditions. The results are in agreement both qualitatively and quantitatively with experiments in the literature.

6:02PM L14.00010 Aeroelastic Flutter Behavior of Cantilevered within a Nozzle-Diffuser Geometry. LUIS PHILIPPE TOSI, TIM COLONIUS, Caltech, STEWART SHERRITT, HYEONG JAE LEE, Jet Propulsion Laboratory — Aeroelastic flutter arises when the motion of a structure and its surrounding flowing fluid are coupled in a constructive manner, causing large amplitudes of vibration in the immersed solid. A cantilevered beam in axial flow within a nozzle-diffuser geometry exhibits exciting resonance behavior that presents good prospects for internal flow energy harvesting. Different modes can be excited as a function of throat velocity, nozzle geometry, fluid and cantilever material parameters. This work explores the relationship between the aeroelastic flutter instability boundaries and relevant non-dimensional parameters via experiments. Results suggest that for a linear expansion diffuser geometry, a non-dimensional stiffness, non-dimensional mass, and non-dimensional throat size are the critical parameters in mapping the instability. This map can serve as a guide to future work concerning possible electrical output and failure prediction in energy harvesters.

6:15PM L14.00011 Flapping Instability of Two Tandem Flexible Foils in Uniform Axial Flow. PARDHA SARADHI GURUGUBELLI, RAJEV KUMAR JAIMAN, CASSEY CHUA, National University of Singapore — We present a numerical analysis on the stability and coupled dynamics of two tandem flexible foils clamped at their leading edges in a uniform axial flow. The flexible foils considered for this study correspond to the fixed-point stable regime of the single flexible foil where the flexible foil aligns itself in the flow direction with no significant trailing edge oscillations. A high-order nonlinear coupled solver based on the variational formulation has been considered for analyzing the effects of gap between the foils on the stability and coupled behaviour of both the upstream and downstream foils. As a function of the gap, it is observed that the tandem foil configuration is more prone to flapping instability than its single flexible foil counterpart. The evolution of the instability for the downstream foil shows two distinct dynamical scenarios: (i) only the downstream foil exhibits flapping motion and (ii) both the upstream and the downstream foils perform flapping. With the aid of a rigid foil in the upstream of a flexible foil, we further present a detailed analysis on the effects of the upstream wake and vortex shedding on the stability and flapping dynamics of the downstream foil.

6:28PM L14.00012 Small-Scale Vortical Motions induced by Aerodynamically Fluttering Reed for Enhanced Heat Transfer in a Rectangular Channel. SOURABH JHA, PABLO HIDALGO, ARI GLEZER, Georgia Inst of Tech — Small-scale vortical motions effected by an aerodynamically fluttering thin reed cantilevered across the span of a rectangular channel are exploited for heat transfer enhancement at transitional Reynolds numbers. The reed’s convex/concave surface undulations lead to the time-periodic formation, advection, and shedding of vorticity concentrations that scale with the motion amplitude. The reed motion is captured using phase-locked imaging and its interactions with the core flow and surface boundary layers are investigated using high-resolution PIV. Phase-averaged distributions of the reed’s mechanical energy demonstrate variations of the vibration modes across the channel. The reed’s impact on the surface is accompanied by transitory vortex shedding coupled with a local increase in the turbulent kinetic energy that results in a strong increase in heat transfer. The reciprocal interactions between the reed dynamics and the channel flow are captured using cross stream velocity distributions along the channel (\(L/W = 50\)) that link the kinetic energy shape factor to the rise in heat transfer (e.g., \(Nu\)) relative to the base flow. It is shown that the reed-induced heat transfer increases with Re and results in significant improvement in the global coefficient of performance.

1Supported by AFOSR.
4:05 PM L15.00001 Vorticity dynamics in the interaction of a single bubble with a vortex ring. RAGHURAMAN GOVARDHAN, NARSING JHA, Indian Institute of Science — Bubbly turbulent flows occur in a number of engineering applications, such as in drag reduction using bubbles. In the present work, we study an idealization of this problem, namely, the interaction of a single bubble with a single vortex ring. The vortex ring is generated in water using a piston-cylinder arrangement, and an air bubble in injected close to it. The changes in vorticity during the interaction are measured using time-resolved PIV, while the bubble dynamics are visualized using high speed imaging. Interactions are studied over a large range of Weber numbers, which is defined using the vortex ring strength. The results show that the interactions can significantly affect the vortex ring, including reduction in its convection speed, and fragmentation of its core with a resultant large decrease in its enstrophy. We present vorticity fields during the interaction over a range of Weber numbers to help understand the physics of the interaction. The present results for the bubble ring interactions show many phenomena also seen in bubbly turbulent flows such as reduction in enstrophy, which suggests that results from the present study may help to better understand interaction of bubbles with vortical structures in turbulent flows.

4:18 PM L15.00002 PIV investigation of the intake flow in a parallel valves diesel engine cylinder. P. HENRIK ALFREDSSON, JEAN RABAULT, JULIE A. VERNET, Kungliga Tekniska Högskolan KTH, BJÖRN LINDGREN, Scania CV AB — The flow of air (gas) inside the cylinder of internal combustion engines prior to compression may have a large influence on the combustion process. The structure of the in-cylinder flow, which can be swirl or tumble dominated, is to a large extent controlled by the design of the intake ports. In this study the admission flow generated by a parallel valves diesel engine cylinder head was investigated in a steady flow test bench through planar and stereo PIV measurements in both the swirl and tumble planes. By combining several sets of measurements a full three-dimensional, three-component reconstruction of the mean flow field was made. The flow out of the valves has a radial jet character, making the air hit the cylinder wall before flowing down along the cylinder wall. This leads to the formation of a recirculation bubble in the tumble plane. In the swirl plane complex jet dominated structures are found just below the valves giving rise to a counter-rotating vortex pair, where the strongest vortex becomes predominant giving rise to a single coherent swirling structure away from the cylinder head. Variations of the location and strength of the swirling structure may give rise to cycle-to-cycle variations and its stability was analysed by tracking the vortex centre.

1Supported by SSF, Swedish Foundation for Strategic Research and Scania CV AB

4:31 PM L15.00003 Pressure fluctuations and time scales in turbulent channel flow. KAMTHON SEPTHAM, JONATHAN MORRISON, SOURABH DIWAN, Imperial College London — Pressure fluctuations in turbulent channel flow subject to globally stabilising linear feedback control are investigated at \( Re_x = 400 \). The passivity-based control is adopted and explained by the conservative characteristics of the nonlinear terms contributing to the Reynolds-Orr equation (Sharma et al. Phys. Fluids 2011). The linear control operates via \( U'' \); the maximum forcing is located at \( y^+ \approx 20 \), corresponding to the location of the maximum in the mean-square pressure gradient. The responses of the rapid (linear) and slow (nonlinear) pressure fluctuations to the linear control are investigated using the Green’s function representations. It demonstrates that the linear control operates via the linear source terms of the Poisson equation for pressure fluctuations. Landahl’s timescales of the minimal flow unit (MFU) in turbulent channel flow are examined at \( y^+ = 20 \). It shows that the timescales of MFU agree well with the theoretical values proposed by Landahl (1993). Therefore, the effectiveness of the linear control to attenuate wall turbulence is explained by Landahl’s theory for timescales, in that the control proceeds via the shear interaction timescale which is significantly shorter than both the nonlinear and viscous timescales.

4:44 PM L15.00004 The effect of butterfly-scale inspired patterning on leading-edge vortex growth. JACOB WILROY, AMY LANG, Univ of Alabama - Tuscaloosa — Leading edge vortices (LEVs) are important for generating thrust and lift in flapping flight, and the surface patterning (scales) on butterfly wings is hypothesized to play a role in the vortex formation of the LEV. To simplify this complex flow problem, an experiment was designed to focus on the alteration of 2-D vortex development with a variation in surface patterning. Specifically, the secondary vorticity generated by the LEV interacting at the patterned surface was studied and the subsequent affect on the growth rate of the circulation in the LEV. For this experiment we used butterfly inspired grooves attached to a flat plate and compared the vortex formation to a smooth plate case as the plate moved vertically. The plate is impulsively started in quiescent water and flow fields at \( Re = 1500, 3000, \text{ and } 6000 \) are examined using Digital Particle Image Velocimetry (DPIV). The vortex formation time is 3.0 and is based on the flat plate travel length and chord length.

1We would like to thank the National Science Foundation REU Site Award 1358991 for funding this research.

4:57 PM L15.00005 Flow around new wind fence with multi-scale fractal structure in an atmospheric boundary layer. SARAH MCCLURE, Cleveland State University, SANG-JOON LEE, Pohang University of Science and Technology, WEI ZHANG, Cleveland State University — Understanding and controlling atmospheric boundary-layer flows with engineered structures, such as porous wind fences or windbreaks, has been of great interest to the fluid mechanics and wind engineering community. Previous studies found that the regular mono-scale grid fence of 50% porosity and a bottom gap of 10% of the fence height are considered to be optimal over a flat surface. Significant differences in turbulent flow structure have recently been noted behind multi-scale fractal wind fences, even with the same porosity. In this study, wind-tunnel tests on the turbulent flow and the turbulence kinetic energy transport of 1D and 2D multi-scale fractal fences under atmospheric boundary-layer were conducted. Velocity fields around the fractal fences were systematically measured using Particle Image Velocimetry to uncover effects of key parameters on turbulent flows around the fences at a Reynolds number of approximately 3.6x10^4 based on the free-stream speed and fence height. The turbulent flow structures induced by specific 1D/2D multi-scale fractal wind fences were compared to those of a conventional grid fence. The present results would contribute to the design of new-generation wind fences to reduce snow/sand deposition on critical infrastructure such as roads and bridges.
5:10PM L15.00006 Energetically efficient Proportional-Integral control of flow past a circular cylinder\textsuperscript{1} , PRAMODE KESAVADAS, VIJAY ANAND, B.S.V. PATNAIK, A.J. SHAHUJ, Indian Institute of Technology Madras, Chennai — In this numerical study, we present an energetically efficient Proportional (P) and Integral (I) control strategy for the cessation of vortex shedding behind a circular cylinder. Reflectionally symmetric controllers are designed such that, they are located on a small sector of the cylinder over which, tangential sliding mode control is imparted. Energetically efficient optimal parameters for the P, I and PI controls have been numerically assessed. An estimation of the time-averaged kinetic energy of different flow regimes using Proper Orthogonal Decomposition (POD) is also carried out. These values are obtained with and without the optimal controllers. The Navier-Stokes equations along with an evolution equation for the PI controller, is numerically solved using finite volume method. The optimization procedure is formulated as a standard Linear Quadratic (LQ) problem and the time-averaged kinetic energy is obtained by summation of POD eigenvalues. The energetic efficiency for the, I controller was observed to be superior compared to the other two classes of controllers. By performing detailed fluid flow simulations, it was observed that, the system is energetically efficient, even when the twin eddies are still persisting behind the circular cylinder.

\textsuperscript{1}The first author wishes to acknowledge the Ministry of Human Resource Development, Govt. of India.

5:23PM L15.00007 Identification of secondary instabilities in the near wake of a blunt trailing edge profiled body\textsuperscript{2} , ROSS CRUIKSHANK, WENYI ZHAO, PHILIPPE LAVOIE, University of Toronto — Aerodynamic research into blunt trailing edge (BTE) airfoils is driven by their structural and aerodynamic advantages over sharp trailing edge airfoils. However, the wake of BTE airfoils is dominated by a vortex street, which causes increased drag. One method to reduce the spanwise coherence of the vortex street is to generate streamwise vorticity in the wake. Recent evidence suggests that the efficiency of this control method can be improved by forcing at the same wavelength as a secondary instability (SI) of the vortex street, present at Reynolds numbers (based on airfoil thickness, $d$) above 470. The objective of the present study was to investigate the variation of the SI wavelength at $2000 < Re_d < 35,000$, and to examine the effect of forcing on the wake topology. The velocity field in the wake of a BTE profiled model was measured using particle image velocimetry, and proper orthogonal decomposition was applied as a filter for measurement noise. It was found that, for a laminar boundary layer, the SI wavelength decreased as $Re_d$ increased. Following boundary layer transition to turbulence, the SI wavelength was insensitive to $Re_d$. This study will also examine the effect of forcing at different wavelengths on the dominant spanwise wavelength of the wake velocity field.

\textsuperscript{2}The authors gratefully acknowledge the support of the Natural Sciences and Engineering Research Council of Canada.

5:23PM L15.00007 Identification of secondary instabilities in the near wake of a blunt trailing edge profiled body\textsuperscript{2} , ROSS CRUIKSHANK, WENYI ZHAO, PHILIPPE LAVOIE, University of Toronto — Aerodynamic research into blunt trailing edge (BTE) airfoils is driven by their structural and aerodynamic advantages over sharp trailing edge airfoils. However, the wake of BTE airfoils is dominated by a vortex street, which causes increased drag. One method to reduce the spanwise coherence of the vortex street is to generate streamwise vorticity in the wake. Recent evidence suggests that the efficiency of this control method can be improved by forcing at the same wavelength as a secondary instability (SI) of the vortex street, present at Reynolds numbers (based on airfoil thickness, $d$) above 470. The objective of the present study was to investigate the variation of the SI wavelength at $2000 < Re_d < 35,000$, and to examine the effect of forcing on the wake topology. The velocity field in the wake of a BTE profiled model was measured using particle image velocimetry, and proper orthogonal decomposition was applied as a filter for measurement noise. It was found that, for a laminar boundary layer, the SI wavelength decreased as $Re_d$ increased. Following boundary layer transition to turbulence, the SI wavelength was insensitive to $Re_d$. This study will also examine the effect of forcing at different wavelengths on the dominant spanwise wavelength of the wake velocity field.

5:36PM L15.00008 Feedback Control of Bistability in the Turbulent Wake of an Ahmed Body , ROWAN BRACKSTON, ANDREW WYNN, JUAN MARCOS GARCIA DE LA CRUZ, Imperial College London, GEORGIOS RIGAS, University of Cambridge, JONATHAN MORRISON, Imperial College London — Three-dimensional bluff body wakes have seen considerable interest in recent years, not least because of their relevance to road vehicles. A key feature of these wakes is spatial symmetry breaking, reminiscent of the large scale structures observed during the laminar and transitional regimes. For the flat backed Ahmed body, this feature manifests itself as a bistability of the wake in which the flow switches randomly between two asymmetric states. This feature is associated with instantaneous lateral forces on the body as well as increased pressure drag. Starting from the modelling approach of Rigas et al. (J. Fluid Mech. 778, R2, 2015) we identify a linearised model for this mode of the flow, obtaining parameters via a system identification. The identified model is then used to design a linear feedback controller with the aim of restoring the flow to the unstable, symmetric state. The controller is implemented experimentally at $Re \sim 3 \times 10^5$ and is found to both suppress the bistability of the flow and reduce the drag on the body. Furthermore, the control system is found to have a positive energy balance, providing a key demonstration of efficient feedback control applied to a 3D bluff body at Reynolds numbers representative of road vehicle wakes.

5:49PM L15.00009 A statistical approach characterizing the effectiveness of flow control on a dynamically pitching airfoil , KEITH TAYLOR, MICHAEL AMITAY, Rensselaer Polytech Inst — The presence of dynamic stall on wind turbines complicates the goal of energy production, as variations in input loading runs counter to the end goal of producing continuous level power output from a wind turbine. While dynamic stall has been extensively studied experimentally and computationally, the control of dynamic stall through active flow control is still a nascent field of research. In order to understand the flow field around a dynamically pitching finite span airfoil, a new method of characterizing the effectiveness of flow control in a statistical sense is presented. This method leverages the gamma one criterion on Particle Image Velocimetry images to identify the vortices shed, then statistically describes how the distribution of the circulation strength of identified vortices changes during dynamic stall. This is in contrast to previous work, which only addressed the phase averaged flow field, which does not fully illustrate how the flow field varies loop by loop, as there is significant variation between phase averaged flow fields and instantaneous flow fields measured. The purpose of this work is to present a new method of characterizing the effectiveness of control under dynamic conditions, without the need to capture PIV at high frequencies.

6:02PM L15.00010 Characterization of base pressure fluctuations in a blunt trailing edge wake with three-dimensional forcing , HEATHER CLARK, PHILIPPE LAVOIE, University of Toronto — The wakes of many nominally two-dimensional bluff bodies exhibit multiple intrinsic three-dimensional instabilities whose spatiotemporal structure and growth rate depend on geometry and Reynolds number. Here, these features are investigated experimentally for a blunt trailing edge profiled body using simultaneous measurements of velocity and fluctuating surface pressure on the model rear face near separation. Passive three-dimensional forcing of the wake is implemented with an array of vortex generators that are distributed according to the characteristic spanwise wavelength of the dominant secondary instability. For a Reynolds number of 8000 based on model thickness, the control strategy is found to increase the base pressure coefficient by 26% while globally reducing the amplitude of base pressure fluctuations, relative to the unforced flow. Additionally, amplitude modulation of the pressure signals that is observed in the natural wake decreases in strength with distributed forcing as a result of the modified three-dimensional flow structure. The spanwise distribution of pressure will be further examined for the baseline and controlled flows via temporal spectral analysis and spatial modal decomposition.
6:15PM L15.00011 On vortex pairing in several free shear layer containing high Reynolds number flows 1, MO SAMIMY, MICHAEL CRAWLEY, The Ohio State University — There are several free shear flows with well known Kelvin-Helmholtz instability, which contain an additional instability mechanism. For example, a jet has shear layer and jet column instabilities, a stalled airfoil has shear layer and wake instabilities, and a cavity flow has a shear layer instability and Rossiter modes. The shear layer’s most amplified frequency is normally several times larger than that of the other instability. Typically, the structures associated with the lower frequency instability are observed in the experiments. There is no much information in the literature, especially in high Reynolds number flows, on whether these structures are generated directly or by multiple merging of smaller structures generated by the shear layer instability. Single or multiple merging has been shown in the literature in only low Reynolds number flows (e.g. in jets). Our recent experimental results in high Reynolds number flows excited by plasma actuators seem to show the occurrence of multiple merging events before the observation of lower frequency large-scale coherent structures. The experimental PIV images obtained in jets using reconstructed flow and in stalled airfoils obtained using phase averaging.

6:28PM L15.00012 Interaction of Suction and Pulsed Blowing with a Laminar Boundary Layer, AVRAHAM SEIFERT, Tel Aviv Univ, LIAD MAROM, Tel Aviv Univ. — The presentation will describe a fundamental study of active flow control (AFC) using the steady suction and oscillatory blowing actuator (SaOB), identifying its effects on a laminar boundary layer. Recent experiments showed this effective and efficient actuator as a drag reduction device (e.g., Wilson et al, AIAA J, 2013). However, improved fundamental understanding of the boundary layer (BL) interaction with suction and oscillatory blowing and the combination of these two effects in close proximity is desired. The current experiment, performed in a laminar flow, will result in improved efficiency of the actuator and will enable development of a reliable predictive capability of this flow control method. The interaction with a laminar BL is crucial for the project due to the lack of interaction with the random turbulence, the thicker BL and lower skin-friction that enables greater effect of the controlled BL. Furthermore, fundamental interaction principles could be easier to identify and understand in laminar flows, where critical trends will not be masked by turbulence, and the averaging process will better represent the time dependent flow. The results demonstrate that while the oscillatory blowing is robust and has a strong effect on the flow evolution, the steady suction introduced upstream has a crucial role in the efficient operation of the AFC system.

Monday, November 23, 2015 4:05PM - 6:41PM  
Session L16 Aerodynamics: Unsteady Aerodynamics I 204 - Douglas Bohl, Clarkson University

4:05PM L16.00001 Experimental Investigation of Dynamic Stall on a NACA0012 Airfoil Undergoing Sinusoidal Pitching 1, DOUGLAS BOHL, Clarkson University, MELISSA GREEN, Syracuse University — In this work, the flow field around a NACA0012 Airfoil undergoing large amplitude sinusoidal pitching is investigated using Particle Image Velocimetry (PIV). The airfoil is pitched symmetrically about the quarter chord point with a peak angle of 20 deg, at reduced frequencies of k=0.2-0.6 and Re=12000. Sixteen different Fields of View are phase averaged and combined to quantify the flow field from 0.75c upstream of the leading edge to 1c downstream of the trailing edge. This provides spatially and temporally resolved data sets that include the downstream evolution of the flow fields. The velocity and vorticity fields, both around the airfoil and downstream of the trailing edge, will be investigated as a function of the reduced frequency to better understand the dynamics (i.e. formation, separation and development) of the leading edge vortex and the resulting downstream flow evolution.

1This work was supported by the Office of Naval Research under ONR Award No. N00014-14-1-0418

4:18PM L16.00002 Experimental Investigation of Dynamic Stall on a Finite Span NACA 0012 Wing 1, WYATT SPELLMAN, DOUGLAS BOHL, Clarkson University — In this work the velocity and vorticity fields around finite and “infinite” span wings with a NACA 0012 profile undergoing constant rate pitching were quantified using Molecular Tagging Velocimetry (MTV). The “infinite” span wing was bounded by walls to reduce the effects of the wing tip vortex while the finite span wing was bounded by a wall on one end and unbounded on the other. The wings were pitched from α = 0 to 55° with a constant non-dimensional pitch rate of 0°/s at Re = 12000. The Dynamic Stall Vortex (DSV) was identified and tracked using the “I” criteria. The results showed that the formation and trajectory of the DSV for the finite wing case varied with the spanwise location, with the location of the DSV remaining progressively closer to the airfoil surface towards the wingtip. These results were consistent with the “Omega” vortex structure previously observed in flow visualization. The DSV was also found to remain closer, and convect away from the airfoil surface slower, at all spanwise measurement planes when compared to the infinite span results.

1This work was supported by NSF Grant #845882.

4:31PM L16.00003 Analysis of the aerodynamic interaction between two plunging plates in tandem at low Reynolds number for maximum propulsive efficiency 1, JOAQUIN ORTEGA-CASANOVA, RAMON FERNANDEZ-FERIA, Universidad de Malaga (Spain) — The thrust generated by two heaving plates in tandem is analysed for two particular sets of configurations of interest in forward flight: a plunging leading plate with the trailing plate at rest, and the two plates heaving with the same frequency and amplitude, but varying the phase difference. The thrust efficiency of the leading plate is augmented in relation to a single plate heaving with the same frequency and amplitude in most cases. In the first configuration, we characterize the range of nondimensional heaving frequencies and amplitudes of the leading plate for which the stationary trailing plate contributes positively to the global thrust. The maximum global thrust efficiency, reached for an advance ratio slightly less than unity and a reduced frequency close to 5, is about the same as the maximum efficiency for an isolated plate. But for low frequencies the tandem configuration with the trailing plate at rest is more thrust efficient than the isolated plate. In the second configuration, we find that the maximum thrust efficiency is reached for a phase lag of 180 (counterstroking), particularly for an advance ratio unity and a reduced frequency 4.4, and it is practically the same as in the other configuration and that for a single plate.

1Supported by the Ministerio de Economía y Competitividad of Spain Grant no. DPI2013-40479-P.
4:44PM L16.00004 Experimental investigation of a large aspect ratio flat plate encountering a stea... - ANYA JONES, University of Maryland — While humans are capable of mimicking, and even outperform, the kinematic capabilities of natural flyers, birds and insects are still way ahead of us when it comes to anticipating and dealing with turbulent and gusty flow conditions. To tailor and improve flight control capabilities of low Reynolds number flyers in real weather, we need to bridge this gap of knowledge. As a first step, we experimentally studied the aerodynamic influence of a simplified stream-wise gust on a large aspect ratio flat plate. The experiments were conducted in the $7 \times 1.5 \times 1m^3$ towing tank at UMD which was equipped with a 4-axis computer-controlled motion system. The effect of a stream-wise gust was simulated by accelerating or decelerating the wing to a new constant velocity after an initial constant surge. A high-speed camera and light sheet optics were attached to the tow carriage allowing for time-resolved particle image velocimetry along the entire motion in addition to direct force measurements. A proper orthogonal decomposition of the flow field was carried out to study the time scales related to changes induced by the sudden acceleration or deceleration in addition to analyzing the size, position and trajectory of prominent vortices and associated forces during the gust encounter.

5:00PM L16.00005 Investigation into the Recovery of a Translating Flat Plate Exposed to a Streamwise Acceleration — PETER MANCINI, Univ of Maryland-College Park, KAREN MULLENERS, EPFL, ANYA JONES, Univ of Maryland-College Park — This study explores the unsteady aerodynamic response of a wing to streamwise accelerations. A wing was towed through a water tank until reaching steady state (30 chords), after which the wing accelerated over a prescribed distance to a new constant velocity. Several velocity profiles were investigated, including acceleration and deceleration, as well as various angles of attack ($0^\circ$ - $50^\circ$). Direct force measurements and particle image velocimetry were conducted simultaneously throughout the full length of the motion. This was made possible through the implementation of a unique imaging setup that allows the wing, laser sheet, and camera to move together rigidly down the length of the tank, placing all PIV measurements in the wing-fixed reference frame and allowing for an uninterrupted measurement of the entire wing motion. Lift force results showed that for cases of high leading edge flow separation ($\alpha > 20^\circ$), the distance required to reach steady state is drastically lower when recovering from a streamwise acceleration than from accelerating from rest. The concept of vortex formation time was also explored, and via PIV and force results it was confirmed that the formation number consistently lies within the range of 3.6-4.5 for each acceleration profile.

5:10PM L16.00006 The Direct Numerical Simulation of the Deflected Wake Phenomenon around a Plunging NACA0012 Airfoil at Low Reynolds Numbers — MEHMET SAHIN, SALIHA BANU YUCE, Istanbul Technical University, MEHMET FEVZI UNAL, MEF University — The deflected wake phenomenon reported by Jones and Platzer (2009) is investigated in detail using direct numerical simulations around a NACA0012 airfoil undergoing harmonic plunging motion. An Arbitrary Lagrangian-Eulerian (ALE) formulation based on unstructured side-centered finite volume method is utilized in order to simulate the unsteady twodimensional unsteady incompressible flow around the NACA0012 airfoil at a reduced frequency of $\pi f c / U_{\infty}$ and the plunge amplitude non-dimensionalized with respect to chord are set to 0.12, 0.2, and 0.3. The effect of plunge on the formation of vortices in plunge motion ($k = 2\pi f c / U_{\infty}$) and the plunge amplitude non-dimensionalized with respect to chord are set to 12.3 and 0.12, respectively, as in the experimental study of Jones and Platzer (2009). The present numerical simulations reveal a highly persistent transient effect and it takes two orders of magnitude larger than the heave period to reach the time-periodic state. In addition, the three-dimensional simulation reveals that the flow field is highly three-dimensional around the leading edge. The calculation reproduces the deflected wake and shows a very good agreement with the experimental results by Lyons, Fing. Time Lypnov Exponent (FTLE) fields and particle traces are presented along with the aerodynamic parameters including the lift and thrust coefficients.

5:23PM L16.00007 Wing Rock Motion and its Flow Mechanism over a Chined-Body Configuration — YANKUI WANG², QIAN LI², WEI SHI¹, Beijing University — Wing rock motion is one kind of uncommanded oscillation around the body axis over the most of the aircraft at high enough angle of attack and has a strong threat to the flight safety. The purpose of this paper is to investigate the wing rock motion over a typical body-wing configuration with a chined fuselage at fixed angle of attack firstly and four kinds of wing rock motion are revealed based on the flow phenomena, namely non-oscillation, lateral deflection, limit-cycle oscillation and irregular oscillation. Simultaneously, some special relationship between the wing rock motion and the flow over the chined body configuration is discussed. In addition, the evolution of wing rock motion and its corresponding modes when the model undergoes pitching up are also given out. All the experiments have been conducted in a low-speed wind tunnel at a Reynolds number of 1.87*10E5 and angle of attack from 0deg to 65deg.

1National Natural Science Foundation of China(11472028) and Open fund from State Key Laboratory of Aerodynamics
2Prof. Yankui Wang received a PhD degree in Fluid Mechanics from Beihang University and served as a full Professor from 2005. His current research interest is fluid mechanics and complex flow.
3Qian Li is currently a Ph.D. candidate of fluid mechanics in Beihang University. His main research interest is complex flow.
4Wei Shi is currently pursuing a Ph.D. degree in Beihang University. His main research interest is complex flow.

5:36PM L16.00008 Unsteady Aerodynamics of “Roll-Tacking” in Olympic Class Sailboats — RILEY SCHUTT, CHK WILLIAMSON, Cornell University — When tacking a sailboat (turning a boat through the wind during upwind sailing), racers employ a “roll-tacking” technique. During a roll-tack, sailors use body weight movements to roll the boat through extreme angles of heel. This contrasts with a flat-tack, where the boat remains upright throughout the turn. The dynamic heeling motion of a roll-tack causes the sail to vigorously sweep through the air, resulting in large-scale vortex shedding and increased propulsion. In this research, we use a characteristic roll-tack motion derived from on-the-water data. On-the-water data is collected from a full-scale Olympic racing boat sailed by a national champion in the Laser sailboat class. Using this data, we run a series of representative experiments in the laboratory. Two dimensional flexible sail extrusions are built using rapid-prototyping and are tested in a three degree-of-freedom (X, Y, and theta) towing tank. Particle Image Velocimetry and force measurements are used to compare vortex dynamics and propulsive forces generated by roll-tacks versus flat-tacks. An increase in thrust observed during roll-tack tests agrees with on-the-water experiments, which show a racing advantage greater than one boatlength when a roll-tack is performed relative to a flat tack.

5:49PM L16.00009 Aerodynamics of Unsteady Sailing Kinetics — COLIN KEIL, RILEY SCHUTT, JENNIFER BORSHOFF, PHILIP ALLEY, MAXIMILIEN DE ZEGHER, CHK WILLIAMSON, Cornell University — In small sailboats, the bodyweight of the sailor is proportionately large enough to induce significant unsteady motion of the boat and sail. Sailors use a variety of kinetic techniques to create sail dynamics which can provide an increment in thrust, thereby increasing the boatspeed. In this study, we experimentally investigate the unsteady aerodynamics associated with two techniques, “upwind leech flicking” and “downwind 5-turns”. We explore the dynamics of an Olympic class sailboat equipped with a GPS, IMU, wind sensor, and camera array, sailed expertly by a member of the US Olympic team. The velocity heading of a sailing boat is oriented at an apparent wind angle to the flow. In contrast to classic flapping propulsion, the heaving of the sail section is not perpendicular to the sail’s motion through the air. This leads to heave with components parallel and perpendicular to the incident flow. The characteristic motion is recreated in a towing tank where the vortex structures generated by a representative 2-D sail section are observed using Particle Image Velocimetry and the measurement of thrust and lift forces. Amongst other results, we show that the increase in driving force, generated due to heave, is larger for greater apparent wind angles.

Wei Shi is currently pursuing a Ph.D. degree in Beihang University. His main research interest is complex flow.

Qian Li is currently a Ph.D. candidate of fluid mechanics in Beihang University. His main research interest is complex flow.
6:02PM L16.00010 On leading-edge vortex attachment in rotary systems: Incident flow effects, ALBERT MEDINA, U.S. Air Force Research Laboratory, Wright-Patterson Air Force Base, ANYA R. JONES, University of Maryland, College Park — The mechanism governing the stall attachment of the leading-edge vortex (LEV) in rotating systems has been believed to be rooted in a balance between the rate vorticity production from the leading-edge shear layer and the convection of vorticity-bearing mass from within the LEV to the surrounding flow field. In such a relation, the accumulation of vorticity within a vortical structure is regulated by convective influences effectively draining the structure of circulatory strength. This work numerically investigates the shear rate-convection balance assertion in low-aspect ratio rectangular flat plates undergoing unidirectional rotation in a steady freestream. The freestream is oriented parallel to the rotational axis and the effect of advance ratio on the resulting flow structures is compared with a rotary plate operating in a quiescent fluid. Depending on advance ratio, the incidence angle of the plate is adjusted to maintain a constant effective attack angle of $\alpha = 45^\circ$ based on plate tip speeds. Of interest is the response of the system over a Reynolds number range $Re = [10^2 : 10^3]$ where axial flow prominence shifts from aft of the leading-edge vortex to within the structure.

6:15PM L16.00011 Effect of Trailing Edge Shape on the Unsteady Aerodynamics of Reverse Flow Dynamic Stall, ANDREW LIND, ANYA JONES, University of Maryland — This work considers dynamic stall in reverse flow, where flow travels over an oscillating airfoil from the geometric trailing edge towards the leading edge. An airfoil with a sharp geometric trailing edge causes early formation of a primary dynamic stall vortex since the sharp edge acts as the aerodynamic leading edge in reverse flow. The present work experimentally examines the potential merits of using an airfoil with a blunt geometric trailing edge to delay flow separation and dynamic stall vortex formation while undergoing oscillations in reverse flow. Time-resolved and phase-averaged flow fields and pressure distributions are compared for airfoils with different trailing edge shapes. Specifically, the evolution of unsteady flow features such as primary, secondary, and trailing edge vortices is examined. The influence of these flow features on the unsteady pressure distributions and integrated unsteady airloads provide insight on the torsional loading of rotor blades as they oscillate in reverse flow. The airfoil with a blunt trailing edge delays reverse flow dynamic stall, but this leads to greater downward-acting lift and pitching moment. These results are fundamental to alleviating vibrations of high-speed helicopters, where much of the rotor operates in reverse flow.

6:28PM L16.00012 Formation and Development of the Dynamic Stall Vortex on a Wing with Leading Edge Tubercles, JOHN HRYNUK, Army Research Lab - Vehicles Technology Directorate, DOUGLAS BOHL, Clarkson University — Humpback whales are unique in that their flippers have leading edge “bumps” or tubercles. Past work on airfoils inspired by whale tubercles has centered on the static aerodynamic characteristics of these airfoils. The current study uses Molecular Tagging Velocimetry (MTV) to investigate the effects of tubercles on dynamically pitching NACA 0012 airfoils. A baseline (i.e. straight leading edge) wing and one modified with leading edge tubercles are investigated. Tracking of the Dynamic Stall Vortex (DSV) is performed to quantitatively compare the DSV formation location, path, and convective velocity for tubercled and baseline wings. The results show that the spanwise variation in the initial formation location and motion of the DSV on the modified wing. Once formed, the DSV aligns into a more uniform spanwise structure. As the pitching motion progresses, the DSV on the modified wing convects away from the airfoil surface later and slower than is observed for the baseline airfoil. The results indicate that the tubercles may delay stall when compared to the baseline airfoil.

1This work was supported by NSF Grant # 0845882.

Monday, November 23, 2015 4:05PM - 6:41PM
Session L17 Flow Instability: Richtmyer-Meshkov III
205 - Juan Gustavo Wouchuk, Universidad de Castilla-La Mancha

4:05PM L17.00001 Analytical expressions for the asymptotic velocities in the linear Richtmyer-Meshkov instability when a shock is reflected, FRANCISCO COBOS, JUAN GUSTAVO WOUCHUK, E.T.S.I.Industriales-INEI-Cytema-Universidad de Castilla la Mancha-Ciudad Real-Spain — When a planar shock hits a corrugated contact surface between two fluids, hydrodynamic perturbations are generated in both fluids that result in asymptotic normal and tangential velocity perturbations in the linear stage. In this work, explicit and exact analytical expansions of the asymptotic velocities are presented for the general case in which a shock is reflected back. The expansions are derived from the conservation equations and takes into account the whole perturbation history between the transmitted and reflected fronts. The important physical limits of weak and strong shocks, high density ratio at the contact surface and very compressible/incompressible are studied. The expansions are compared with the exact solution obtained by iteration from a set of functional equations and the limit of validity of those expansions is discussed. An approximate expression for the normal velocity, valid even for strong shocks in some regimes, is given. This work is a continuation of the calculations shown in Phys. Rev. E 90, 053007 (2014) for a single shock moving into one fluid.

4:18PM L17.00002 Richtmyer-Meshkov mixing: Modeling and simulation of experiments, NICHOLAS DENISSEN, SUSAN KURIEN, Los Alamos National Laboratory — Hydrodynamic instabilities that result from the interaction of a shock-wave with a perturbed interface are known as Richtmyer-Meshkov instabilities (RMI). RMI is important in a wide variety of applications including Inertial Confinement Fusion. Recent experiments at Los Alamos National Laboratory (LANL) have focused on careful measurement of initial conditions and repeated statistical measurements of the instability growth and transition to turbulence. This talk will discuss ongoing efforts to model these experiments using weakly non-linear theoretical models, one dimensional Reynolds–Averaged Navier-Stokes models and three-dimensional Implicit Large Eddy Simulations (ILES). Analysis of the experimental data supplies the initial condition for the theoretical model and the ILES calculations. The effect of different initial conditions and mesh resolutions will be examined in light of interest in international collaboration on an RMI test problem. Comparison of the different models to experimental data will be presented. All calculations are performed in the arbitrary Lagrangian/Eulerian (ALE) code FLAG, developed at LANL. The ALE framework allows us to assess the effects of numerical diffusion on RMI computations by varying the remap strategy.

4:31PM L17.00003 Interactions of Blast Waves with Perturbed Interfaces, MARC HENRY DE FRAHAN, ERIC JOHNSON, Univ of Michigan — The Richtmyer-Meshkov and Rayleigh-Taylor instabilities induce hydrodynamic mixing in many important physical systems such as inertial confinement fusion, supernova collapse, and scramjet combustion. Blast waves interacting with perturbed interfaces are prevalent in such applications and dictate the mixing dynamics. This study increases our understanding of blast-driven hydrodynamic instabilities by providing models for the time-dependent perturbation growth and vorticity production mechanisms. The strength and length of the blast wave determine the different growth regimes and the importance of the Richtmyer-Meshkov or Rayleigh-Taylor growth. Our analysis is based on simulations of a 2D planar blast wave modeled by a shock (instantaneous acceleration) followed by a rarefaction (time-dependent deceleration), interacting with a sinusoidal perturbation at an interface between two fluids. A high-order accurate Discontinuous Galerkin method is used to solve the multifluid Euler equations.
4:44PM L17.00004 Evolution of the air/SF6 turbulent mixing zone for different lengths of SF6: shock tube visualizations and 3D simulations. JEAN-FRANCOIS HAAS, JEROME GRIFFON, DENIS SCHOFLAND, CEA DAM IDF, GHAZI BOUZGAROUN, YANNICK BURY, STEPHANE JAMME, ISAE DAEP — A turbulent mixing zone (TMZ) is created in a vertical shock tube (based in ISAE DAEP) when a Mach 1.2 shock wave in air accelerates impulsively to 70 m/s an air/SF6 interface. The gases are initially separated by a thin nitrocellulose membrane maintained flat and parallel to the shock by two wire grids. The upper grid (SF6 side) of square mesh spacing $h_u$ 1.8 or 12.1 mm is expected to seed perturbation for the Richtmyer-Meshkov instability (RMI) while the lower grid with $h_l$ 1 mm is needed to prevent the membrane from bulging prior to the shot. The experiments were carried out for different lengths L of SF6 between the initial interface and the shock tube’s end plate : 10, 15, 20, 25 and 30 cm. The time resolved Schlieren image processing based on space and frequency filtering yields similar evolution for the TMZ thickness. Before reshock, the thickness grows initially fast then slows down and reaches different values (10 to 14 mm) according to L. Soon after reshock, the TMZ thickness growths rate is 21 mm/ms independently of L and $h_u$. Numerical Schlieren images generated from 3D numerical simulations (performed at CEA DAM IDF) are analyzed as the experimental ones for L 15 and 25 cm and for $h_u$ 1.8 and 12.1 mm. The very weak experimental dependence on $h_u$ is not obtained by simulation as expected from dimensional reasoning. This discrepancy remains paradoxical.

4:57PM L17.00005 Initial condition spectral content effects on shock-driven turbulent mixing. FERNANDO GRINSTEIN, NICHOLAS NELSON, LANL — We report simulations of a shocked heavy band using the RAGE code in the implicit LES context [1]. We consider a shock-tube conuration with a band of high density SF6 gas embedded in low density air. A shock with Mach number 1.26 is transported through the band, resulting in transition to turbulence driven by the Richtmyer-Meshkov instability. The evolution of the system is followed as the primary shock traverses the SF6 band, reflects off the end-wall, propagates back and reshocks the mixing layers. We apply a variety of initial perturbations to the interfaces between the two uids in which the physical standard deviation, wave number range, and the spectral slope of the perturbations are held constant, but the number of modes initially present is varied. By decreasing the density of initial spectral modes of the interface, we nd that we can achieve as much as 25% less total mixing at late times. Analysis is based on the evolution of mixing widths, mixedness, turbulent kinetic energy, and effective Reynolds number estimates. [1] Phys. Rev. E92, 013014, 2015.

5:10PM L17.00006 Shock driven instability of a multi-phase particle-gas system. JACOB MCFARLAND, WOLFGANG BLACK, JEEVAN DAHAL, University of Missouri, BRANDON MORGAN, Lawrence Livermore National Laboratory — A computational study of a shock driven instability of a multiphase particle-gas system is presented. This instability can evolve in a similar fashion to the Richtmyer-Meshkov (RM) instability, but has addition parameters to be considered. Particle relaxation times, and density differences of the gas and particle-gas system can be adjusted to produce results which are different from the classical RM instability. We will show simulation results from the Ares code, developed at Lawrence Livermore National Laboratory, which uses a particle-in-cell approach to study the effects of the particle-gas system parameters. Mixing parameters will be presented to highlight the suppression of circulation and gas mixing by the phase particle.

5:23PM L17.00007 Freeze-out of the linear Richtmyer-Meshkov instability. JUAN GUSTAVO WOUCHUK, FRANCISCO COBOS CAMPOS, E.T.S.I.Industriales-INI-Cytema-UCLM-Ciudad Real-Spain, TAKAYOSHI SANO, Institute of Laser Engineering-Osaka University-Suita, Osaka 565-0871, Japan — When a planar shock refracts a corrugated contact surface separating two fluids with different thermodynamic properties, a transmitted and reected wavefront run inside each uid. Due to the surface ripple, pressure, density and velocity perturbations are generated in both materials. When the fronts separate away, a steady normal velocity develops at the contact surface. For speciic choices of the pre-shock parameters, the nal value of the normal velocity at the surface ripple may become zero. This eect, known as “freeze-out” has been proposed by G.Fraley [Phys. Fluids 29, 376 (1986)] and has been later on studied by K. Mikaelian [Phys. Fluids 6, 356 (1994)]. We present here an analytical study to freeze-out in both situations of shock and rarefaction reected at the contact surface. Freeze-out contours are derived as well as the detailed temporal evolution of the pressure and velocity perturbations in linear theory. Weak/strong incident shock limits are discussed. [1] J. G. Wouchuk and K. Nishihara, Phys. Rev. E 70, 026305 (2004). [2] J. G. Wouchuk and T. Sano, Phys. Rev. E 91, 023005 (2015).

5:36PM L17.00008 Effect of pressure field fluctuations on the nonlinear evolution of Richtmyer-Meshkov coherent structure. AKLANT BHOWMICK, SNEZHANA ABARZHI, Carnegie Mellon University — We consider the effect of pressure fluctuations on the evolution of Richtmyer-Meshkov (RM) flows. The pressure fluctuations are induced by non-uniformity in the fluid bulk and are modeled as a time dependent acceleration with the power-law exponent [-2]. We consider a large scale periodic coherent structure of bubbles and spikes in a two-dimensional RM flow, and obtain asymptotic solutions describing nonlinear dynamics of the structure using group theory analysis. We show that regular asymptotic solutions describing the bubble dynamics form a one-dimensional family. The family can be parametrized by the curvature of the bubble front. The stability of the family solutions is analyzed. The physically signicant solution in the family is interpreted as the stable solution with the maximum velocity. By decreasing the density of initial spectral modes of the interface, we nd that we can achieve as much as 25% less total mixing at late times. Analysis is based on the evolution of mixing widths, mixedness, turbulent kinetic energy, and effective Reynolds number estimates. [1] Phys. Rev. E92, 013014, 2015.

5:49PM L17.00009 Influence of interference of perturbation waves on the dynamics of Richtmyer-Meshkov flows. ARUN PANIDIAN, SNEZHANA ABARZHI, Carnegie Mellon University — We study the dynamics of structures that are formed due to Richtmyer-Meshkov instability (RMI) at the interface between two uidic with different densities when a strong shock wave refractions it [1]. While previous research in this area was focused on the eects of the wavelength and amplitude of the interface perturbation, the information was largely ignored on the inuences of the relative phase of a multi-wave perturbation and the interference of the perturbation waves on RMI evolution. Applying group theory analysis and Smooth Particle Hydrodynamics simulations, we study the eects of the relative phase of the interfacial sinusoidal waves on the structure of bubbles and spikes that is formed at the interface after the shock passage. A number of new qualitative and quantitative eects are found, and the eect of the wave interference on RMI evolution is observed. In particular, evidences so far indicate that the symmetry of the interface strongly inuences the spike morphology as compared to asymmetric cases. We discuss how one may control the growth of RM by controlling the phases of waves of the initial perturbation.
6:02PM L17.00010 Direct Numerical Simulation of Richtmeyer-Meshkov Instability Using pWAMR1. TEMISTOCLE GRENGA, SAMUEL PAOLUCCI, University of Notre Dame — The parallel Wavelet Adaptive Multiresolution Representation (pWAMR) method is used to simulate the Richtmeyer-Meshkov instability caused by a shock interacting with a density-stratified interface. The physical problem is studied in several configurations. We present results of numerical studies that investigate the influence of initial condition parameters (amplitude and wavelength of perturbations) on mixing and transition. In addition, the evaluation of turbulence statistics provides a measure of the mixing across the scales and the correlation with the initial condition parameters. The problem is modeled using the compressible reactive Navier-Stokes equations for a gas mixture, including multi-component diffusion, Soret and Dufour effects, and state dependent thermodynamic and transport properties. Since the amplitudes of wavelets provide a direct measure of the local error, the method is able to efficiently capture to any desired accuracy a wide range of spatial scales using a relatively small number of degrees of freedom by evolving the dynamically adaptive grid. In an effective fashion, the multilevel structure of the algorithm provides a simple way to adapt computational refinements to local demands of the solution, thus automatically producing verified solutions.

1Supported by C-SWARM through the Department of Energy, National Nuclear Security Administration, under Award Number DE-NA0002377.

6:15PM L17.00011 Measurements of the turbulent development of Richtmeyer-Meshkov instability VITALIY KRIVETS, EVEREST SEWELL, QIAN XU, JEFFREY JACOBS, University of Arizona — A vertical shock tube is used for experiments on the Richtmeyer-Meshkov instability in which a membrane-less interface is formed by opposed gas flows where the light and heavy gases enter the shock tube from the top and from the bottom of the driven section. An air/Ar gas combination is used and an $M = 1.2$ incident shock wave impulsively accelerates the interface. Initial perturbations are generated by harmonically oscillating the gases vertically, using two loudspeakers mounted in the shock tube walls, to produce Faraday resonance resulting in a random short wavelength perturbation. Planar Mie scattering is used to visualize the flow using a laser sheet to illuminate smoke particles seeded in one of the two gases. In addition, particle image velocimetry is used to obtain velocity measurements in which both gases are seeded. Image sequences are captured using high-speed video cameras. New experiments are presented quantifying the growth of the integral mixing layer width in addition to the molecular mixing evolution produced by the instability.

6:28PM L17.00012 Experimental Investigation of Velocity Evolution in the Richtmeyer-Meshkov Instability DANIEL REESE, JASON OAKLEY, DAVE ROTHAMER, RICCARDO BONAZZA, University of Wisconsin — The present work describes the evolution of the Richtmeyer-Meshkov instability through a focus on the development of velocity fluctuations. In the Wisconsin Shock Tube Laboratory at the University of Wisconsin, a broadband, shear-layer initial condition is created at the interface between helium and argon. This shear layer is seeded with particulate TiO$_2$, which is used to track the flow and allow for the Mie scattering of light. Once impulsively accelerated by a $M = 1.4$ shock wave, the interface is imaged twice in close succession using planar laser imaging to create particle image pairs. Velocity fields are obtained from these particle images using the Insight 4G software package from TSI. This process is repeated, capturing a total of four different times in the development of the instability, allowing for the study of velocity development in the RMI. For each post-shock time, velocity field structure is investigated, and probability density functions of velocity fluctuations are compared. Using known length scales from previous studies, these newfound RMS velocity values are also used to give an estimate of the Reynolds number.

Monday, November 23, 2015 4:05PM - 6:28PM
Session L18 Flow Instability: Interfacial and Thin Films III 206 - Ranga Narayanan, University of Florida

4:05PM L18.00001 Reactive thin film flows over spinning discs1. KUN ZHAO, ALEX WRAY, JUNFENG YANG, OMAR MATAR, Imperial College London — We consider the dynamics of a thin film flowing over a spinning disc in the presence of a chemical reaction, and associated heat and mass transfer. We use a boundary-layer approximation in conjunction with the Karman-Polhausen approximation for the velocity distribution in the film to derive a set of coupled one-dimensional evolution equations for the film thickness, radial and azimuthal flow rates, concentration of the reagents and products, and temperature. These highly nonlinear partial differential equations are solved numerically to reveal the formation of large-amplitude waves that travel from the disc inlet to its periphery. The influence of these waves on the concentration and temperature profiles is analysed for a wide range of system parameters: the Damkohler and Schmidt numbers, the thermal Peclet numbers, and the dimensionless disc radius (a surrogate for the Eckman number). It is shown that these waves lead to significant enhancement of the rates of heat and mass transfer associated with the reactive flow; these are measured by tracking the temporal evolution of local and spatially-averaged Nusselt and Sherwood numbers, respectively.

1EPSRC Programme Grant, MEMPHIS, EP/K0039761/1

4:18PM L18.00002 Thin-film coating of surfactant-laden liquids on rotating cylinders WEIHUA LI, SATISH KUMAR, University of Minnesota — Motivated by the need to improve fundamental understanding of the coating of discrete objects, the influence of surfactants on the flow of thin liquid films around rotating cylinders is considered in this work. The lubrication approximation is applied to derive the three coupled nonlinear evolution equations describing the variation of the film thickness, surfactant surface concentration, and surfactant bulk concentration as a function of time and the angular coordinate. In the absence of gravitational effects, linear stability analysis reveals that Marangoni stresses suppress the growth rate of instabilities driven by centrifugal forces and hinder the leveling of perturbations to the film thickness. When gravitational effects are present, Marangoni stresses lower the critical rotation rate needed to cause motion of a liquid lobe around the cylinder. These stresses also lead to faster damping of oscillations in the film thickness at relatively short times, but at longer times can increase the oscillation amplitude. In all cases examined, surfactant solubility has the effect of weakening the influence Marangoni stresses.
4:31PM L18.00003 Dynamics and stability of thin films and drops subjected to magnetic fields. DEVEN CONROY, ALEX WRAY, OMAR MATAR, Imperial College London — We consider the interfacial dynamics of a thin, ferrofluidic film flowing down an inclined substrate, under the action of a magnetic field, bound above by an inviscid gas. The fluid is assumed to be weakly-conducting. Its dynamics are governed by a coupled system of the steady Maxwell’s, the Navier-Stokes, and continuity equations. The magnetisation of the film is a function of the magnetic field, and is prescribed by a Langevin function. We make use of a long-wave reduction in order to solve for the dynamics of the pressure and velocity fields inside the film. The potential in the gas phase is solved via the use of Fourier Transforms. Imposition of appropriate interfacial conditions allows for the construction of an evolution equation for the interfacial shape via use of the kinematic condition. The magnetic effects give rise to a non-local contribution. We conduct a parametric study of the system stability to spanwise perturbations in order to evaluate the effects of the magnetic field. Two canonical configurations are considered: constant volume, and constant flux, corresponding to a thin drop and a thin film flowing down the incline.

1EPSRC Programme Grant, MEMPHIS, EP/K0039761/1, EPSRC DPF Studentship (AWW)

4:44PM L18.00004 Faraday instability in electrostatically forced liquid-air systems. KEVIN WARD, Univ. of Florida, FARZAM ZOUESHTIAGH, Institut de Electronique, de Microelectronique et de Nanotechnologie (IEMN), UMR CNRS 8520, University of Lille 1, SATOSHI MATSUMOTO, Institute of Space and Astronautical Science Japan Aerospace Exploration Agency, RANGA NARAYANAN, Univ. of Florida — When stacked multi-fluid systems are periodically accelerated in a direction normal to the fluid interfaces, complex patterns on the interfaces develop as a result of resonance between the imposed oscillation frequency and the natural frequency of the system, a phenomenon known as Faraday instability. Experimental research at JAXA has successfully generated Faraday instability in electrostatically oscillated liquid-air systems constrained radially by insulating sidewalls and axially by externally controlled electrodes. In this work, we present the design and methodology for electrostatically oscillated experiments, experimentally determined stability curves, and the distinctions and commonalities between mechanically and electrostatically forced Faraday systems. Multiple geometries are examined in order to illustrate the effects of aspect ratio and sidewall boundary conditions. Pure AC forcing and AC forcing with a DC offset are both discussed. Videos of the generated patterns are shown and compared to theoretical predictions of their corresponding Faraday mode shape. We compare our observations with respect to the base state oscillation of the system are observed upon the onset of the instability, a feature indicative of parametric instabilities.

1Supported by NSF 0968313, CASIS NNL11CD70A, NSF EAPS 1514711, NSF DGE-1315138.

4:57PM L18.00005 Coherent-structure theory and bound-state formation in electrified falling films. TE-SHENG LIN, National Chiao Tung University, DMITRI TSELUIKO, Loughborough University, MARK BLYTH, University of East Anglia, SERAFIM KALLIADASIS, Imperial College London — We consider a perfectly conducting viscous liquid film flowing down an inclined wall and subjected to a normal electric field. The electric field introduces a destabilizing non-local term in the long-wave evolution equation [1] and the solutions may evolve into arrays of interacting pulses. We develop a weak-interaction theory for these pulses using elements from previous coherent-structure interaction theories we have developed [2,3]. We show that the standard first-neighbor approximation is no longer valid and it is essential to take into account long-range interactions. We also develop numerical continuation techniques to explore bifurcations in systems possessing translational symmetry, including transient and spatially-varying time-periodic solutions. We find that each bound state bifurcates from the primary branch when continuing with respect to the domain size, and we then construct full bifurcation diagrams taking into account all the bound states. Finally, we compare the bound states for the long-wave evolution equation with the ones found in Stokes calculations and find excellent agreement.


5:10PM L18.00006 Numerical and experimental study of rotating jet flows. SEUNGWON SHIN, Hongik University, Republic of Korea, ZHIZHAO CHE, LYES KAHOUADJI, OMAR MATAR, Imperial College London, JALEL CHERGUI, DAMIR JURIC, LUMSI-CNRS — Rotating jets are investigated through experimental measurements and numerical simulations. The experiments are performed on a rotating jet rig and the effects of a range of parameters controlling the liquid jet are investigated, e.g. jet flow rate, rotation speed, jet diameter, etc. Different regimes of the jet morphology are identified, and the dependence on several dimensionless numbers is studied, e.g. Reynolds number, Weber number, etc. The breakup process of droplets is visualized through high speed imaging. Full three-dimensional direct numerical simulations are performed using BLUE, a massively parallel two-phase flow code. The novel interface algorithms in BLUE track the gas-liquid interface through a wide dynamic range including ligament formation, break up and rupture.

1EPSRC Programme Grant, MEMPHIS, EP/K0039761/1

5:23PM L18.00007 Instability and pattern formation in electriﬁed liquid layers. QIMING WANG, York University, DEMETRIOS PAPAGEORGIOU, Imperial College London — The stability and axisymmetric deformation of two immiscible, viscous, perfect or leaky dielectric ﬂuids conﬁned in the annulus between two concentric cylinders are studied in the presence of radial electric ﬁelds. The ﬁelds are set up by imposing a constant voltage potential difference between the inner and outer cylinders. We derive a set of equations for the interface in the long-wavelength approximation which retains the essential physics of the system and allows for interfacial deformations to be as large as the annular gap height. As a result of this simpliﬁcation, we are able to solve the evolution equation [1] and the solutions may evolve into arrays of interacting pulses. We develop a weak-interaction theory for these pulses using elements from previous coherent-structure interaction theories we have developed [2,3]. We show that the standard first-neighbor approximation is no longer valid and it is essential to take into account long-range interactions. We also develop numerical continuation techniques to explore bifurcations in systems possessing translational symmetry, including transient and spatially-varying time-periodic solutions. We find that each bound state bifurcates from the primary branch when continuing with respect to the domain size, and we then construct full bifurcation diagrams taking into account all the bound states. Finally, we compare the bound states for the long-wave evolution equation with the ones found in Stokes calculations and find excellent agreement.


5:36PM L18.00008 Feedback control of falling liquid films. ALICE THOMPSON, SUSANA GOMES, Imperial College London, DMITRI TSELUIKO, Loughborough University, GRIGORIOS PAVLIOTIS, DEMETRIOS PAPAGEORGIOU, Imperial College London — Falling liquid films become unstable when the Reynolds number increases above a critical value dependent on slope angle. In the unstable regime, the system first exhibits two-dimensional travelling waves, followed eventually by a transition to chaos. For applications such as coating, a flat, smooth film is desired, while for heat and mass transfer purposes, a non-uniform interface is beneﬁcial. Here we discuss the use of feedback control, based on observations of the film thickness, to enhance or suppress the instability. We use two contrasting long wave models to characterise the system dynamics, and investigate robustness to a number of static and dynamic control strategies for control towards uniform or non-uniform target states. We discuss the differences in control strategy required if feedback is to be delivered by the actuation of suction, heating, or time-dependent topography.

5:49PM L18.00009 ABSTRACT WITHDRAWN
6:02PM L18.00010 Self-similarity of solitary pulses on falling liquid films1. FABIAN DENNER, ALEXANDROS CHAROGIANNIS, Imperial College London, MARC PRADAS, The Open University, CHRISTOS N. MARKIDES, BEREND G.M. VAN WACHEM, SERAFIM KALLIADASIS, Imperial College London — A gravity-driven liquid film is unstable to long-wave perturbations above a critical Reynolds number. At low frequencies these perturbations evolve into fast solitary pulses. These strongly non-linear structures have a dominant elevation with a long tail and steep front, typically with capillary ripples preceding the main wave hump. We present the results of a comprehensive numerical study of solitary pulses on gravity-driven inertia-dominated water films flowing down an inclined substrate for a range of inclination angles (45-90 degrees), Reynolds numbers (Re=20-120) and Kapitza numbers (Ka=2765-3887). Our results reveal a self-similarity of solitary pulses on falling films and provide an in-depth understanding of the driving physical mechanisms of such pulses. We formulate a consistent characterisation of the shape and non-linear dispersion of solitary pulses, founded on a newly proposed scaling derived from the Nusselt flat film solution. We present and discuss our findings and resulting correlations with respect to the self-similarity of the shape and non-linear dispersion of solitary pulses as well as the influence of gravity and surface tension on solitary pulses in general.

1We acknowledge financial support from the Engineering and Physical Sciences Research Council (EPSRC) through Grant No. EP/K008595/1 and Grant No. EP/M021556/1.

6:15PM L18.00011 Parallelised direct numerical simulation of three-dimensional wavy falling films 1. DAMIR JURIC, JALEL CHERGUI, LIMSI-CNRS, LYES KAHOUADJI, OMAR MATAR, Imperial College London, SEUNGWON SHIN, Hongik University, Republic of Korea — We present a computational study of falling liquid films in a three-dimensional inclined rectangular domain using the new massively parallel code, BLUE. Calculations are carried out in order to obtain several wave patterns such as occasional solitary waves, which travel downstream at a constant velocity, or less coherent structures. BLUE uses parallelization algorithms based on MPI and algebraic domain decomposition. The velocity field is solved by a parallel GMRES method for the viscous terms and the pressure by a parallel multigrid method. The method for the treatment of the fluid interfaces and capillary forces uses a parallelized Front Tracking/Level Set technique which defines the interface both by a discontinuous density field as well as by a local triangular Lagrangian mesh. This structure allows the interface to undergo large deformations including the rupture and/or coalescence of fluid interfaces.

Monday, November 23, 2015 4:05PM - 6:41PM — Session L19 Vortex Dynamics: Vortex Rings 207 - Bartosz Protas, McMaster University

4:05PM L19.00001 Linear Stability of Hill’s Vortex to Axisymmetric Perturbations. BARTOSZ PROTAS, McMaster University — We consider the linear stability of Hill’s vortex with respect to axisymmetric perturbations. Given that Hill’s vortex is a solution of a free-boundary problem, this stability analysis is performed by applying methods of shape differentiation to the computation of the evolution of Hill’s vortex in a 3D system of axisymmetric perturbations on the linearized evolution of the vortex. The resulting singular integro-differential operator defined on the vortex boundary is discretized with a spectral approach. This operator has two unstable and two stable eigenvalues complemented by a continuous spectrum of neutrally-stable eigenvalues. By considering a family of suitably regularized (smoothed) problems we demonstrate that the corresponding eigenfunctions are in fact singular objects in the form of infinitely sharp peaks localized at the front and rear stagnation points. These findings thus refine the results of the classical analysis by Moffatt & Moore (1978).

4:18PM L19.00002 An experimental study on the formation of negatively-buoyant vortex rings. JEFF X. WU, GARY R. HUNT, Univ of Cambridge — Experiments to examine the formation of dense saline vortex rings projected vertically upwards into a quiescent freshwater environment were conducted. The setup was designed to dispense a cylindrical column of source fluid with aspect ratio L/D (the length L of dispensed saline column to the nozzle diameter D) over a pre-set time interval. In an effort to execute an impulsive start and finish, a controlled flow circulation driven by a gear pump was developed to approximate a top-hat profile of source exit velocity versus time. Our measurements focus on describing the evolving morphology of the vortex rings with time and with source conditions (L/D and source Froude number). Our results reveal distinct formation regimes and our estimates of time required for formation as a function of density difference confirm predictions from previously published numerical simulations. The volume-based approach we adopt provides potentially a new angle for investigating the physics of these flows.

4:31PM L19.00003 The effect of entrainment on starting vortices. GIUSEPPE ROSI, DAVID RIVAL, Queen’s University — Recent work shows that vortex detachment behind accelerating plates coincides with when streamlines enclosing the starting vortex (SV) form a full saddle.I In the case of a linearly accelerating plate, it can be shown that vorticity-containing mass, and thus the SV’s development scale with only dimensionless groups: azimuthal mode number, and thus the SV’s development scale with only dimensionless groups, while the SV’s circulation scales with the acceleration rate. This results in shear-layer instabilities whose structure is Reynold-number independent, but whose strength scale with Reynolds number. It is hypothesized that the increased strength of the instabilities promotes entrainment, which causes the formation of the full saddle and thereby detachment to occur at an earlier dimensionless time. To test this hypothesis, a circular plate is linearly accelerated from rest to pinch-off with chord-based Reynolds numbers of 10^3, 10^4, and 10^5 at the midpoint of the motion. Planar PIV data is acquired, from which FTLE and enstrophy fields are calculated. Vortex detachment is identified from the dynamics of the FTLE saddles, while the enstrophy fields are used to calculate both the vorticity-containing mass entering from the shear layer and the mass entrained from the quiescent surroundings.


4:44PM L19.00004 The effect of aspect ratio on vortex pinch-off over laminar and turbulent regimes. JOHN FERNANDO, DAVID RIVAL, Queen’s University — In the current study, vortex rings formed behind accelerating flat plates are investigated to determine the role of aspect ratio on pinch-off over a range of 10^3 ≤ Re ≤ 10^5. We begin by demonstrating that aspect ratio plays a primary role in pinch-off, while the role of plate-edge curvature is of secondary importance. For vortex rings produced in the wake of elliptical plates (AR≥1), the point of vortex pinch-off has been shown to be coterminous with the formation of the full saddle and thereby detachment to occur at an earlier Reynolds number. For circular plates (AR=1), pinch-off is not clearly identified, and the vortex ring eventually breaks down in the wake. It is hypothesized that with increasing Reynolds number the vortex rings develop more quickly due to increased levels of mixing (entrainment) across the shear-layer interface. As such, vortex pinch-off is hastened for the circular plate with increasing Reynolds number, yet remains unchanged for the elliptical plate, for which the timescales of vortex-ring deformation (i.e. detachment) are faster than the rate of fluid entrainment. Force and velocimetry measurements are used to support this hypothesis.
The process of baroclinic vorticity production is different for upward or downward moving vortices (from dense to light and from light to dense fluids). When a vortex ring crossing the interface between two stratified miscible liquids, we study the evolution of the ring while crossing the interface considering positive and negative density contrasts. The velocity, density and vortex ring crossing the interface are analyzed by calculating their circulation, energy and trajectories. The cutting of vortex rings below center line leads to the formation of secondary vortex rings and the propagation of initial vortex ring in the downstream of the plate are analyzed by calculating their circulation, energy and trajectories.

The vortex rings elongate radially along the collision plane, while the two vortex cores approach one another. When the distance between the vortex cores reaches a critical length scale, they either reconnect into secondary vortex rings or break down and dissipate into a turbulent cloud, depending on their initial Reynolds number. By filtering this collision at high speeds, while illuminating it with a scanning laser sheet, we can reconstruct the intricate three-dimensional flow structure at the collision plane. We find that the onset of the vortex ring breakdown is triggered by a sequential cascade of instabilities that interact with the vortex cores. Understanding the role of these instabilities in the breakdown of vortex rings could provide new insight into the evolution and stabilization of vortices.

The vortex rings elongate radially along the collision plane, while the two vortex cores approach one another. When the distance between the vortex cores reaches a critical length scale, they either reconnect into secondary vortex rings or break down and dissipate into a turbulent cloud, depending on their initial Reynolds number. By filtering this collision at high speeds, while illuminating it with a scanning laser sheet, we can reconstruct the intricate three-dimensional flow structure at the collision plane. We find that the onset of the vortex ring breakdown is triggered by a sequential cascade of instabilities that interact with the vortex cores. Understanding the role of these instabilities in the breakdown of vortex rings could provide new insight into the evolution and stabilization of vortices.

Financial support from the State Key Development Program of Basic Research of China (2014CB744802) is gratefully acknowledged.

Acknowledge support of ARL Distinguished Undergraduate Research Program.

The study aims to fill the gap in the current knowledge of dynamics of an elliptic vortex ring in a viscous fluid and also to address the issue of whether elliptic ring undergoes vortex stretching and compression during oscillatory deformation. In the presence of a linear shear flow, the elliptic vortex ring undergoes not only oscillatory deformation and stretching but also tilting as it propagates.

The evolution of a viscous elliptic vortex ring in an initially quiescent fluid or a linear shear is numerically simulated. A wide range of parameters are considered, for aspect ratios (AR) (1 ≤ AR ≤ 8), radius to ring radius ratios (c) (0.1 ≤ c ≤ 0.3), Reynolds number (Re) (500 ≤ Re ≤ 3000) and shear rate (K) (0 ≤ K ≤ 0.12). The study aims to fill the gap in the current knowledge of dynamics of an elliptic vortex ring in a viscous fluid and also to address the issue of whether elliptic vortex ring undergoes vortex stretching and compression during oscillatory deformation. In the presence of a linear shear flow, the elliptic vortex ring undergoes not only oscillatory deformation and stretching but also tilting as it propagates.

The evolution of an elliptic vortex ring in a viscous fluid marks an important milestone, as it provides insights into the behavior of vortex rings in complex flows.
are statistics affected by two-way coupling numerical scheme?

1

numbers, mass-loadings, and particle diameters to grid size. simulations of the Navier-Stokes equations to study decaying homogeneous isotropic turbulence laden with particles with different particle Stokes between the disturbed and undisturbed fluid velocity at the location of each particle. The objective of this work is to demonstrate how different of momentum exchange between the two phases. A simple interpolation correction scheme has been proposed which accounts for the difference

Stanford University — This work builds on our recent analyses of two-way coupled particle-fluid systems under laminar conditions, in which we

of strong vortices enhance relative diffusion of inertial particles of

constant of $\epsilon$ which is in agreement with Richardson (1926) and Obukhov (1941). Here

value of $St$ using cubic spline interpolation for the velocity data in the DNS. Here

rate vector and a long axis.

eigenvector of the left Cauchy-Green tensor. The vorticity also aligns in this same direction leading to alignment between the particle rotation into the Lagrangian evolution of vorticity and strain in turbulence. Particles tend to align with a long axis along the maximum extensional eigenvector of the left Cauchy-Green tensor. The vorticity also aligns in this same direction leading to alignment between the particle rotation rate vector and a long axis.

4:18PM L20.00002 Alignment of Disks with Lagrangian Stretching in Turbulence . CONOR HUNT, LYDIA TIERNEY, STEFAN KRAMEL, GREG VOTH, Wesleyan University — We study Lagrangian stretching in isotropic turbulence in order to understand both the rotations of disks and the preferential alignment of vorticity with the intermediate strain rate eigenvector. Using velocity gradient tensors from a numerical simulation of homogeneous isotropic turbulence at $R_e = 180$, we calculate the Cauchy-Green strain tensors whose eigenvectors provide a natural basis for studying stretching phenomenon. Previous work has shown that rods preferentially align with the vorticity as a result of both quantities independently aligning with the extensional Cauchy-Green eigenvector. In contrast, disks orient with their symmetry axis perpendicular to vorticity and preferentially align with the compressional Cauchy-Green eigenvector. We also find that the intermediate strain rate eigenvector is aligned with the extensional Cauchy-Green eigenvector. A natural consequence is that the intermediate strain rate eigenvector is aligned with the vorticity vector since conservation of angular momentum aligns vorticity with the direction it has been stretched.

4:31PM L20.00003 Distribution and velocity of inertial particles in a turbulent channel flow , FILIPPO COLETTI, KEE ONN FONG, ANDRAS NEMES, NICHOLAS SLOAN, University of Minnesota — The segregation of inertial particles in specific regions of turbulent fluid flows is a well known phenomenon, but experimental observations of its three-dimensional nature are lacking. Here we are concerned with the transport of small inertial particles in wall-bounded turbulence. In particular we consider a fully developed vertical channel flow. The working fluid is air laden with size-selected glass particles. The volume and mass loading are kept low and the particle diameter is smaller than the viscous length scale, so that the turbulence is unaffected by the dispersed phase. Tomographic particle image velocimetry is used to reconstruct the position and velocity of the inertial particles. In particular, the tendency of the particles to concentrate intermittently (turbulence clustering) and to drift towards the wall (turbophoresis) are quantitatively characterized by the instan-
taneous and mean concentration fields, respectively. The findings are discussed in relation to the results of previous studies which have used one-way coupled direct numerical simulations, and on which the current understanding of this class of flows is based.

4:44PM L20.00004 Relative diffusion of a pair of inertial particles in the inertial sub-range of turbulence , KEI ENOHATA, Nagoya University, KOJI MORISHITA, Kobe University, TAKASHI ISHIHARA, JST CREST, Nagoya University — Turbulent diffusion of a pair of inertial particles in 3-dimensional homogeneous and isotropic turbulence was studied using direct numerical simulation (DNS) with 2048$^3$ grid points; the Taylor micro-scale Reynolds number in the DNS is approximately 425. For each set of the inertial particles with different values of the Stokes number ($St = 0, 0.1, 0.2, 0.5, 1, 2, 5, 10$), $256^3$ particles are tracked using cubic spline interpolation for the velocity data in the DNS. Here $St = 0$ corresponds to fluid particles. The DNS showed that for each value of $St$, the mean square of the distance $\delta x$ between the two inertial particles grows with time $t$ as $\langle \delta x^2 \rangle \sim C t^{1/3}$ in the inertial subrange, which is in agreement with Richardson (1926) and Obukhov (1941). Here $\epsilon$ is the mean energy dissipation rate per unit mass, and $C$ is a constant of $O(1)$ depending on the value of $St$ and the initial distance between the inertial particles. The DNS shows also that large clusters of strong vortices enhance relative diffusion of inertial particles of $St > 1$.

4:57PM L20.00005 Simulations of decaying turbulence laden with particles: how are statistics affected by two-way coupling numerical scheme?1. JEREMY HORWITZ, ALI MANI, Stanford University — This work builds on our recent analyses of two-way coupled particle-fluid systems under laminar conditions, in which we demonstrated that standard interpolation schemes in conjunction with the point-particle approximation can lead to significant underprediction of momentum exchange between the two phases. A simple interpolation correction scheme has been proposed which accounts for the difference between the disturbed and undisturbed fluid velocity at the location of each particle. The objective of this work is to demonstrate how different interpolation and projection schemes affect flow and/or particle statistics in a mean-stationary turbulent environment. We use direct numerical simulations of the Navier-Stokes equations to study decaying homogeneous isotropic turbulence laden with particles with different particle Stokes numbers, mass-loadings, and particle diameters to grid size.

1Supported by DOE, J. H. Supported by NSF GRF
5:10PM L20.00006 Inertial Range Scaling of Rotation Rates of Particles in Turbulence, BRENDAN COLE, STEFAN KRAMEL, GREG VOTH, Wesleyan Univ. — We measure mean-squared rotation rates of 3D-printed particles with sizes spanning the inertial range in a turbulent flow between oscillating grids. Tetrads, composed of four slender rods in the tetrahedral symmetry, and triads, three slender rods in triangular planar symmetry, are tracked in a flow with \( \text{Re}_r = 156 \) and \( \text{Re}_\lambda = 214 \) using four high-speed cameras. Tetrads rotate like spheres and triads rotate like disks. Measurements of tetrads’ rotation rates as a function of particle size are direct measurements of the coarse-grained vorticity and provide a new way to measure inertial range scaling in turbulent flows. Similar measurements of rods, performed by Parsa and Voth, were consistent with the \( < \omega^2 > \sim r^{-3} \) scaling prediction, but the preferential alignment of rods affects their rotation rate and this preferential alignment could not be directly measured. Our triads allow measurement of the full solid-body rotation rate as well as the particle orientation and so we can quantify the preferential alignment for disks.

5:23PM L20.00007 Settling of inertial particles through quiescent, weakly turbulent and strongly turbulent air, ALEC PETERSEN, DOUGLAS CARTER, LUCI BAKER, FILIPPO COLETTI, University of Minnesota — The fall speed of inertial particles suspended in a turbulent flow is an important parameter for numerous industrial applications, including the study of natural phenomena. Although this behavior has been subject to various investigations, it has been difficult to assess. Previous studies have provided several mechanisms which may either hinder or enhance the settling rate, but comparisons between them often show qualitative and quantitative discrepancies. We experimentally study the case of a suspension of size-selected microscopic particles falling through air. The mass fraction of the particles is low enough to neglect their influence on the fluid. Using randomly actuated jets of adjustable intensity, we gradually increase the mean velocity fluctuations of the air without introducing a significant mean flow. Because the integral length scale is kept approximately constant, the Kolmogorov scales become progressively smaller as the Reynolds number increases. We measure the average and fluctuating settling velocities for the different cases, and discuss the compatibility of our observations with the various proposed mechanisms of settling alteration by turbulence.

5:36PM L20.00008 Wake-driven dynamics of finite-sized buoyant spheres in turbulence, VARGHESE MATHAI, VIVEKN PRAKASH, JON BRONS, CHAO SUN, DETLEF LOHSE, Univ of Twente, PHYSICS OF FLUIDS GROUP TEAM — Particles suspended in turbulent flows are affected by the turbulence, and at the same time act back on the flow. The resulting coupling can give rise to rich variability in their dynamics. Here we report experimental results from an investigation on finite-sized buoyant spheres in turbulence. We find that even a marginal reduction in the particles density from that of the fluid can result in strong modification of the particle dynamics. In contrast to classical spatial filtering arguments, we find that the particle acceleration variance increases with size. We trace this reversed trend back to the growing contribution from wake-induced forces.

5:49PM L20.00009 Effects of ambient turbulence on a particle plume, ADRIAN C.H. LAI, Singapore-MIT Alliance for Research and Technology Centre, J.W. ER, ADRIAN W.K. LAW, Nanyang Technological University, E. ERIC ADAMS, Massachusetts Institute of Technology. — We investigated experimentally the effects of ambient turbulence on a particle plume. Homogeneous and isotropic turbulent ambient water was generated by a random jet array in a glass tank. Glass beads of different particle diameters were released continuously into this turbulent ambient using a submerged hourglass, forming particle plumes with a constant efflux velocity; different initial velocities were tested for each particle size. We focused on the region in which the integral length scale of the ambient eddies is larger than that of the particle plume size. Following the arguments of Hunt (1994) and the observation of Hubner (2004) on a single-phase plume, it is expected that in this region, the internal structure or Lagrangian spreading of the particle plume, will not be significantly affected, but the plume centerline would meander due to the ambient turbulence leading to an increase in the Eulerian width. In the presentation, first, we will present our preliminary experimental data which showed that this is also true for two-phase particle plumes. Second, based on this observation, we developed a theoretical framework using a stochastic approach to predict the spreading of the plume. Predictions of the model will be compared with our experimental data.

1This research was supported by the National Research Foundation Singapore through the Singapore MIT Alliance for Research and Technology’s Center for Environmental Sensing and Modeling interdisciplinary research program.

6:02PM L20.00010 Dynamics of Small Inertia-Free Spheroidal Particles in a Turbulent Channel Flow, NIRANJAN REDDY CHALLABOTLA, LIHAO ZHAO, HELGE I. ANDERSSON, Norwegian Univ Tech (NTNU), DEPARTMENT OF ENERGY AND PROCESS ENGINEERING TEAM — The study of small non-spherical particles suspended in turbulent flows is of interest in view of the potential applications in industry and the environment. In the present work, we investigated the dynamics of inertia-free spheroidal particles suspended in fully-developed turbulent channel flow at \( \text{Re}_r = 180 \) by using the direct numerical simulations (DNS) for the Eulerian fluid phase and the Lagrangian point-particle tracking. We considered inertia-free spheroidal particles with a wide range of aspect ratios from 0.01 to 50, i.e. from flat disks to long rods. Although the spheroids passively translate along with the fluid, the particle orientation and rotation strongly depend on the particle shape. The flattest disks were preferentially aligned with their symmetry axis normal to the wall, whereas the longest rods aligned parallel to the wall. Strong mean rotational spin was observed for spherical particles and this has been damped with increasing asphericity both for rod-like and disk-like spheroids. The anisotropic mean and fluctuating fluid vorticity resulted in particle spin anisotropies which exhibited a complex dependence on the particle asphericity.

1The Research Council of Norway, Notur and COST Action FP1005 are gratefully acknowledged.

6:15PM L20.00011 Single particle measurements of material line stretching in turbulence: Experiments, STEFAN KRAMEL, Wesleyan University, SASKIA TYMPEL, FEDERICO TOSCHI, TU Eindhoven, GREG VOTH, Wesleyan University — We find that particles in the shape of chiral dipoles display a preferential rotation direction in three dimensional isotropic turbulence. The particles consist of two helical ends with opposite chirality that are connected by a straight rod. They are fabricated using 3D printing and have an aspect ratio of 10 and a length in the inertial range of our flow between oscillating grids. Due to their high aspect ratio, they move like material lines. Because material lines align with the extentional eigenvectors of the velocity gradient tensor they experience a mean stretching in turbulence. The stretching of a chiral dipole produces a rotation about the dipole axis and so chiral dipoles experience a non-zero mean spinning rate in turbulence. These results provide a first direct experimental measurement of the rate of material line stretching in turbulence.
6:28PM L20.00012 Single particle measurements of material line stretching in turbulence: Numerics, SASKIA TYMPEL, University of Technology Eindhoven, STEFAN KRAMEL, Los Alamos National Laboratory — This work was supported by Japan Society for the Promotion of Science KAKENHI Grant Number 26709066. Computer time was provided by the K computer at the RIKEN Advanced Institute for Computational Science through the HPCI System Research project hp150035.

Monday, November 23, 2015 4:05PM - 6:41PM — Session L21 Turbulence: Compressible Boundary Layers and Jets 209 - Steven Beresh, Sandia National Laboratories

4:05PM L21.00001 Log-law and compressibility effects in transcritical turbulent boundary layers at supercritical pressure\(^1\). Soshi Kawai, Department of Aerospace Engineering, Tohoku University — In this talk, we discuss the log-law and effects of compressibility in transcritical heated turbulent boundary layers on a zero-pressure-gradient flat plate at supersonic pressure conditions by solving the compressible Navier-Stokes equations using direct numerical simulation. In the transcritical fluids (especially at transcritical conditions), due to the strong real fluid effects thermodynamic properties vary abruptly within a narrow temperature range through the pseudo-critical temperature and significantly deviate from the ideal fluid. Peculiar interactions between the strongly non-linear real fluid effects and wall turbulence, and its resultant log-law and turbulence statistics are discussed, which have never been seen in the ideal-fluid turbulent boundary layers. We also show non-negligible compressibility effects in the flow even in the low-Mach number regime considered in this study.

\(^1\)This work was supported by Japan Society for the Promotion of Science KAKENHI Grant Number 26709066. Computer time was provided by the K computer at the RIKEN Advanced Institute for Computational Science through the HPCI System Research project hp150035.

4:18PM L21.00002 Turbulent Eddies in a Compressible Jet in Crossflow Measured using Pulse-Burst PIV, Steven Beresh, Justin Wagner, John Henfling, Russell Spillers, Brian Pruett, Sandia National Laboratories — Pulse-burst Particle Image Velocimetry (PIV) has been employed to acquire time-resolved data at 25 kHz of a supersonic jet exhausting into a subsonic compressible crossflow. Data were acquired along the windward boundary of the jet shear layer and used to identify turbulent eddies as they convect downstream in the far-field of the interaction. Eddies were found to have a tendency to occur in closely-spaced counter-rotating pairs and are routinely observed in the PIV movies, while the variable orientation of these pairs makes them difficult to detect statistically. Correlated counter-rotating vortices are more strongly observed to pass by at a larger spacing, both leading and trailing the reference eddy. This indicates the paired nature of the turbulent eddies and the tendency for these pairs to convect through the field of view at repeatable spacings. Velocity spectra reveal a peak at a frequency consistent with this larger spacing between shear-layer vortices rotating with identical sign. Super-sampled velocity spectra to 150 kHz reveal a power-law dependency of -5/3 in the inertial subrange as well as a -1 dependency at lower frequencies attributed to the scales of the dominant shear-layer eddies.

4:31PM L21.00003 Development of turbulent variable density mixing in jets with coflow, John Charonko, Kathy Prestridge, Physics Division, Los Alamos National Laboratory — Fully turbulent jets with coflow at two density ratios (At=0.1 & 0.6) were studied as a statistically stationary system for improving our understanding of variable density mixing in turbulent flows. The exit Reynolds number was matched for both flows at ~19,000 and simultaneous planar PIV and acetone PLIF measurements were acquired so the coupled evolution of the velocity and density statistics could be examined in terms of density-weighted average quantities. Measurements were taken on 10,000 snapshots of the flow at three locations to insure statistical convergence, and the spatial resolution (288 \(\mu\)m) was at or below the Taylor microscale. In agreement with our previous work at lower Reynolds numbers, for large density ratios turbulent kinetic energy and Reynolds stresses are preserved or increased with downstream distance, contrasting with the behavior at low density ratios. Furthermore, in regions where the buoyancy effects began dominating the initial momentum-driven flow (~30 jet diameters), the jet is still not developing toward a self-similar state. Instead, a region of homogeneous turbulence appeared to establish itself in the center of the jet even for the lower density ratio condition, in contrast with classical results for single-fluid jets.

4:44PM L21.00004 Identifying Coherent Structures in a 3-Stream Supersonic Jet Flow using Time-Resolved Schlieren Imaging, Andrew Tenney, Syracuse Univ, Thomas Coleman, Matthew Berry, Andy MAGSTADT, Syracuse University, Sivaram Gogineni, Spectral Energies LLC, Barry Kiel, AFRL — Shock cells and large scale structures present in a three-stream non-axisymmetric jet are studied both qualitatively and quantitatively. Large Eddy Simulation is utilized first to gain an understanding of the underlying physics of the flow and direct the focus of the physical experiment. The flow in the experiment is visualized using long exposure Schlieren photography, with time resolved Schlieren photography also a possibility. Velocity derivative diagnostics are calculated from the grey-scale Schlieren images are analyzed using continuous wavelet transforms. Pressure signals are also captured in the near-field of the jet to correlate with the velocity derivative diagnostics and assist in unraveling this complex flow. We acknowledge the support of AFRL through an SBIR grant.
4:57PM L21.00005 Hydro-acoustic instabilities in compressible turbulent channel flow with porous walls, CARLO SCALO1, IMAN RAHBARI2, Purdue University — C. Scalò, J. Bodart, and S. K. Lele, Phys. Fluids (2015) manipulated wall-bounded compressible turbulence by applying impedance boundary conditions (IBC) acoustically tuned to the characteristic time scale of the large-scale eddies. Near-wall turbulence was overhauled by hydro-acoustic instabilities - comprised of coherent spanwise Kelvin-Helmholtz rollers driven by Helmholtz-like acoustic resonance - while outer-layer turbulence was left structurally unaltered. We discuss linear modeling results of the observed flow response, supported by new high-fidelity simulations up to transonic bulk Mach numbers. For IBCs with zero reactance, corresponding to a Darcy-like formulation for porous walls, two dominant modes are identified whose Reynolds stress distributions overlap with the impermeable-wall turbulent buffer layer, directly affecting the near-wall turbulence cycle. For the range of wavenumbers investigated, the transition from subcritical to supercritical permeability does not significantly alter the structure of the unstable modes, showing that wall-permeability accentuates pre-existing, otherwise stable, modes. Implications on flow control strategies for compressible boundary layers over porous walls are discussed.

1School of Mechanical Engineering
2School of Mechanical Engineering

5:10PM L21.00006 Geometric invariance of compressible turbulent boundary layers, WEI-TAO BI, BIN WU, ZHEN-SU SHE, Peking University, FAZLE HUSSAIN, Texas Tech University — A symmetry based approach is applied to analyze the mean velocity and temperature fields of compressible, flat plate turbulent boundary layers (CTBL). A Reynolds stress length scale and a turbulent heat flux length scale are identified to possess the same defect scaling law in the CTBL bulk, which is solely owing to the constraint of the wall to the geometry of the wall-attached eddies, but invariant to compressibility and wall heat transfer. This invariance is called the geometric invariance of CTBL eddies and is likely the origin of the Mach number invariance of Morkovin’s hypothesis, as well as the similarity of energy and momentum transports. A closure for the turbulent transport by using the invariant lengths is attained to predict the mean velocity and temperature profiles in the CTBL bulk—superior to the van Driest transformation and the Reynolds analogy based relations for its sound physics and higher accuracy. Additionally, our approach offers a new understanding of turbulent Prandtl number.

5:23PM L21.00007 A symmetry based approach to quantifying the compressible turbulent boundary layer, BIN WU, WEI-TAO BI, ZHEN-SU SHE, Peking University, FAZLE HUSSAIN, Texas Tech University — Developing analytical description of the compressible turbulent boundary layer (CTBL) is of great importance to many technological applications and to the understanding and modeling of compressible turbulence. Here a symmetry-based approach is applied to analyze the CTBL data acquired from DNS, covering a wide range of Reynolds number (Re), Mach number (Ma) and wall temperature. The Reynolds stress length scale displays a four-layer structure in the direction normal to the wall and obeys the dilation group invariance as in the incompressible TBL. A newly-identified turbulent heat flux length scale behaves similarly, which is the classical temperature mixing length weighted by the mean temperature. A significant result is the identification of three physical parameters for each length function to characterize the adiabatic flow: a bulk flow constant, a buffer layer thickness and a boundary layer edge, which vary with Re and Ma. For the diabatic flow, the sublayer thickness and the inner layer scaling exponents vary additionally with the wall temperature. These parameters are modeled empirically, leading to a highly accurate prediction of the mean fields of the CTBL. Thus we reveal that the symmetry principle found in canonical wall-bounded flows holds also for the CTBL, and a quantitative mean field theory is viable with appropriate symmetry considerations.

5:36PM L21.00008 Effect of Pulsed Plasma Jets on the Recovering Boundary Layer Downstream of a Reflected Shock Interaction1, BENTON GREENE, NOEL CLEMENS, University of Texas System, PATRICK MAGARI, DANIEL MICKA, MATTHEUS UECKERMANN, Creare, LLC — Shock-induced turbulent boundary layer separation can have many detrimental effects in supersonic inlets including flow distortion and instability, structural fatigue, poor pressure recovery, and unstart. The current study investigates the effect of pulsed plasma jets on the recovering boundary layer downstream of a reflected shock wave-boundary layer interaction. The effects of pitch and skew angle of the jet as well as the heating parameter and discharge time scale are tested using several pulsing frequencies. In addition, the effect of the plasma jets on the undisturbed boundary layer at 6 mm and 11 mm downstream of the jets is measured. A pitot-static pressure probe is used to measure the velocity profile of the boundary layer 35 mm downstream of the plasma jets, and the degree of boundary layer distortion is compared between the different models and run conditions. Additionally, the effect of each actuator configuration on the shape of the mean separated region is investigated using surface oil flow visualization. Previous studies with lower energy showed a weak effect on the downstream boundary layer. The current investigation will attempt to increase this effect using a higher-energy discharge.

1Funded by AFRL through and SBIR in collaboration with Creare, LLC

5:49PM L21.00009 Acoustic Radiation from a Mach 14 Turbulent Boundary layer1, CHAO ZHANG, LIAN DUAN, Missouri Univ of Sci & Tech, MEELAN CHOUDHARI, NASA Langley Research Center — Direct numerical simulations (DNS) are used to examine the pressure fluctuations generated by a high-speed turbulent boundary layer with a nominal freestream Mach number of 14 and wall temperature of 0.18 times the recovery temperature. The emphasis is on characterizing the acoustic radiation from the turbulent boundary layer and comparing it with previous simulations at Mach 2.5 and Mach 6 to assess the Mach-number dependence of the freestream pressure fluctuations. In particular, the numerical database is used to provide insights into the pressure disturbance spectrum and amplitude scaling with respect to the freestream Mach number as well as to understand the acoustic source mechanisms at very high Mach numbers. Such information is important for characterizing the freestream disturbance environment in conventional (i.e., noisy) hypersonic wind tunnels. Spectral characteristics of pressure fluctuations at the surface are also investigated.

1Sponsored by Air Force Office of Scientific Research
6:02PM L21.00010 Investigation of Shock-Induced Laminar Separation Bubble in a Supersonic Boundary Layer1, JAYAHAR SIVASUBRAMANIAN, HERMANN FASEL, University of Arizona — The interaction between an impinging oblique shock and a laminar boundary-layer on a flat plate is investigated using DNS. In particular, the two-dimensional separation bubble resulting from the shock/boundary-layer interaction (SBLI) at freestream Mach number of 2.0 is investigated in detail. The flow parameters used for the present investigation match the laboratory conditions in the experiments by Hakkinen et al. The skin friction and pressure distribution from the simulations are compared to the experimental measurements and numerical results available in the literature. Our results confirm the asymmetric nature of the separation bubble as reported in the literature. In addition to the steady flow field calculations, the response to low-amplitude disturbances is investigated in order to study the linear stability behavior of the separation bubble. For comparison, both the development of two-dimensional and three-dimensional (oblique) disturbances are studied with and without the impinging oblique shock. Furthermore, the effects of the shock incidence angle and Reynolds number are also investigated. Finally, three-dimensional simulations were performed in order to explore the laminar-turbulent transition process in the presence of a laminar separation bubble.

1Funded by the Air Force Office of Scientific Research under grant FA9550-14-1-0195

6:15PM L21.00011 MOVED TO D4.010 —

6:28PM L21.00012 Investigation of corner shock boundary layer interactions to understand inlet unstart, MORGAN FUNDERBURK, North Carolina State University — Inlet unstart is a detrimental phenomenon in dual-mode ramjet/scramjet engines that causes severe loss of thrust, large transient structural load, and potentially a loss of the aircraft. In order to analyze the effects that the corner shock boundary layer interaction (SBLI) has on initiating and perpetuating inlet unstart, a qualitative and quantitative investigation into mean and dynamic features of corner SBLI at various Mach numbers is made. Surface streakline visualization showed that the corner SBLI is highly three-dimensional with a dominant presence of corner separation vortex. Further, the peak r.m.s. pressure was located at the periphery of corner separation vortex, suggesting that the unsteady loading is caused by the corner vortex. Power spectral densities of wall-pressure fluctuations in the peak r.m.s. location were analyzed in order to characterize the dominant frequencies of oscillation of the flow structures and to unravel the dynamic interactions between them in order to expand the operating margin of future hypersonic air breathing vehicles.

Monday, November 23, 2015 4:05PM - 6:41PM — Session L22 Turbulent Boundary Layers IV 210 - Julian Hunt, University College London

4:05PM L22.00001 Thin shear layers in homogeneous high Reynolds number turbulence and in turbulent boundary layers, TAKASHI ISHIHARA, JST CREST, Nagoya University, KOJI MORISHITA, Kobe University, JULIAN HUNT, University College London — Direct numerical simulations (DNS) at high Reynolds number show for forced homogeneous isotropic turbulence at Reλ ∼ 1000 that randomly moving, thin shear layers form in the interior (T/In), in which there are high-enstrophy, micro-scale vortex tube structures. These layers have thicknesses of the order of the Taylor micro-scale and the interfaces at the outside of the layers act as a partial barrier to the fluctuations on either side of the layers. In the turbulent boundary layers (TBL) at Reλ ∼ 100, conditional statistics show three different types of thin shear layers; at the outer edge (T/NT), in the interior (T/In) and within the buffer layer near the wall (T/W). These layers act as barriers to the fluctuations on either side and can have controlling effects on the overall flow. The internal and external characteristics and role of the thin shear layers in homogeneous turbulence and in TBL are compared.

4:18PM L22.00002 General mechanisms of thin layers in high Reynolds number turbulent flows, JULIAN HUNT, University College London, TAKASHI ISHIHARA, JST CREST, Nagoya University, KOJI MORISHITA, Kobe University — Mechanisms and computation are presented for the three types of thin, high vorticity, randomly moving shear layers at high Reynolds number. They decorrelate eddy motions on each side and, in the first two types, have an internal micro-scale, dissipative structure. Their form also depends on the mean strain/shear outside the layer, and the proximity of any resistive boundaries. The first type (T/NT) lie between regions of sheared turbulence and external non-turbulent motions. Depending on whether the inflection points of the conditional mean shear profile, ⟨U(y − yi)⟩, relative to the interface coordinate yi, are on the outside or inside edges of the layer, the forms of the interface are “nibbling” motions on the scale of the layer thickness or large “engulfing” motions, which affect the overall flow structure. In the second type (T/In), which occurs in the interior of turbulent flows, because the interface instabilities are suppressed, the stretching increases more than in T/NT, causing the micro-scale vorticity, velocity and dissipation to greatly exceed Kolmogorov’s theory. The third type (T/W) within the buffer wall layer, by blocking outer eddies, determines the displaced form of the mean logarithmic profile, and fluctuations of wall shear stress.

4:31PM L22.00003 Passive scalars in turbulent channel flow at high Reynolds number1, SERGIO PIROZZOLI, MATTEO BERNARDINI, PAOLO ORLANDI, Sapienza Università di Roma, Dipartimento di Ingegneria Meccanica e Aerospaziale — We study passive scalars in turbulent plane channels at computationally high Reynolds number, which allows to observe previously unnoticed effects. The mean scalar profiles are found to obey a generalized logarithmic law which includes a linear correction term in the whole lower half-channel, and they follow a universal parabolic defect profile in the core region. This is consistent with recent findings regarding the mean velocity profiles in channel flow. The scalar variances also exhibit a near universal parabolic distribution in the core flow, and hints of a sizeable log layer, unlike the velocity variances. The energy spectra highlight the formation of large scalar-bearing eddies spanning each half-channel, which are caused by production excess over dissipation, and which are clearly visible in the flow visualizations. Close correspondence of the velocity and scalar eddies is observed, the main difference being that the latter have more convoluted interfaces, which translates into higher scalar dissipation. Another notable Reynolds number effect is the decreased correlation of the scalar field with the vertical velocity field, which is traced to the reduced effectiveness of ejection events.

1We acknowledge that the results reported in this paper have been achieved using the PRACE Research Infrastructure resource FERMI based at CINECA, Casalvecchio di Reno, Italy.
Mean flow statistics were well understood in case of zero pressure gradient flows. However, almost all turbulent boundary layers in technical applications, such as aircrafts, are subjected to a streamwise pressure gradient. When subjecting turbulent boundary layers to adverse pressure gradients—significant changes in the statistical behavior of the near-wall flow have been observed in experimental studies conducted. However, the details dynamics and characteristics of these flows have not been fully resolved. The sensitivity to Reynolds number and the dependency on several parameters, including the dependence on the pressure gradient parameter, is still under debate and very little information exists about statistically averaged quantities such as the mean velocity profile or Reynolds stresses. In order to improve the understanding of wall-bounded turbulence, this work experimentally investigates turbulent boundary layer subjected to favorable and adverse pressure gradients by means of Particle Image Velocimetry over a wide range of Reynolds numbers, 4200 < Re < 13400. The contribution of the coherent structures to the mean flow statistics was found to increase significantly for a flow subjected to an adverse pressure gradient.

The Reynolds shear stress maxima \( \tau_{max} / \tau_w \) occur at a point where ratio of mesolayer to outer lengths is of order \( R_e^{1/2} = (\nu/\delta u') \), and at this point DNS and experimental data predict \( U_m / U_e = 2/3 \) (where \( U_m \) = mesolayer velocity and \( U_e \) = velocity at boundary edge). The turbulent burst time period also scale with mesolayer time. The shape factor in a TBL shows linear behavior with non-dimensional mesolayer length scale. In special case \( U_m / U_e = 1/2 \), is due to Iizakon and Millikan. The above predictions are supported by experimental and DNS data.

The self-similar inertial range—the region between the mesolayer and the outer layer—has been analyzed. Below this crucial point the mesolayer inner limit matches with with outer limit of wall layer. Above this crucial point the outer limit of mesolayer matches with inner limit of outer layer. The log law velocity and Reynolds in two overlap regions, above and below the critical point, have been presented. The Reynolds shear stress maxima \( \tau_{max} / \tau_w \) occurs at a point where ratio of mesolayer to outer lengths is of order \( R_e^{1/2} = (\nu/\delta u') \), and at this point DNS and experimental data predict \( U_m / U_e = 2/3 \) (where \( U_m \) = mesolayer velocity and \( U_e \) = velocity at boundary edge). The turbulent burst time period also scale with mesolayer time. The shape factor in a TBL shows linear behavior with non-dimensional mesolayer length scale. In special case \( U_m / U_e = 1/2 \), is due to Iizakon and Millikan. The above predictions are supported by experimental and DNS data.

The self-similar inertial range—the region between the mesolayer and the outer layer—has been analyzed. Below this crucial point the mesolayer inner limit matches with with outer limit of wall layer. Above this crucial point the outer limit of mesolayer matches with inner limit of outer layer. The log law velocity and Reynolds in two overlap regions, above and below the critical point, have been presented. The Reynolds shear stress maxima \( \tau_{max} / \tau_w \) occurs at a point where ratio of mesolayer to outer lengths is of order \( R_e^{1/2} = (\nu/\delta u') \), and at this point DNS and experimental data predict \( U_m / U_e = 2/3 \) (where \( U_m \) = mesolayer velocity and \( U_e \) = velocity at boundary edge). The turbulent burst time period also scale with mesolayer time. The shape factor in a TBL shows linear behavior with non-dimensional mesolayer length scale. In special case \( U_m / U_e = 1/2 \), is due to Iizakon and Millikan. The above predictions are supported by experimental and DNS data.
6:02PM L22.00010 DNS of self-similar adverse pressure gradient turbulent boundary layer at incipient separation

JULIO SORIA, VASSILI KITSIOS, CALLUM ATKINSON, Monash University, JUAN SILLERO, GUILLÉM BORRELL, Universidad Politécnica de Madrid, AYSE GUNGAR, Istanbul Technical University, JAVIER JIMENEZ, Universidad Politécnica de Madrid — A direct numerical simulation of a self-similar adverse pressure gradient turbulent boundary layer (APG-TBL) flow at incipient separation has been carried out. The maximum Reynolds number based on the momentum thickness, \( Re_\delta \), reached in this DNS is 6,500. A wall-normal far-field boundary condition to effect the desired APG that will lead to the desired self-similar flow at the verge of separation has been developed. The self-similar analysis of the mean turbulent boundary layer equations yields the necessary conditions for a self-similar mean flow to exists. These conditions are tested using the DNS APG-TBL data base. First and second order statistics of the velocity across the APG-TBL are also presented in the light of the self-similar analysis results and compared to the results of a zero pressure gradient turbulent boundary layer DNS with similar mean inflow characteristics as the APG-TBL.

1The support of the ARC, NCI and Pawsey SCC funded by the Australian and Western Australian governments as well as the support of PRAE funded by the European Union are gratefully acknowledged.

---

6:15PM L22.00011 Structural features of the \( k_x^{-1} \) region of turbulent pipe flow at \( Re_\tau =3008 \)

JUNSUN AHN, HYUNG JIN SUNG, KAIST — Structural features of a turbulent pipe flow were explored by using the direct numerical simulation data at \( Re_\tau =3008 \) (Ahn et al. 2015). Based on the pre-multiplied streamwise energy spectra of the streamwise velocity fluctuations, three spectral regions were classified: the inner site, the outer site and \( k_x^{-1} \) region. The inner site was created by the self-sustaining near-wall cycle with \( \lambda_x \approx 1000 \), where \( \lambda_x \) is the streamwise wavelength. The outer site was made due to very-large-scale motions with \( \lambda_x \approx 10 \), which were generated by the streamwise pseudo-alignment of the adjacent large-scale motions. Between the inner and outer sites, the \( k_x^{-1} \) region appeared at \( y^+=90-300 \), where \( y^+ \geq 20 \) and \( \lambda_x \approx 10 \). By using the conditional averaging, self-similar structures of the streamwise velocity fluctuations structures in the \( k_x^{-1} \) region were retained, which were considered as the attached eddies proposed by Townsend (1976). In addition, the vortical structures in the \( k_x^{-1} \) region were examined by two-point correlation of the velocity components and the vortices in order to find the dominant behavior of the structures.

1This work was supported by the Creative Research Initiatives (No. 2015-001828) program of the National Research Foundation of Korea (MSIP).

---

6:28PM L22.00012 Vortex packet recovery in a turbulent boundary layer perturbed by an array of cylinders

YAN MING TAN, ELLEN LONGMIRE, Univ of Minn - Minneapolis — PIV measurements were acquired in a zero pressure gradient turbulent boundary layer (\( Re_\tau =2500 \)) perturbed by a narrowly spaced (0,2\%) array of cylinders. Two array heights were considered with one extending to the top of the log region and the other to the top of the boundary layer. Wall-parallel measurements were obtained at three locations in the log region by fixed and flying PIV. The measurement system for flying PIV moves with the flow to track the evolution of structures upstream and downstream of the array. Initially, both arrays disrupt the packets such that none are apparent. Then, packets either recover or re-initiate at some distance downstream. A packet signature was denoted by a low region of turbulent pipe flow at \( Re_\tau =3008 \)
One-dimensional model for the intracranial pulse morphological analysis during hyperventilation and \( \text{CO}_2 \) inhalation tests.\textsuperscript{1} JAIYOUNG RYU, University of California, Berkeley, XIAO HU, University of California, San Francisco, SHAWN C. SHADDEN, University of California, Berkeley — The brain’s \( \text{CO}_2 \) reactivity mechanism is coupled with cerebral autoregulation and other unique features of cerebral hemodynamics. We developed a one-dimensional nonlinear model of blood flow in the cerebral arteries coupled to lumped parameter (LP) networks. The LP networks incorporate cerebral autoregulation, \( \text{CO}_2 \) reactivity, intracranial pressure, cerebrospinal fluid, and cortical collateral blood flow models. The model was used to evaluate hemodynamic variables (arterial deformation, blood velocity and pressure) in the cerebral vasculature during hyperventilation and \( \text{CO}_2 \) inhalation tests. Tests were performed for various arterial blood pressure (ABP) representing normal and hypotensive conditions. The increase of the cerebral blood flow rates agreed well with the published measurements for various ABP measurements taken during clinical \( \text{CO}_2 \) inhalation tests. The changes in distal vasculature affected the reflected pulse wave energy, which caused the waveform morphological changes at the middle cerebral, common and internal carotid arteries. The pulse morphological analysis demonstrated agreement with previous clinical measurements for cerebral vasodilation and vasodilatation.

Mathematical modelling of blood-brain barrier failure and edema.\textsuperscript{2} SARAH WATERS, GEORGINA LANG, DOMINIC VELLA, ALAIN GORIELY, University of Oxford — Injuries such as traumatic brain injury and stroke can result in increased blood-brain barrier permeability. This increase may lead to water accumulation in the brain tissue resulting in vasogenic edema. Although the initial injury may be localised, the resulting edema causes mechanical damage and compression of the vasculature beyond the original injury site. We employ a biphasic mixture model to investigate the consequences of blood-brain barrier permeability changes within a region of brain tissue and the onset of vasogenic edema. We find that such localised changes can indeed result in brain tissue swelling and that the type of damage that results (stress damage or strain damage) depends on the ability of the brain to clear edema fluid.

Modelling Brain Temperature and Perfusion for Cerebral Cooling.\textsuperscript{3} STEPHEN BLOWERS, PRASHANT VALLURI, Institute of Materials and Processes, University of Edinburgh, IAN MARSHALL, Neuroimaging Sciences, Centre for Clinical Brain Sciences, University of Edinburgh, PETER ANDREWS, BRIDGET HARRIS, Critical Care Unit, NHS Lothian, Centre for Clinical Brain Sciences, University of Edinburgh, MICHAEL THRIPPLETON, Neuroimaging Sciences, Centre for Clinical Brain Sciences, University of Edinburgh — Brain temperature relies heavily on two aspects: i) blood perfusion and porous heat transport through tissue and ii) blood flow and heat transfer through embedded arterial and venous vasculature. Moreover brain temperature cannot be measured directly unless highly invasive surgical procedures are used. A 3D two-phase fluid-porous model for mapping flow and temperature in brain is presented with arterial and venous vessels extracted from MRI scans. Heat generation through metabolism is also included. The model is robust and reveals flow and temperature maps in unprecedented 3D detail. However, the Karmen-Kozeny parameters of the porous (tissue) phase need to be optimised for expected perfusion profiles. In order to optimise the K-K parameters a reduced order two-phase model is developed where 1D vessels are created with a tree generation algorithm embedded inside a 3D porous domain. Results reveal that blood perfusion is a strong function of the porosity distribution in the tissue. We present a qualitative comparison between the simulated perfusion maps and those obtained clinically. We also present results studying the effect of scalp cooling on core brain temperature and preliminary results agree with those observed clinically.

A dimensionless parameter for classifying hemodynamics in intracranial.\textsuperscript{4} HAFEZ ASGHARZADEH, IMAN BORAZJANI, The State University of New York at Buffalo (SUNY) — Rupture of an intracranial aneurysm (IA) is a disease with high rates of mortality. Given the risk associated with the aneurysm surgery, quantifying the likelihood of aneurysm rupture is essential. There are many risk factors that could be implicated in the rupture of an aneurysm. However, the most important factors correlated to the IA rupture are hemodynamic factors such as wall shear stress (WSS) and oscillatory shear index (OSI) which are affected by the IA flows. Here, we carry out three-dimensional high resolution simulations on representative IA models with simple geometries to test a dimensionless number (first proposed by Le et al., ASME J Biomech Eng, 2010), denoted as An, to classify the flow mode. An number is defined as the ratio of the time takes the parent artery flow transports across the IA neck to the time required for vortex ring formation. Based on the definition, the flow mode is vortex if An>1 and it is cavity if An<1. We show that the specific definition of Le et al works for sidewall but needs to be modified for bifurcation aneurysms. In addition, we show that this classification works on three-dimensional geometries reconstructed from three-dimensional rotational angiography of human subjects. Furthermore, we verify the correlation of IA flow mode and WSS/OSI on the human subject IA.

This work was supported partly by the NIH grant R03EB014860, and the computational resources were partly provided by CCR at UB. We thank Prof. Hui Meng and Dr. Jianping Xiang for providing us the database of aneurysms and helpful discussions.

Numerical simulations of post-surgical flow and thrombosis in basilar artery aneurysms. SANTHOSH SESHADHRI, Department of Neurosurgery, Medical College of Wisconsin, Milwaukee, WI, USA, MICHAEL LAWTON, Department of Neurological Surgery, University of California, San Francisco, San Francisco, CA, USA, LOIC BOUSSEL, Department of Radiology, Louis Pradel Hospital, Creatis- LRMN, Lyon, France, DAVID SALONER, Department of Radiology and Biomedical Imaging, University of California, San Francisco, San Francisco, CA, USA, VITALITY RAYZ, Department of Neurosurgery, Medical College of Wisconsin, Milwaukee, WI, USA — Surgical treatment of basilar artery aneurysms presents a major challenge since it is crucial to preserve the flow to the vital brainstem perforators branching of the basilar artery. In some cases, basilar aneurysms can be treated by clipping vessels in order to induce flow reduction and aneurysm thrombosis. Patient-specific CFD models can provide guidance to clinicians by simulating postoperative flows resulting from alternative surgeries. Several surgical options were evaluated for four basilar aneurysm patients. Patient-specific models were generated from preoperative MR angiography and MR velocimetry data and modified to simulate different procedures. The Navier-Stokes equations were solved with a finite-volume solver Fluent. Virtual contrast injections were simulated by solving the advection-diffusion equation in order to estimate the flow residence time and determine thrombus-prone regions. The results indicated on procedures that reduce intra-aneurysmal velocities and flow regions which are likely to become thrombosed. Thus CFD modeling can help improve the outcome of surgeries altering the flow in basilar aneurysms.
5:36PM L23.00008 Vortex Imprints at the Wall, But Not in the Bulk, Distinguish Ruptured from Unruptured Intracranial Aneurysms.\textsuperscript{1} NICOLE VARBLE, HUI MENG, University at Buffalo, The State University of New York — Intracranial aneurysms affect 3% of the population. Risk stratification of aneurysms is important, as rupture often leads to death or permanent disability. Image-based CFD analyses of patient-specific aneurysms have identified low and oscillatory wall shear stress to predict rupture. These stresses are sensed biologically at the luminal wall, but the flow dynamics related to aneurysm rupture requires further understanding. We have conducted two studies: one examines vortex dynamics, and the other, high frequency flow fluctuations in patient-specific aneurysms. In the first study, based on Q-criterion vortex identification, we developed two measures to quantify regions within the aneurysm where rotational flow is dominate: the ratio of volume or surface area where Q $>$ 0 vs. the total aneurysmal volume or surface area, respectively termed volume vortex fraction (VVF) and surface vortex fraction (SVF). Statistical analysis of 204 aneurysms shows that SVF, but not VVF, distinguishes ruptured from unruptured aneurysms, suggesting that once again, the local flow patterns on the wall is directly relevant to rupture. In the second study, high-resolution CFD (high spatial and temporal resolutions and second-order discretization schemes) on 56 middle cerebral artery aneurysms shows the presence of temporal fluctuations in 8 aneurysms, but such flow instability bears no correlation with rupture.

\textsuperscript{1}Support for this work was partially provided by NIH grant (R01 NS091075-01) and a grant from Toshiba Medical Systems Corp.

5:49PM L23.00009 Fluid-Structure Interaction Modeling of Intracranial Aneurysm Hemodynamics: Effects of Different Assumptions. HAMIDREZA RAJABZADEH OGHAZ, ROBERT DAMIANO, HUI MENG, University at Buffalo, State University of New York — Intracranial aneurysms (IAs) are pathological outpouchings of cerebral vessels, the progression of which are mediated by complex interactions between the blood flow and vasculature. Image-based computational fluid dynamics (CFD) has been used for decades to investigate IA hemodynamics. However, the commonly adopted simplifying assumptions in CFD (e.g. rigid wall) compromise the simulation accuracy and mask the complex physics involved in IA progression and eventual rupture. Several groups have considered the wall compliance by using fluid-structure interaction (FSI) modeling. However, FSI simulation is highly sensitive to numerical assumptions (e.g. linear-elastic wall material, Newtonian fluid, initial vessel configuration, and constant pressure outlet), the effects of which are poorly understood. In this study, a comprehensive investigation of the sensitivity of FSI simulations in patient-specific IAs is investigated using a multi-stage approach with a varying level of complexity. We start with simulations incorporating several common simplifications: rigid wall, Newtonian fluid, and constant pressure at the outlets, and then we stepwise remove these simplifications until the most comprehensive FSI simulations. Hemodynamic parameters such as wall shear stress and oscillatory shear index are assessed and compared at each stage to better understand the sensitivity of in FSI simulations for IA to model assumptions.

\textsuperscript{1}Supported by the National Institutes of Health (1R01 NS 091075-01)

6:02PM L23.00010 ABSTRACT WITHDRAWN

6:15PM L23.00011 Coupled simulation of vascular growth and remodeling, hemodynamics and stress-mediated mechanotransduction. JIACHENG WU, SHAWN C. SHADDEN, Univ of California - Berkeley — A computational framework to couple vascular G&R, blood flow simulation and stress-mediated mechanotransduction is derived for patient specific geometry. A hyperelastic constitutive relation is considered for vascular material and vessel wall is modeled via constrained mixture theory. The coupled simulation is divided into three time scales - G&R (weeks-years), hemodynamics (seconds) and stress-mediated mechanotransduction (much less than 1 second). G&R is simulated and vessel wall deformation (and tension) is computed to obtain the current vessel geometry, which defines the new boundary for blood flow. Hemodynamics are then simulated in the updated domain to calculate WSS field. A system of ODE’s is derived based on conservation law and phenomenological models to describe the signaling pathways from mechanical stimuli (WSS, wall tension) to mass production rate of vascular constituents, which, in turn, changes the kinetics of G&R. To reduce computation cost, blood flow is only simulated when G&R causes significant change to geometry, and steady state response of the ODE system for mechanotransduction is used to characterize the influence of WSS and wall tension on G&R, due to separation of three time scales.

Monday, November 23, 2015 4:05PM - 6:41PM — Session L24 Biofluids: Cardiovascular Fluid Dynamics II

4:05PM L24.00001 Fluid Dynamics of the Generation and Transmission of Heart Sounds: (2): Direct Simulation using a Coupled Hemo-Elastodynamic Method. JUNG-HEE SEO, HANI BAKHSHAEI, CHI ZHU, RAJAT MITTAL, Johns Hopkins University — Patterns of blood flow associated with abnormal heart conditions generate characteristic sounds that can be measured on the chest surface using a stethoscope. This technique of ‘cardiac auscultation’ has been used effectively for over a hundred years to diagnose heart conditions, but the mechanisms that generate heart sounds, as well as the physics of sound transmission through the thorax, are not well understood. Here we present a new computational method for simulating the physics of heart murmur generation and transmission and use it to simulate the murmurs associated with a modeled aortic stenosis. The flow in the model aorta is simulated by the incompressible Navier-Stokes equations and the three-dimensional elastic wave generation and propagation on the surrounding viscoelastic structure are solved with a high-order finite difference method in the time domain. The simulation results are compared with experimental measurements and show good agreement. The present study confirms that the pressure fluctuations on the vessel wall are the source of these heart murmurs, and both compression and shear waves likely play an important role in cardiac auscultation.

\textsuperscript{1}Supported by the NSF Grants IOS-1124804 and IIS-1344772, Computational resource by XSEDE NSF grant TG-CTS100002
4:18PM L24.00002  Computational 3D fluid-structure interaction for the aortic valve

HAOXIANG LUO, YE CHEN, Vanderbilt University, WEI SUN, Georgia Institute of Technology — Three-dimensional fluid-structure interaction (FSI) involving large deformations of flexible bodies is common in biological systems. A typical example is the heart valves. Accurate and efficient numerical approaches for modeling such systems are still lacking. In this work, we report a successful case of combining an immersed-boundary flow solver with a nonlinear finite-element solid-dynamics solver, both in-house programs, specifically for three-dimensional simulations. Based on the Cartesian grid, the viscous incompressible flow solver can handle boundaries of large displacements with simple mesh generation. The solid-dynamics solver has separate subroutines for analyzing general three-dimensional bodies and thin-walled structures composed of frames, membranes, and plates. Both geometric nonlinearity associated with large displacements and material nonlinearity associated with large strains are incorporated in the solver. The FSI is achieved through a strong coupling and partitioned approach. We have performed several benchmarking cases to validate the FSI solver. Application to the native aortic valve will be demonstrated.

1Supported by the NSF grant (CBET-1066962).

4:31PM L24.00003  Calibration of Blood Flow in Simulations via Multi-fidelity Bayesian Optimization

PARIS PERDIKARIS, Massachusetts Institute of Technology, GEORGE KARNIADAKIS, Brown University — We present a mathematical and computational framework for model inversion based on multi-fidelity information fusion and Bayesian optimization. The proposed methodology targets the accurate construction of high-dimensional response surfaces, and the effective identification of global optima while keeping the number of expensive function evaluations at a minimum. We train families of correlated surrogates on available variable fidelity data using auto-regressive stochastic models via recursive co-kriging, and exploit the resulting predictive inference schemes within a Bayesian optimization setting. The effectiveness of the proposed framework is illustrated through examples involving the calibration of outflow boundary conditions in blood flow simulations using multi-fidelity information from 3D and 1D models.

4:44PM L24.00004  Time-resolved X-ray PIV measurements of hemodynamic information of real pulsatile blood flows

HANWOOK PARK, EUNSEOP YEOM, SANG JOON LEE, Department of Mechanical Engineering, Pohang University of Science and Technology — X-ray imaging technique has been used to visualize various bio-fluid flows in the past. As a nondestructive methodology of obtaining physiologically informed data from pulsatile blood flow in a circular conduit without changes in hemorheological properties, a bypass loop is established by connecting a microtube between the jugular vein and femoral artery of a rat. Biocompatible CO\textsubscript{2} microbubbles are used as tracer particles. After mixing with whole blood, CO\textsubscript{2} microbubbles are injected into the bypass loop. Particle images of the pulsatile blood flows in the bypass loop are consecutively captured by the time-resolved X-ray PIV system. The velocity field information are obtained with varying flow rate and pulsatility. To verify the feasibility of the use of CO\textsubscript{2} microbubbles under in vivo conditions, the effects of the surrounding-tissues are also investigated, because these effects are crucial for deteriorating the image contrast of CO\textsubscript{2} microbubbles. Therefore, the velocity information of blood flows in the abdominal aorta are obtained to demonstrate the viability and usefulness of CO\textsubscript{2} microbubbles under ex vivo conditions.

1This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP) (No. 2008-0061991).

4:57PM L24.00005  Effects of a protein glycocalyx in the hemodynamics of small blood vessels

YIANNIS DIMAKOPOULOS, GEORGE DELIDAKIS, JOHN TSAMOPOULOS, Univ of Patras — Glycocalyx is a protein layer of approximate thickness 0.5\textmu m that lines vessel walls. We study the effects this layer has on the blood flow inside arterioles and venules, where the relative size of the glycocalyx is significant. To properly describe phenomena that naturally occur in blood flow, such as the inhomogeneous distribution of red blood cells and their aggregation, we use an improved viscoelastic constitutive model. The glycocalyx layer is modeled as fixed porous media. Cells cannot penetrate inside it, since its hydraulic permeability is very low, and the flow inside this layer is described by the equations for a viscous fluid with an extra Brinkman term to account for the effects the porous medium has on the flow. The closed set of equations is solved using the Finite Element method, assuming steady-state with dependence only in the r-direction. Our results are favorably compared with the in vivo velocity profiles in venules of mice produced by Damiano et al., (2004) and the formation of cell-free layer near glycocalyx. Flow inside the glycocalyx layer is found to be severely attenuated due to the low hydraulic permeability, which can have interesting implications in the transport of various substances form the blood to the tissues or in the use of shear stresses as signals for the endothelial surface cells. Finally, we simulate the transient blood flow under pulsatile conditions.

5:10PM L24.00006  Particle tracking velocimetry using echocardiographic data resolves flow in the left ventricle

KAUSHIK SMPATH, THURA T. ABD, RICHARD T. GEORGE, JOSEPH KATZ, Johns Hopkins Univ — Two dimensional contrast echocardiography was performed on patients with a history of left ventricular (LV) thrombus. The 636 x 434 pixels electrocardiograms were recorded using a GE Vivid 9E system with (MSS-D and 4V-D) probes in a 2-D mode at a magnification of 0.3 mm/pix. The concentration of 2-4.5 micron seed bubbles was adjusted to allow individually discernable traces, and a data acquisition rate of 60-90 fps kept the inter-frame displacements suitable for matching traces, and calculating vectors, but yet low enough to allow a scanning depth and width of upto 13 cm and 60 degrees respectively. Particle tracking velocimetry (PTV) guided by initial particle image velocimetry (PIV) was used to obtain the velocity distributions inside the LV with vector spacing of 3-5 mm. The data quality was greatly enhanced by implementing an iterative particle specific enhancement and tracking algorithm. Data covering 20 heart beats facilitated phase averaging. The results elucidated blood flow in the intra-ventricular septal region, lateral wall region, the apex of the LV and the mitral valve region.

5:23PM L24.00007  Data Assimilation and Propagation of Uncertainty in Multiscale Cardiovascular Simulation

DANIELE SCHIAVAZZI, ALISON MARSDEN, Stanford University — Cardiovascular modeling is the application of computational tools to predict hemodynamics. State-of-the-art techniques couple a 3D incompressible Navier-Stokes solver with a boundary circulation model and can predict local and peripheral hemodynamics, analyze the post-operative performance of surgical designs and complement clinical data collection minimizing invasive and risky measurement practices. The ability of these tools to make useful predictions is directly related to their accuracy in representing measured physiologies. Tuning of model parameters is therefore a topic of paramount importance and should include clinical data uncertainty, revealing how this uncertainty will affect the predictions. We propose a fully Bayesian, multi-level approach to data assimilation of uncertain clinical data in multiscale circulation models. To reduce the computational cost, we use a stable, condensed approximation of the 3D model build by linear sparse regression of the pressure/flow rate relationship at the outlets. Finally, we consider the problem of non-invasively propagating the uncertainty in model parameters to the resulting hemodynamics and compare Monte Carlo simulation with Stochastic Collocation approaches based on Polynomial or Multi-resolution Chaos expansions.
5:36PM L24.00008 Decoding Hemodynamics of Large Vessels via Dispersion of Contrast Agent in Cardiac Computed Tomography¹, PARASTOU ESLAMI, JUNG-HEE SEO, Johns Hopkins University, THURA T. ABD, RICHARD GEORGE, ALBERT C. LARDÖ, Johns Hopkins School of Medicine, MARCUS Y. CHEN, National Institute of Health, NHLBI, RAJAT MITTAL, Johns Hopkins University — Computed tomography angiography (CTA) has emerged as a powerful tool for the assessment of coronary artery disease and other cardiac conditions. Continuous improvements in the spatial and temporal resolution of CT scanners are revealing details regarding the spatially and temporally varying contrast concentration in the vasculature, that were not evident before. These contrast dispersion patterns offer the possibility of extracting useful information about the hemodynamics from the scans. In the current presentation, we will describe experimental studies carried out with CT compatible phantoms of coronary vessels that provide insights into the effect of imaging artifacts on the observed intracoronary contrast gradients. In addition, we will describe a series of computational fluid dynamics studies that explore the dispersion of contrast through the ascending-descending aorta with particular focus on the effect of the aortic curvature on the dispersion patterns.

¹PE is supported by the NIH Graduate Partnership Program. RM and ACL pending patents in CTA based flow diagnostics and have other significant financial interests in these technologies.

5:49PM L24.00009 Multiple equilibrium states for blood flow in microvascular networks, HALLEY POLLOCK-MUSKIN, CECILIA DIEHL, NORA MOHAMED, Olin College, NATHAN KARST, Babson College, JOHN GEDDES, BRIAN STOREY, Olin College — When blood flows through a vessel bifurcation at the microvascular scale, the hematocrits in the downstream daughter vessels are generally not equal. This phenomenon, known as plasma skimming, can cause heterogeneity in the distribution of red blood cells inside a vessel network. Using established models for plasma skimming, we investigate the equilibrium states in a microvascular network with simple topologies. We find that even simple networks can have multiple equilibrium states for the flow rates and distributions of red blood cells inside the network for fixed inlet conditions. In a ladder network, we find that for certain inlet conditions the network can have 2² observable equilibrium states where N is the number of rungs in the ladder. For ladders with even just a few rungs, the complex equilibrium curves make it seemingly impossible to set the internal state of the network by controlling the inlet flows. Microfluidic experiments are being used to confirm the model predictions.

6:02PM L24.00010 Fluid-Structure interaction modeling in deformable porous arteries¹, RANA ZAKERZADEH, PAOLO ZUNINO, Univ of Pittsburgh — A computational framework is developed to study the coupling of blood flow in arteries interacting with a poroelastic arterial wall featuring possibly large deformations. Blood is modeled as an incompressible, viscous, Newtonian fluid using the Navier-Stokes equations and the arterial wall consists of a thick material which is modeled as a Biot system that describes the mechanical behavior of a homogeneous and isotropic elastic skeleton, and connecting pores filled with fluid. Discretization via finite element method leads to the system of nonlinear equations and a Newton-Raphson scheme is adopted to solve the resulting nonlinear system through consistent linearization. Moreover, interface conditions are imposed on the discrete level via mortar finite elements or Nitsche's coupling. The discrete linearized coupled FSI system is solved by means of a splitting strategy, which allows solving the Navier-Stokes and Biot equations separately. The numerical results investigate the effects of proeoroelastic parameters on the pressure wave propagation in arteries, filtration of incompressible fluids through the porous media, and the structure displacement.

¹The fellowship support from the Computational Modeling & Simulation PhD program at University of Pittsburgh for Rana Zakerzadeh is gratefully acknowledged.

6:15PM L24.00011 Fluid Dynamics of the Generation and Transmission of Heart Sounds: (1) A Cardiothoracic Phantom Based Study of Aortic Stenosis Murmurs¹, HANI BAKHSHEES, JUNG-HEE SEO, CHI ZHU, NATHANIEL WELSH, GUILLAUME GARREAU, GASPAR TCNETTI, ANDREAS ANDREOU, RAJAT MITTAL, Johns Hopkins University — A novel and versatile cardiothoracic phantom has been designed to study the biophysics of heart murmurs associated with aortic stenosis. The key features of the cardiothoracic phantom include the use of tissue-mimetic gel to model the sound transmission through the thorax and the embedded fluid circuit that is designed to mimic the heart sound mechanisms in large vessels with obstructions. The effect of the lungs on heart murmur propagation can also be studied through the insertion of lung-mimicking material into gel. Sounds on the surface of the phantom are measured using a variety of sensors and the spectrum of the recorded signal and the streamwise variation in total signal strength is recorded. Based on these results, we provide insights into the biophysics of heart murmurs and the effect of lungs on sound propagation through the thorax. Data from these experiments is also used to validate the results of a companion computational study.

¹Authors want to acknowledge the financial supports for this study by SCH grant (IIS 1344772) from National Science Foundation.

6:28PM L24.00012 A universal number for wave reflection optimization of the mammalian cardiovascular system, NIEMA PAHLEVAN, MORTEZA GHARIB, California Institute of Technology — Quantifying the optimum arterial wave reflection and systemic arterial function is essential to the evaluation of optimal cardiovascular system (CVS) operation. The CVS function depends on both the dynamics of the heart and wave dynamics of the arterial network. Here, we are introducing a universal dimensionless number, called wave condition number ($\alpha$) that quantifies the arterial wave reflection. An in-vitro experimental approach, utilizing a unique hydraulic model was used to quantify $\alpha$ in human aorta with a wide range of aortic rigidities. Our results indicate that the optimum value of the wave condition number is 0.1 at each level of aortic rigidity. Looking into mammals of various size (from mice to elephant), our results show that the optimum wave condition number remains 0.1 and is universal among all mammals. Clinical applications and the relevancy of the wave condition number will also be discussed.

Monday, November 23, 2015 4:05PM - 6:02PM Session L25 Biofluids: Cell Interactions and Transport 304 - Jeremie Palacci, UCSD

4:05PM L25.00001 Emergent properties in experiments with synthetic microswimmers, JEREMIE PALACCI, UCSD, Department of Physics, UCSD/NYU TEAM — Self-propelled micro-particles are intrinsically out-of-equilibrium. This renders their physics far richer than passive colloids and give rise to the emergence of complex phenomena e.g. collective behavior, swarming... I will present a variety of non-equilibrium phenomena observed with experimental realization of synthetic micro swimmers: self-assembly, sensing of the environment, or effective interactions, in the absence of any potential.
4:18PM L25.00002 Chaotic mixing by microswimmers moving on quasiperiodic orbits, MIR ABBAS JALALI, Department of Astronomy, University of California, Berkeley, California 94720, USA, ATEFEH KOHSNOOD, Renewable Energy Research Institute, Palomar College, 46-4303, USA, MEHDI JABBARZADEH, Department of Mechanical Engineering, University of California, Berkeley, California 94720, USA — Life on the Earth is strongly dependent upon mixing across a vast range of scales. For example, mixing distributes nutrients for microorganisms in aquatic environments, and balances the spatial energy distribution in the oceans and the atmosphere. From industrial point of view, mixing is essential in many microfluidic processes and lab-on-a-chip operations, polymer engineering, pharmaceutics, food engineering, and biotechnology. Efficient mixing, typically characterized by chaotic advection, is hard to achieve in low Reynolds numbers because of the linear nature of the Stokes equation that governs the motion. We report the first demonstration of chaotic mixing induced by a microswimmer that strokes on quasiperiodic orbits with multi-loop turning paths. Our findings can be utilized to understand the interactions of microorganisms with their environments, and to design autonomous robotic mixers that can sweep and mix an entire volume of complex-geometry containers.

4:31PM L25.00003 Preferential Transport Theory for Beta-Amyloid Clearance from the Brain, MIKHAIL COLOMA, DAVID SCHAFFER, PAUL CHIAROT, PETER HUANG, State University of New York at Binghamton — The failure to clear beta-amyloid from the aging brain leads to its accumulation within the walls of arteries and to Alzheimer’s disease. However, there is yet another biophysical mechanism that is unknown. Previous investigations indicate that sheer flows are important to formation, suggesting that amyloid crystallization and not growth phenomena is not a simple shear-transport-limited reaction. Shear-induced crystallization of protein monomers has been proposed, yet extensional forces used in most experiments are insufficient to pull apart the hydrogen bonds that constrain protein monomers in a folded state. Other hypotheses suggest that flow induces fibrillization via alignment of protein monomers or by enhancing transport to hydrophobic interfaces. Experiments using a uniform Couette device with a rotating outer wall have shown that even minute Reynolds numbers result in enhanced crystallization kinetics. Furthermore, experiments using two highly similar proteins with different protein-protein binding affinities have provided clues towards isolation of the biophysical mechanism. Experimental evidence from the current work will be presented alongside evidence from the literature, and the relative merits of different hypotheses regarding the mechanism of shear-induced crystallization will be discussed.

4:44PM L25.00004 Probing the Biophysics behind Flow-induced Amyloid Crystallization1, SAMANTHA MCBRIDE, SEAN SANFORD, Rensselaer Polytechnic Institute, JUAN LOPEZ, Arizona State University, AMIR HIRSA, Rensselaer Polytechnic Institute — Agitation of fluid is known to induce formation of amyloidogenic species from native protein, yet the exact biophysical mechanism is unknown. Previous investigations indicate that shear flows are important to formation, suggesting that amyloid crystallization is not a simple shear-transport-limited reaction. Shear-induced crystallization of protein monomers has been proposed, yet extensional forces used in most experiments are insufficient to pull apart the hydrogen bonds that constrain protein monomers in a folded state. Other hypotheses suggest that flow induces fibrillization via alignment of protein monomers or by enhancing transport to hydrophobic interfaces. Experiments using a uniform Couette device with a rotating outer wall have shown that even minute Reynolds numbers result in enhanced crystallization kinetics. Furthermore, experiments using two highly similar proteins with different protein-protein binding affinities have provided clues towards isolation of the biophysical mechanism. Experimental evidence from the current work will be presented alongside evidence from the literature, and the relative merits of different hypotheses regarding the mechanism of shear-induced crystallization will be discussed.

4:57PM L25.00005 Optimizing an undulating magnetic microswimmer for cargo towing1, YIZHAR OR, EMILIYA GUTMAN, Technion - Israel Institute of Technology — One of the promising applications of robotic microswimmers is towing a cargo for controlled drug delivery, micro-surgery or tumor detection. This capability has been demonstrated by the magnetically-actuated microswimmer of Dreyfus et al [Nature 2005] in which a red blood cell was attached to a chain of magnetic beads connected by flexible DNA links. A key question is what should be the optimal size of the magnetic tail for towing a given cargo. This question is addressed here for the simplest theoretical model of a magnetic microswimmer under planar undulations - a spherical load connected by a torsion spring to a magnitized rigid slender link. The swimmer’s dynamics is formulated assuming negligible hydrodynamic interaction and leading-order expressions for the resulting motion are obtained explicitly under small amplitude approximation. Optimal combinations of magnetic actuation frequency, torsion stiffness, and tail length for maximizing displacement or average speed are obtained. The theoretical results are compared with several reported magnetic microswimmers, and also agree qualitatively with recent results on cargo towing by screw rotation of magnetic helical tails [Walker et al, ACS Nano Letters 2015].

5:10PM L25.00006 Self-assembled controllable microswimmers, GALIEN GROSJEAN, University of Liege, GUILLAUME LAGUBEAU, University of Santiago, Chile, ALEXIS DARRAS, GEOFFROY LUMAY, MAXIME HUBERT, NICOLAS VANDEVALLE, University of Liege — Because they cause a deformation of the interface, floating particles interact. In particular, identical particles attract each other. To counter this attraction, particles possessing a large magnetic moment $m$ are used. When $m$ is perpendicular to the surface, dipole-dipole interaction is repulsive. This competition of forces can lead to the spontaneous formation of organized structures. By using submillimetric steel spheres for which $m \propto B$, interdistances in the system can be precisely tuned. Here, we deform these self-assembly by adding a horizontal contribution $m_x$ to the magnetic moment. Time reversal symmetry is broken in the system, leading to locomotion at low Reynolds number. Moreover, swimming direction depends on the orientation of field, meaning that swimming trajectories can be finely controlled. This allows to understand the vibrational modes gives further informations on the dynamics of this system. Because this system forms by self-assembly, it allows miniaturization with applications such as cargo transport or solvent flows. It is highly versatile, being composed of simple passive particles and controlled by magnetic fields.

5:23PM L25.00007 Viscous constraints on squirmer microswimmers approaching suspended particles, MEHDI JABBARZADEH, HENRY C. FU, University of Nevada, Reno — Microscopic self-propelled organisms often approach other particles to capture food, mate, or find new environments. The viscous Stokes flow around these small organisms push away particles, severely hindering approach. Previously, we investigated approach hydrodynamics by modeling a swimming organism as a sphere pushed by a constant force towards a force-free spherical target particle. We measured approach efficiency by examining how far the swimmer will be pushed by a constant force towards a force-free spherical target particle. We measured approach efficiency by examining how far the swimmer can approach targets only when the squirmer parameter is less than 1; for values larger than 1, the swimmer cannot get close to the target.
MINJUN KIM, Drexel University, HENRY FU, University of Nevada, Reno — We investigate magnetic microswimmers actuated by a rotating magnetic field that may be useful for drug delivery, micro-surgery, or diagnostics in human body. For modular swimmers, assembly and disassembly requires understanding the interactions between the swimmer and other modules in the fluid. Here, we discuss possible mechanisms for a frequency-dependent attraction/repulsion between a three-bead, achiral swimmer and other magnetic particles, which represent modular assembly elements. We first investigate the hydrodynamic interaction between a swimmer and nearby particle by studying the Lagrangian trajectories in the vicinity of the swimmer. Then we show that the magnetic forces can be attractive or repulsive depending on the spatial arrangement of the swimmer and particle, with a magnitude that decreases with increasing frequency. Combining magnetic and hydrodynamic effects allows us to understand the overall behavior of magnetic particles near the swimmer. Interestingly, we find that the frequency of rotation can be used to control when the particle can closely approach the swimmer, with potential application to assembly.

Magnetic microswimmers: Controlling particle approach through magnetic and hydrodynamic interaction, FARSHAD MESHKATI, University of Nevada, Reno, U KEI CHEANG, MINJUN KIM, Drexel University, HENRY FU, University of Nevada, Reno — We investigate magnetic microswimmers actuated by a rotating magnetic field that may be useful for drug delivery, micro-surgery, or diagnostics in human body. For modular swimmers, assembly and disassembly requires understanding the interactions between the swimmer and other modules in the fluid. Here, we discuss possible mechanisms for a frequency-dependent attraction/repulsion between a three-bead, achiral swimmer and other magnetic particles, which represent modular assembly elements. We first investigate the hydrodynamic interaction between a swimmer and nearby particle by studying the Lagrangian trajectories in the vicinity of the swimmer. Then we show that the magnetic forces can be attractive or repulsive depending on the spatial arrangement of the swimmer and particle, with a magnitude that decreases with increasing frequency. Combining magnetic and hydrodynamic effects allows us to understand the overall behavior of magnetic particles near the swimmer. Interestingly, we find that the frequency of rotation can be used to control when the particle can closely approach the swimmer, with potential application to assembly.

I. Introduction

A. Objectives

B. Research Significance

II. Materials and Methods

A. Swimmer Description

B. Experimental Setup

C. Data Analysis

III. Results

A. Hydrodynamic Interaction

B. Magnetic Force

IV. Discussion

A. Implications for Drug Delivery

B. Future Directions

References

Monday, November 23, 2015 4:05PM - 6:19PM –
Session L26 Biofluids: Complex Fluids: Locomotion and Rheology

4:05PM L26.00001 A numerical investigation into the effects of fluid rheology and stroke kinematics on swimming alga cells in complex fluids, CHUANBIN LI, UC Davis — It is observed in experiments that when the fluid viscosity or elasticity is changed, Chlamydomonas reinhardtii exhibits changes in both flagellar kinematics and the swimming speed. In order to understand the effects of rheology on both gait and swimming performance, we develop a computational model of the swimmer. We use flagellar strokes fit from experimental data to set up a constrained system, determining the forces on the swimmer and its swimming velocity. Our approach to simulating the swimming behavior demonstrates low computational costs even in three dimensions. In our simulations, stroke patterns and fluid rheologies are changed separately, so that we can dissect the contributions of stroke kinematics of the alga and the fluid environment, which can not be achieved with experiments.

4:18PM L26.00002 Running and tumbling with E. coli in polymeric solutions1, ALISON PATTESON, Department of Mechanical Engineering, SEAS, University of Pennsylvania, ARVIND GOPINATH, Department of Physics and Astronomy, Haverford College, PAULO ARRATIA, Department of Mechanical Engineering, SEAS, University of Pennsylvania — Bacteria commonly utilize a run-and-tumble swimming behavior to navigate through complex environments such as mucus in the lungs or digestive system. This swimming behavior has been extensively studied in water-like fluids; yet, investigations on the role of particles or polymers in the ambient fluid on the run-and-tumble behavior are limited. Here, we experimentally investigate the swimming dynamics of E. coli in polymeric solutions. We find that small amounts of polymer drastically change the run-and-tumble behavior of E. coli cells, significantly enhancing translational diffusion and reducing rotational diffusion. The average cell velocity increases with polymer concentration (and viscosity) and the mean run times are enhanced. By varying polymer molecular weight and visualizing interactions between single E. coli and fluorescently-stained DNA-polymer molecules, we show that enhanced translation is a result of two mechanisms: (1) suppression of cell wobbling due to elasticity and (2) enhancement of run times due to viscosity. Our results show that the transport of chemotactic cells can be independently modified by viscosity and elasticity.

4:31PM L26.00003 Modular microrobot for swimming in heterogeneous environments, U KEI CHEANG, Drexel University, FARSHAD MESHKATI, HENRY FU, University of Nevada, Reno, MINJUN KIM, Drexel University, DREXEL UNIVERSITY TEAM, UNIVERSITY OF NEVADA, RENO TEAM — One of the difficulties in navigating in vivo is to overcome many types of environments. This includes blood vessels of different diameters, fluids with different mechanical properties, and physical barriers. Inspired by conventional modular robotic systems, we demonstrate modular microrobots using magnetic particles as the modular units to change size and shape through docking and undocking. Much like the vast variety of microorganisms navigating many different bio-environments, modular microswimmers have the ability to dynamically adapt different environments by reconfiguring the swimmers’ physical characteristics. We model the docking as magnetic assembly and undocking mechanisms as deformation by hydrodynamic forces. We characterize the swimming capability of the modular microswimmer with different size and shapes. Finally, we demonstrate modular microrobotics by assembling a three-bead microswimmer into a nine-bead microswimmer, and then disassemble it into several independently swimming microswimmers.

4:44PM L26.00004 Characterization of undulatory locomotion in granular media, ZHIWEI PENG, The University of British Columbia, ON SHUN PAK, Santa Clara University, GWYNN ELFRING, The University of British Columbia — Undulatory locomotion is ubiquitous in nature, from the swimming of flagellated microorganisms in biological fluids, to the slithering of snakes on land, or the locomotion of sandfish lizards in sand. Analysis of locomotion in granular materials is relatively less developed compared with fluids partially due to a lack of validated force models but a recently proposed resistive force theory (RFT) in granular media has been shown to modeling the locomotion of a swimming organism. Here we are using this model to investigate the swimming characteristics of an undulating slender filament of both finite and infinite length. For infinite swimmers, similar to results in viscous fluids, the sawtooth waveform is found to be optimal for propulsion speed at a given power consumption. We also compare the swimming characteristics of sinusoidal and sawtooth swimmers with swimming in viscous fluids. More complex swimming dynamics emerge when the assumption of an infinite swimmer is removed. In particular, we characterize the effects of drifting and pitching in terms of propulsion speed and efficiency for a finite sinusoidal swimmer. The results complement our understanding of undulatory locomotion and provide insights into the effective design of locomotive systems in granular media.

1This work was supported by NSF-DMR-1104705 and NSF-CBET-1437482
in-situ.\textsuperscript{1} KELLY CONNELLY, ETHAN YOUNG, JEAN-PIERRE HUBSCHMANN, JEFF ELDREDGE, PIROUZ KAVEHPOUR, None — The vitreous humor is a viscoelastic gel-like fluid that fills and maintains the structure of the eye. Changes in the structure of the network of macromolecules in vitreous occurs naturally during ageing causing pathological conditions such as retinal tears that may lead to blindness. Vitrectomy surgery is a common procedure to remove problematic vitreous from the eye, but must be carefully performed to prevent iatrogenic retinal tears. Minimizing invasiveness and surgical time depends on the viscoelastic behavior of vitreous as it flows from the eye out through a small gauge needle. Rheology has been used to correlate relevant viscoelastic fluid properties with the macromolecular structure previously using parallel plate rheometer geometries, and now with a new patented probe geometry. This improves upon plate geometries because measurements are in situ, so removal of the vitreous from the eye is not necessary. Creep rheological experiments with the probe indicate a region of elastic behavior exists at shorter timescales and steady state apparent viscosity at longer timescales. In-situ creep tests advance understanding of how macromolecular structure alters viscoelasticity, which may allow better predictions of fluid flow during vitrectomy in the future.

\textsuperscript{1}This material is based upon work supported by the National Science Foundation Graduate Research Fellowship Program under Grant No. DGE-1144087

5:10PM L26.00006 Role of elasticity on the Rheological Response of the Uterus Tissue NARIMAN ASHRAFI KHORASANI, Young Researchers and Elites Club, Science and Research Branch, Islamic Azad University, PARASTOO PIROOZRAM, PNU — N. Khorasani and P. piroozram Department of Mechanical Engineering, Payame Noor University, 19395-3697, Tehran, Iran, The effect of uterus tissue viscoelasticity on its internal pressure is explored. The tissue of the uterus is presented by a linear viscoelastic model with two major time constants. A proper user defined function is developed and incorporated in the simulation software, to represent the model. The geometry of the uterus is separately modeled. It is found that viscoelasticity of the tissue which can be controlled and altered by change the concentration can directly affect its internal pressure. It is also observed that the pressure decreases as the moisture of the tissue is increased. The study is repeated for several practical conditions and parameters pertaining to the viscoelasticity of the tissue are evaluated.

5:23PM L26.00007 Two-Point Particle Tracking Microrheology of Nematic Complex Fluids MANUEL GOMEZ-GONZALEZ, JUAN C. DEL ALAMO, University of California, San Diego — Many biological and technological complex fluids exhibit tight microstructural alignment that confers them nematic mechanical properties. However, current microrheological methods are unable to characterize the rheological response of nematic complex fluids along different directions. In this talk, we present a novel directional two-point particle-tracking microrheology method (D2PTM) that allows to measure the viscoelasticity of nematic complex fluids. We establish the theoretical foundation for D2PTM by analyzing the motion of a probing microscopic particle embedded in a nematic complex fluid, and the mutual hydrodynamic interactions between pairs of distant particles. From this analysis, we generalize the formulation of two-point particle tracking microrheology for nematic complex fluids. We test the new D2PTM formulation by simulating the motion of groups of particles undergoing Brownian motion in a nematic complex fluid with prescribed directional shear moduli. Lastly, we illustrate the experimental application of the new technique by measuring nematic F-actin solutions. These experiments constitute the first microrheological measurement of shear moduli in an anisotropic soft material.

5:36PM L26.00008 Local aggregation characteristics of microscale blood flows EF-STATHIOS KALIVITOS, Department of Mechanical and Engineering and Material Science and Engineering, Cyprus University of Technology, JOSEPH M. SHERWOOD, Department of Bioengineering, Imperial College London, JONATHAN DUSTING, None, STAVROULA BALABANI, Department of Mechanical Engineering, UCL — Erythrocyte aggregation (EA) is an important aspect of microvascular flows affecting blood flow and viscosity. Microscale blood flows have been studied extensively in recent years using computational and microfluidic based approaches. However, the relationship between the local structural characteristics of blood and the velocity field has not been quantified. We report simultaneous measurements of the local velocity, aggregation and haematocrit distributions of human erythrocytes flowing in a microchannel. EA was induced using Dextran and flows were imaged using brightfield microscopy. Local aggregation characteristics were investigated using statistical and edge-detection image processing techniques while velocity profiles were obtained using PIV algorithms. Aggregation intensity was found to strongly correlate with local variations in velocity in both the central and wall regions of the channel. The edge detection method showed that near the side wall large aggregates are associated with high local velocities and low local shear rates. In the central region large aggregates occurred in regions of low velocity and high erythrocyte concentration. The results demonstrate the combined effect of haematocrit and velocity distributions on local aggregation characteristics.

5:49PM L26.00009 Spatially and temporally resolved quantification of endothelial cell modification in response to shear stress LORI LAMBERT, University of Nebraska - Lincoln, IRAKLIS PIPINOS, TIMOTHY BAXTER, University of Nebraska Medical Center, RICHARD LEIGHTON, SRI, TIMOTHY WEI, University of Nebraska - Lincoln — This talk contains a report on in vivo measurements made over a confluent layer of bovine endothelial cells in a microchannel. The ultimate goal of the experiments is to understand and model cellular response to fluid stresses and the ensuing transport across the endothelial layer. High resolution \( \mu \) PTV measurements were made to quantify the cellular response to steady shear rates of 5, 10 and 20 dynes/cm\(^2\). Surface topography, shear and pressure distributions were calculated from sets of velocity fields made in planes parallel to the wall. For each experiment, measurements were made in three-hour intervals for eighteen hours. To validate the methodology, the pH of the medium was varied so that the health of the cells would vary. Clear differences in topography and cell orientation were found. Implications for future experiments and research will be discussed.

6:02PM L26.00010 The Effects of Hemodynamic Shear Stress on Stemness of Acute Myelogenous Leukemia (AML)\textsuperscript{1} ANDREW RADDATZ, URSULA, TRIANTAFILLU, YONGHYUN (JOHN) KIM, University of Alabama — Cancer stem cells (CSCs) have recently been identified as the root cause of tumors generated from cancer cell populations. This is because these CSCs are drug-resistant and have the ability to self-renew and differentiate. Current methods of culturing CSCs require much time and money, so cancer cell culture protocols, which maximize yield of CSCs are needed. It was hypothesized that the quantity of Acute myelogenous leukemia stem cells (LSCs) would increase after applying shear stress to the leukemia cells based on previous studies with breast cancer in bioreactors. The shear stress was applied by pumping the cells through narrow tubing to mimic the in vivo bloodstream environment. In support of the hypothesis, shear stress was found to increase the amount of LSCs in a given leukemia population. 

\textsuperscript{1}This work was supported by NSF REU Site Award 1358991
To interpret our results we will compare the changed morphology to related insects. We discuss the implications of the insects' location on the system matches the relaxation time of the visco-elastic liquid.

**Monday, November 23, 2015 4:05PM - 6:41PM**

**Session L27 Biofluids: Insect Flight: Dynamics and Control**

**4:05PM L27.00001 Initial Observations of Fruit Fly’s Flight with its b1 Motor Neuron Altered**

ROBERT NOEST, Z. JANE WANG, JAMES MELFI JR, Cornell University — Recently we have suggested that one of the fly’s 17 steering muscles, the first basalar muscle (b1) is responsible for maintaining flight stability [1]. To test this, we compare the flight behavior of normal flies with genetically modified flies whose motor neuron to the b1 muscle is silenced. We report our initial observation of the difference and similarity between these two lines supplied by Janelia Farm. We also discuss the basic question for quantifying flight, what makes a good flier? Reference: [1] S Chang and ZJ Wang, Predicting fruit fly’s sensing rate with insect flight simulations, PNAS, (2014)

**4:18PM L27.00002 Flight stability analysis under changes in insect morphology**

ROBERT NOEST, Z. JANE WANG, Cornell University — Insect have an amazing ability to control their flight, being able to perform both fast aerial maneuvers and stable hovering. The insect’s neural system has developed various mechanism by which it can control these flying feats, but we expect that insect morphology is equally important in facilitating the aerial control. We perform a computational study using a quasi-steady instantaneous flapping flight model which allows us to freely adapt the insect’s morphological parameters. We picked a fruit fly as the basis for the body shape and wing motion, and study the effect of changes to the morphology for a range of wing stroke amplitudes. In this manner, we determine the preferred regions along the chord for sensor placement and for estimating chordwise loads to inform control decisions in flight.

**4:31PM L27.00003 Sparse Sensing of Aerodynamic Loads on Insect Wings**

KRITHIKA MANOHAR, Applied Mathematics, Univ of Washington — We investigate how insects use sparse sensors on their wings to detect aerodynamic loading and wing deformation using a coupled fluid-structure model given periodically flapping input motion. Recent observations suggest that insects collect sensor information about their wing deformation to inform control actions for maneuvering and rejecting gust disturbances. Given a small number of point measurements of the chordwise aerodynamic loads from the sparse sensors, we reconstruct the entire chordwise loading using a signal processing technique that reconstructs a signal from a small number of measurements using l1 norm minimization of sparse modal coefficients in some basis. We compare reconstructions from sensors randomly sampled from probability distributions biased toward different regions along the wing chord. In this manner, we determine the preferred regions along the chord for sensor placement and for estimating chordwise loads to inform control decisions in flight.

**4:44PM L27.00004 Wing-pitch modulation in maneuvering fruit flies is explained by an interplay between aerodynamics and a torsional spring**

TSEVI BEATUS, ITAI COHEN, Physics Department, Cornell University, Ithaca NY — While the wing kinematics of many flapping insects have been well characterized, understanding the underlying physiological mechanisms that determine these kinematics is still a challenge. Two of the main difficulties arise from the complexity of the interaction between a flapping wing and its own unsteady flow, as well as the intricate mechanics the insect wing-hinge, which is among the most complicated joints in the animal kingdom. These difficulties call for the application of reduced-order approaches. Here, we model the torques exerted by the wing-hinge along the wing-pitch axis of maneuvering fruit flies as a damped torsional spring with elastic and damping coefficients as well as a rest angle. Furthermore, we model the air flows using simplified quasi-static aerodynamics. Our findings suggest that flies take advantage of the passive coupling between aerodynamics and the damped torsional spring to indirectly control their wing-pitch kinematics by modulating the spring damping and elastic coefficients. These results, in conjunction with the previous literature, indicate flies can accurately control their wing-pitch kinematics on a sub-wing-beat time-scale by modulating all three effective spring parameters on longer time-scales.
These changes in the wing motion are critically related to the aerodynamic force generation, which will be discussed in detail.

With the ground, the flapping amplitude of elytron is reduced and the hindwing changes its flapping angular velocity during upstroke of a tree. It is first found that the elytron which is flapped passively due to the motion of hindwing also has non-negligible wing-kinematic parameters. With the ground, the flapping amplitude of elytron is reduced and the hindwing changes its flapping angular velocity during upstroke of a tree. It is first found that the elytron which is flapped passively due to the motion of hindwing also has non-negligible wing-kinematic parameters.

We consider two types of take-off flights; that is, one is the take-off from a flat ground and the other is from a vertical rod mimicking a branch and use the modified direct linear transform algorithm for the reconstruction of measured wing motions. To realize different take-off conditions, forces were measured through the use of strain gauges and 2D phase-locked particle image velocimetry (PIV) was used to visualize the flow generated from flapping. The PIV results show that circulation of the leading edge vortices (LEVs) is attenuated when bristled wings are used. However, improved drag reduction is observed in the bristled wings. Aerodynamic efficiency variation with Re will be discussed.

In this study, we closely examined the kinematics as well as aerodynamics of backward takeoff in dragonflies and compared them to those of forward takeoff. High-speed videography and accurate 3D surface reconstruction techniques were employed to extract details of the wing and body motions as well as deformations during both flight modes. While the velocities of both forward and backward flights were similar, the body orientation as well as the wing kinematics showed large differences. Our results indicate that by tilting the stroke plane angle of the wings as well as changing the orientation of the body relative to the flight path, dragonflies control the direction of the flight like a helicopter. In addition, our detailed analysis of the flow in these flights shows important differences in the wake capture phenomena among these flight modes.

This research was supported by the National Science Foundation (CBET 1512071).

5:10PM L27.00006 Flapping of Insectile Wings, YANGYANG HUANG, EVA KANSO, University of Southern California — Insects use flight muscles attached at the base of the wings to produce impressive wing flapping frequencies. Yet the effects of muscle stiffness on the performance of insect wings remain unclear. Here, we construct an insectile wing model, consisting of two rigid wings connected at their base by an elastic torsional spring and submerged in an oscillatory flow. The wing system is free to rotate and flap. We first explore the extent to which the flyer can withstand roll perturbations, then study its flapping behavior and performance as a function of spring stiffness. We find an optimal range of spring stiffness that results in large flapping amplitudes, high force generation and good storage of elastic energy. We conclude by conjecturing that insects may select and adjust the muscle spring stiffness to achieve desired movement. These findings may have significant implications on the design principles of wings in micro air-vehicles.

5:23PM L27.00007 Kinematics and Aerodynamics of Backward Flying Dragonflies\textsuperscript{1}, AYOJDEJI BODE-OKE, SAMANE ZEYGHAMI, HAIBO DONG, University of Virginia — Highly maneuverable insects such as dragonflies have a wide range of flight capabilities; precise hovering, fast body reorientations, sideways flight and backward takeoff are only a few to mention. In this research, we closely examined the kinematics as well as aerodynamics of backward takeoff in dragonflies and compared them to those of forward takeoff. High-speed videography and accurate 3D surface reconstruction techniques were employed to extract details of the wing and body motions as well as deformations during both flight modes. While the velocities of both forward and backward flights were similar, the body orientation as well as the wing kinematics showed large differences. Our results indicate that by tilting the stroke plane angle of the wings as well as changing the orientation of the body relative to the flight path, dragonflies control the direction of the flight like a helicopter. In addition, our detailed analysis of the flow in these flights shows important differences in the wake capture phenomena among these flight modes.

\textsuperscript{1}This work is supported by NSF CBET-1313217

5:36PM L27.00008 Beetle wings are inflatable origami, RUI CHEN, Georgia Institute of Technology, JING REN, SIQIN GE, University of Chinese Academy of Sciences, DAVID HU, Georgia Institute of Technology — Beetles keep their wings folded and protected under a hard shell. In times of danger, they must unfold them rapidly in order for them to fly to escape. Moreover, they must do so across a range of body mass, from 1 mg to 10 grams. How can they unfold their wings so quickly? We use high-speed videography to record wing unfolding times, which we relate to the geometry of the network of blood vessels in the wing. Larger beetles have longer unfolding times. Modeling of the flow of blood through the veins successfully accounts for the wing unfolding speed of large beetles. However, smaller beetles have anomalously short unfolding times, suggesting they have lower blood viscosity or higher driving pressure. The use of hydraulics to unfold complex objects may have implications in the design of micro-flying air vehicles.

5:49PM L27.00009 Comparative study of solid and bristled wings in flapping flight of tiny insects\textsuperscript{1}, CHRISTOPHER TERRILL, ARVIND SANTHANAKRISHNAN, Oklahoma State University — Small insects such as thrips that are less than 1 mm in size fly at Reynolds numbers (Re) on the order of 10 and use wing-wing interaction during flapping. In this interaction, referred to as ‘clap-and-fling’, the wings come in close contact with each other at the end of upstroke and rotate about the trailing edge during start of downstroke. The wings of these tiny insects consist of an array of bristles as opposed to a solid membrane. The goal of this study is to examine the effects of bristled wings on aerodynamic force generation and flow structures compared to solid wings. We used an experimental model for the study in which two model wings were prescribed to move along a simplified 2D representation of clap-and-fling kinematics. Forces were measured through the use of strain gauges and 2D phase-locked particle image velocimetry (PIV) was used to visualize the flow generated from flapping. The PIV results show that circulation of the leading edge vortices (LEVs) is attenuated when bristled wings are used. However, improved drag reduction is observed in the bristled wings. Aerodynamic efficiency variation with Re will be discussed.

\textsuperscript{1}This research was supported by the National Science Foundation (CBET 1512071).

6:02PM L27.00010 Aerodynamics of a beetle in take-off flights\textsuperscript{1}, BOOGEON LEE, HYUNGMIN PARK, Seoul National University, SUN-TAE KIM, Agency for Defense Development — In the present study, we investigate the aerodynamics of a beetle in its take-off flights based on the three-dimensional kinematics of inner (hindwing) and outer (elytron) wings, and body postures, which are measured with three high-speed cameras at 2000 fps. To track the highly deformable wing motions, we distribute 21 morphological markers and use the modified direct linear transform algorithm for the reconstruction of measured wing motions. To realize different take-off conditions, we consider two types of take-off flights; that is, one is the take-off from a flat ground and the other is from a vertical rod mimicking a branch of a tree. It is first found that the elytron which is flapped passively due to the motion of hindwing also has non-negligible wing-kinematic parameters. With the ground, the flapping amplitude of elytron is reduced and the hindwing changes its flapping angular velocity during up and downstrokes. On the other hand, the angle of attack on the elytron and hindwing increases and decreases, respectively, due to the ground. These changes in the wing motion are critically related to the aerodynamic force generation, which will be discussed in detail.

\textsuperscript{1}Supported by the grant to Bio-Mimetic Robot Research Center funded by Defense Acquisition Program Administration (UD1300701D).
6:15PM L27.00011 Interaction of the elytra and hind wing of a rhinoceros beetle (*Trogopulos dichotomus*) during a take-off mode
d by SEUNGYOUNG OH, SEHYEONG OH, HAECHEON CHOI, BOOGEON LEE, HYUNGMIN PARK, Seoul National University, SUN-TAE KIM, Agency for Defense Development — The elytra are a pair of hardened wings that cover the abdomen of a beetle to protect beetles hind wings. During the take-off, these elytra open and flap in phase with the hind wings. We investigate the effect of the elytra flapping on beetles aerodynamic performance. Numerical simulations are performed at Re=10,000 (based on the wingtip mean velocity and mean chord length of the hind wing) using an immersed boundary method. The simulations are focused on a take-off, and the experiment kinematics used is directly obtained from the experimental observations using high speed cameras. The simulation result shows three-dimensional vortical structures generated by the hind wing of the beetle and their interaction with the elytra. The presence of elytra has a negative effect on the lift generation by the hind wings, but the lift force on the elytra themselves is negligible. Further discussions on the elytra - hind wing interaction will be provided during the presentation.

1 Supported by UD1300701D

6:28PM L27.00012 Numerical and Experimental Investigation of Flow Structures During Insect Flight
d by CAMLI BADRYA, JAMES D. BAEDEER, University of Maryland, Aerospace department, College Park — Insect flight kinematics involves complex interplay between aerodynamics structural response and insect body control. Features such as cross-coupling kinematics, high flapping frequencies and geometrical small-scales, result in experiments being challenging to perform. In this study OVERTURNS, an in-house 3D compressible NavierStokes solver is utilized to simulate the simplified kinematics of an insect wing in hover and forward flight. The flapping wings simulate the full cycle of wing motion, i.e., the upstroke, downstroke, pronation and supination. The numerical results show good agreement against experimental data in predicting the lift and drag over the flapping cycle. The flow structures around the flapping wing are found to be highly unsteady and vortical. Aside from the tip vortex on the wings, the formation of a prominent leading edge vortex (LEV) during the up/down stroke portions, and the shedding of a trailing edge vortex (TEV) at end of each stroke were observed. Differences in the insect dynamics and the flow features of the LEV are observed between hover and forward flight. In hover the up and downstroke cycles are symmetric, whereas in forward flight, these up and downstroke are asymmetric and LEV strength varies as a function of the kinematics and advance ratio.

1 This work was supported by the Micro Autonomous Systems and Technology (MAST) CTA at the University of Maryland
2 Graduate Research assistant
3 Professor

Monday, November 23, 2015 4:05PM - 6:41PM –
Session L28 Surface Tension Effects: Interfacial Phenomena 309 - Linda Smolka, Bucknell University

4:05PM L28.00001 Oil capture from a water surface by a falling sphere by LINDA SMOLKA, CLARE MCLAUGHLIN, Bucknell University, THOMAS WITELSKI, Duke University — When a spherical particle is dropped from rest into an oil lens that floats on top of a water surface, a portion of the oil adheres to the sphere. Once the sphere comes to rest at the subsurface, the oil forms a pendant drop that remains attached in equilibrium to the sphere effectively removing oil from the water surface. Best fit solutions of the Laplace equation to experimental profiles are used to investigate the parameter dependence of the radius of curvature and the filling and contact angles at the three-phase contact line of the pendant drop for spheres with different wetting properties, densities and radii. The volume of oil captured by a sphere increases with a sphere’s mass and diameter. However, lighter and smaller spheres capture more oil relative to their own volume than do heavier and larger spheres (scaling with the sphere mass 

1 The authors wish to acknowledge the support of the National Science Foundation Grant Nos. DMS-0707755 and DMS-0968252.

4:18PM L28.00002 Keeping warm with fur in cold water: entrainment of air in hairy surfaces by ALICE NASTO, MARIANNE REGLI, PIERRE-THOMAS BRUN, Massachusetts Institute of Technology, CHRISTOPHE CLANET, PPMM, ESPCI / LabHyX, Ecole Polytechnique, ANETTE HOSOI, Massachusetts Institute of Technology — Instead of relying on a thick layer of body fat for insulation as many aquatic mammals do, fur seals and otters trap air in their dense fur for insulation in cold water. Using a combination of model experiments and theory, we rationalize this mechanism of air trapping underwater for thermoregulation. For the model experiments, hairy surfaces are fabricated using laser cut molds and casting samples with PDMS. The hairy texture as a network of capillary tubes, the imbibition speed of water into the hairs is obtained through a balance of hydrostatic pressure and viscous stress. In this scenario, the bending of the hairs and capillary forces are negligible. The maximum diving depth that can be achieved before the hairs are wetted to the roots is predicted from a comparison of the diving speed and imbibition speed. The amount of air that is entrained in hairy surfaces is greater than what is expected for classic Landau-Levich-Derjaguin plate plunging. A phase diagram with the parameters from experiments and biological data allows a comparison of the model system and animals.

4:31PM L28.00003 Simulation of drop tipstreaming in a flow focusing geometry with a hybrid numerical method by MICHAEL BOOTY, MICHAEL SIEGEL, New Jersey Institute of Technology, JACEK WROBEL, Tulane University, QIMING WANG, York University, Toronto — A hybrid numerical method that is designed to resolve the influence of soluble surfactant in the limit of large bulk Peclet number is used to simulate tipstreaming from a drop in the Stokes flow regime in a simple, axisymmetric flow focusing geometry. Examples are presented of the influence of flow focusing on tip-thread formation and tipstreaming, and of the influence of various dimensionless flow parameters on flow dynamics.
4:44PM L28.00004 Drinking in Space: The Capillary Beverage Experiment, ANDREW WOLLMAN, MARK WEISLOGEL, Portland State University, RYAN JENSEN, IRPI LLC, JOHN GRAF, DONALD PETTIT, SCOTT KELLY, KJELL LINDGREN, NASA Johnson Space Center, KIMIYA YUI, Japan Aerospace Exploration Agency (JAXA) — A selection from as many as 50 different drinks including coffees, teas, and fruit smoothies are consumed daily by astronauts aboard the International Space Station. For practical reasons, the drinks are generally sipped through straws inserted in sealed bags. We present the performance of a special cup designed to allow the drinking operation in much the same manner as on earth, only with the role of gravity replaced by the combined effects of surface tension, wetting, and special container geometry. One can finally ‘smell the coffee.’ Six so-called Space Cups are currently in orbit as part of the Capillary Beverage Experiment which aims to demonstrate specific passive control of poorly wetting aqueous capillary systems through a fun mealt ime activity. The mathematical fluid mechanical design process with full numerical simulations is presented alongside experimental results acquired using a drop tower and low-g aircraft before complete characterization aboard the Space Station. Astronaut consumption is both humorous and informative, but the insightful experimental results of the potable space experiment testify to the prospects of new no-moving-parts capillary solutions for certain water-based life support operations aboard spacecraft.

4:57PM L28.00005 Bow and Oblique Shock Formation in Soap Film, ILDOO KIM, SHREYAS MANDRE, AAKASH SANE, Brown University — In recent years, soap films have been exploited primarily to approximate two-dimensional flows while their three-dimensional character is relatively unattended. An example of the three-dimensional character of the flow in a soap film is the observed Marangoni shock wave when the flow speed exceeds the wave speed. In this study, we investigated the formation of bow and oblique shocks in soap films generated by wedges with different deflection angles. When the wedge deflection angle is small and the film flows fast, oblique shocks are observed. When the oblique shock cannot exists, bow shock is formed upstream the wedge. We characterized the oblique shock angle as a function of the wedge deflection angle and the flow speed, and we also present the criteria for transition between bow and oblique Marangoni shocks in soap films.

5:10PM L28.00006 Blowing a liquid curtain, H. LHUISSIER, IUSTI, CNRS & Aix-Marseille Université, France, B. BRUNET, MSC, CNRS & Université Paris Diderot, France, S. DORBOLO, GRASP, FNRS & Université de Liège, Belgium — We study the response of a steady free-falling liquid curtain perturbed by focused air jets blowing perpendicularly against it. Asymmetric and symmetric perturbations are applied by using either a single pulsed jet or two identical steady jets facing each other. The response strongly depends on the air jet deflection rate, the rate and the duration of the erturbations. For pulsed asymmetric perturbations of increasing amplitude, sinuous wave, drop ejection, bubble ejection, and hole opening are successively observed. For steady symmetric perturbations, a steady hole forms downstream in the wake. For this latter case, we present a model for the curtain thickness and the location of the hole in the wake which compares favorably to the experiments providing the perturbation is small enough (jet stagnation pressure smaller than curtain stagnation pressure) and the liquid viscosity is negligible.

5:23PM L28.00007 Dynamic wetting failure in surfactant solutions, CHEN-YU LIU, ERIC VANDRE, University of Minnesota, MARCIO CARVALHO, PUC-Rio, SATISH KUMAR, University of Minnesota — The influence of insoluble surfactants on dynamic wetting failure during displacement of Newtonian fluids in a rectangular channel is studied in this work. A hydrodynamic model for steady Stokes flows of dilute surfactant solutions is developed and evaluated using three approaches: (i) a one-dimensional (1D) lubrication-type approach, (ii) a novel hybrid of a 1D description of the receding phase and a 2D description of the advancing phase, and (iii) an asymptotic theory of Cox. Steady-state solution families in the form of macroscopic contact angles as a function of the capillary number are demonstrated. Contact line pinning stresses for new surfactant types are found to be significantly increased. When the air is the primary fluid to advance into the mixture, contact line pinning is observed near the contact line by thinning the air film without significantly changing the capillary-pressure gradients there. As consequence, the limit points shift to lower capillary numbers and the onset of wetting failure is promoted. The model predictions are then used to interpret decades-old experimental observations concerning the influence of surfactants on air entrainment. The hybrid modeling approach developed here can readily be extended to more complicated geometries where a thin air layer is present near a contact line.

5:36PM L28.00008 Dewetting of microliquid film via vapor-mediated Marangoni effect, SEUNGHO KIM, HO-YOUNG KIM, Seoul National University — It is generally conceived that water film residing on a hydrophilic layer is more stable than one on a hydrophobic layer because of the alcohol drop near the film. It is because the concentration gradients of alcohol vapor deposited on water give rise to the Marangoni effect, which pulls the water film away from the alcohol drop. We term this behavior the vapor-mediated Marangoni dewetting. Two different film flow types are observed depending on the thickness of film. For a thin water film, a bulk film recedes from the center where the alcohol vapor concentration is the highest but leaves a thin fringe film. The nanoscale fringe film is then dried, leading to continuous growth of the hole. For a thick water film, no nanoscale fringe films are observed, but the hole growth is limited to a certain radius. The maximum hole radius in the thick film regime is determined by the balance between the hydrostatic pressure and the Marangoni stress. We visualize such novel film dewetting dynamics with a high-speed camera and characterize their salient features by combining experimental and scaling analysis.

5:49PM L28.00009 Dynamic contact angle at the advancing contact line on an accelerating vertical rod, TAKAHIRO ITO, KENTA YOKOI, Nagoya University, KENJI KATOH, TATSURO WAKIMOTO, Osaka City University, YASUFUMI YAMamoto, Kansai University, YOSHIIKU TSUJI, Nagoya University — The motion of the contact line is a critical boundary condition for the prediction of the interface geometry in the wetting or dewetting processes. The establishment of the contact angle is essential to predict the motion of the contact line, since, following the previous theories, the contact angle can be expressed by the parameters including the force of the contact line. In this study, the dynamic contact angle is investigated both experimentally and numerically for a transient state in which a vertical glass rod penetrating the free surface of the test liquid (ethylene glycol) is submerged into the liquid with acceleration motion. The experimentally measured contact angle was smaller than those obtained in the steady state by considering the contact line velocity. The deviation is found to increase with the acceleration of the rod. Numerical simulation showed the acceleration term affects only the surface profiles for 11, >10^−2 with x the distance from the rod surface and l, the Laplace length. This indicates that the inertia effect would not responsible for the deviation of the dynamic contact angle.

6:02PM L28.00010 A fluid-dynamical model for the “anti-surfactant” behaviour of salt solutions, STEPHEN WILSON1, JUSTIN CONN, DAVID PRITCHARD, BRIAN DUFFY, Department of Mathematics and Statistics, University of Strathclyde, PETER HALLING, Department of Pure and Applied Chemistry, University of Strathclyde, KHELLIL SEFIANE, School of Engineering, University of Edinburgh — We formulate and analyse a novel fluid-dynamical model for the flow of a solution with a free surface on which surface tension acts. This model, which uses the concept of surface excess, can describe both classical surfactants and aqueous salt solutions. These latter solutions have the anomalous property that in thermodynamic equilibrium the surface tension increases with increasing salt concentration, i.e., so-called “anti-surfactant” behaviour. We demonstrate the utility of the model by considering the stability of a deep layer of initially quiescent fluid, and identify the possibility of an anti-surfactant instability driven by Marangoni effects.

1Supported by Leverhulme Trust Research Fellowship RF-2013-355 “Small Particles, Big Problems: Understanding the Complex Behaviour of Nanofluids”.


1. When the two reactants of an A + B → C reaction are brought into contact, a reaction front is formed. The reaction occurs in time due to the interdiffusion of A and B. The properties of such fronts are well studied in reaction-diffusion systems where no flow can affect the dynamics. Here we consider horizontal aqueous solutions where the three species A, B, and C can affect the surface tension of the solution, thereby driving Marangoni flows. The resulting dynamics is studied by numerically integrating the incompressible Navier-Stokes equations coupled to reaction-diffusion-convective equations for the three chemical species. We show that the front propagation cannot be predicted anymore on the sole basis of the reaction-diffusion properties as was still possible in the presence of buoyancy-driven flows around such fronts. We relate this observation to the structure of the Marangoni-driven flow and propose a classification of the convective effects on A + B → C reaction fronts as a function of the different Marangoni numbers quantifying the effect of each species on the surface tension.

2. Surfactant transport arises in many natural or industrial settings. Examples include lipid tear layers in the eye, pulmonary surfactant replacement therapy, or industrial coating flows. Surfactant transport is often studied in the context of thin film flows for which the lubrication approximation hold, we demonstrate how the knowledge of this free surface flow field provides sufficient information to reconstruct the surfactant concentration. Flows driven by the surface tension gradient which arises as a consequence of surfactant concentration inhomogeneity, also known as Marangoni-driven flows, have attracted the attention of fluid dynamists for several decades and has led to the development of sophisticated models and the undeniable advancement of the understanding of such flows. Yet, experimental confirmation of these models has been hampered by the difficulty in reliably and accurately measuring the surfactant concentration and its temporal evolution. In this contribution, we propose a methodology which may help shed some light on surfactant transport at the surface of thin liquid films. The surface stress induced by surfactant concentration induces a flow at the free surface which is visible and measurable. In the context of thin film flows for which the lubrication approximation hold, we demonstrate how the knowledge of this free surface flow field provides sufficient information to reconstruct the surfactant tension field. From the surface tension and an assumed equation of state, the local surfactant concentration can also be calculated and other transport parameters such as the surfactant surface diffusivity indirectly inferred. In this contribution, the proposed methodology is tested with synthetic data generated by the forward solution of the governing partial differential equations in order to illustrate the feasibility of the algorithm and highlight numerical challenges.

3. The interaction of balanced abyssal ocean flow with submarine topography is expected to generate lee waves, which can carry energy into the ocean interior, as well as local turbulent mixing near the boundary. We report observations of lee waves and turbulence, and measurements of the mixing rate, in laboratory experiments with a topographic ridge towed through a density stratification. The experiments span three parameter regimes including linear lee waves, nonlinear wave radiation and an evanescent regime in which wave radiation is not possible. The stratification evolves from an initially uniform buoyancy frequency to a mixed boundary layer and pycnocline. Full field density measurements provide the depth-dependence of energy loss to turbulent mixing. The ratio of the local mixing in the turbulent wake and remote mixing by wave radiation takes a nearly constant value that is not sensitive to the stratification or dynamical regime; the average value $q_{mix} = 0.90 \pm 0.06$ in the linear lee wave regime, is three times larger than that assumed in parameterizations of internal wave-induced mixing in the ocean. The results suggest that mixing by local nonlinear mechanisms close to abyssal ocean topography may be much greater than remote mixing by lee waves.

4. In the linear lee wave regime, is three times larger than that assumed in parameterizations of internal wave-induced mixing in the ocean. The results suggest that mixing by local nonlinear mechanisms close to abyssal ocean topography may be much greater than remote mixing by lee waves.
4:44PM L29.00004 Transient triadic instability of internal gravity wave a new track to turbulence in the lee of a topography1. JEAN-MARC CHOMAZ, GATAN LERISSON, Laboratoire d’Hydrodynamique, LadHyX, CNRS-Ecole Polytechnique — Internal gravity waves in a continuously stratified fluid propagate energy away from the source and are particularly important to understand the ocean mixing. We study the stability of different gravity wave inhomogeneous in space through fully non-linear direct numerical simulation and linear global stability analysis and transient growth computation. In particular the steady flow over an arbitrary topography is computed using the selective frequency algorithm and the stability properties of the flow are analysed applying the Arnoldi-Krylov technique applied to the direct linearisation to retrieve the global spectrum and to the direct-adjoint technique to optimize transient growth. We show that, both exponential and transient growths are linked to the triadic instability of the lee wave but correspond respectively to the large scale branch and the small scale branch also known as the parametric subharmonic instability. Interpretation of this surprising selection principle is proposed in term of the absolute and convective instability of the 2D periodic planar wave (see the presentation by G. Lerisson et al.)

1Support of DGA and Labex LaSIPS are acknowledged

4:57PM L29.00005 Impact of a mean current on internal tide energy dissipation at the critical latitude. OCANE RICHER, JEAN-MARC CHOMAZ, CAROLINE MULLER, ladhyx, cole polytechnique — In many regions of the ocean, the abyssal flow is dominated by tidal flow. A large fraction of the tidal energy input in the ocean is dissipated via the generation of internal waves above rough topography. Idealised simulations suggest that internal tide energy is transferred and dissipated at small scales by the formation of a resonant triad between near-inertial waves, internal tides and subharmonics waves. Furthermore, the energy dissipation is enhanced at the critical latitude (28.8°), corresponding to the Parametric Subharmonic Instability (PSI). In the ocean, the presence of background flow, for instance due to the passage of a mesoscale eddy, can modify energy transfer mechanisms and the amount of energy dissipation. In this study, we investigate the generation and dissipation of internal tides in the presence of a background flow. We use a high-resolution two-dimensional nonhydrostatic numerical model (the MITgcm), with realistic multiscale topography representing the Brazil basin region. The purpose of this study is to understand the impact of the mean flow on the generation and dissipation of tidal waves. Our particular interest is how the maximum of energy dissipation at the critical latitude is impacted by the mean flow.

5:10PM L29.00006 Determining Pressure and Velocity Fields from Experimental Schlieren Data1. FRANK M. LEE, MICHAEL R. ALLSHOUSE, P.J. MORRISON, HARRY L. SWINNEY, UT Austin — Internal gravity waves generated by tidal flow over bottom topography in the ocean are important because they contribute significantly to the energy composition of the ocean. Determination of the instantaneous internal wave energy flux requires knowledge of the pressure and velocity fields, each of which is difficult to measure in the ocean or the laboratory. However, the density perturbation field can be measured using a laboratory technique known as “synthetic schlieren.” We present an analytical method for deducing both the pressure and velocity fields from the density perturbation field. This yields the instantaneous energy flux of linear internal waves. Our method is verified in tests with data from a Navier-Stokes direct numerical simulation. The method is then applied to laboratory schlieren data obtained for the conditions in the numerical simulations.

1MRA and HLS were supported by ONR. FML and PJM supported by DOE contract DE-FG02-04ER-54742.

5:23PM L29.00007 Internal Wave Generation by Tide-Topography Interactions in the Presence of a Vertically Sheared Background Current. KEVIN LAMB, Univ of Waterloo, MICHAEL DUNPHY, IFREMER — Vertically sheared background currents alter the generation of internal waves by tide-topography interactions by introducing asymmetries and minimum phase speeds for horizontally propagating vertical modes. A linear theory for internal wave generation for arbitrary stratifications and background currents, restricted to lie above two-dimensional topography, has been developed. Rotational affects have not been considered. In this talk the results of fully nonlinear simulations of the internal wave generation process will be presented and compared with predictions of the linear theory. We have found that the theory gives good predictions for wide subcritical ridges.

5:36PM L29.00008 Experiments on topographies lacking tidal conversion1. LEO MAAS, NIOZ Royal Netherlands Institute for Sea Research, Texel, the Netherlands, ALEXANDRE PACI, CNRM-GAME, METEO FRANCE & CNRS, Toulouse, France, BING YUAN, IMAU, Utrecht University, Utrecht, the Netherlands — In a stratified sea, internal tides are supposedly generated during the surface tide’s passage over the irregular bottom. We here demonstrate this in a lab-experiment. However, for any such topography, subsequently changing the surface tide’s frequency does lead to tidal conversion. The upshot of this is that a tidal wave passing over an irregular bottom is for a substantial part trapped to this irregularity, and only partly converted into freely propagating internal tides.

1Financially supported by the European Communities 7th Framework Programme HYDRALAB IV

5:49PM L29.00009 Transient Growth in Internal Solitary Waves. KARL HELFRICH, Department of Physical Oceanography, Woods Hole Oceanographic Institution, Woods Hole, MA, USA, PIERRE-YVES PASSAGGIA, BRIAN WHITE, Department of Marine Sciences, University of North Carolina, Chapel Hill, NC 27599, USA — Internal solitary waves of large amplitude are common in the atmosphere and ocean and play an important role in mixing and transport. While these waves can propagate over long distances, observations suggest they are susceptible to a range of instabilities, which promote breakdown, overturning, and mixing. To gain insight into these instabilities, we consider the optimal transient growth of a family of solitary waves, which are solutions to the Dubreil-Jacotin-Long (DJL) equation for increasing phase speed and varying background stratification. Optimal initial disturbances are computed by means of direct-adjoint iterations of the Navier-Stokes system in the Boussinesq approximation. The most amplified disturbances resemble Kelvin-Helmholtz instabilities and are localized near the bottom of the wave, where the Richardson number is minimum, and are maximized for short time horizons. The optimal transient growth of these perturbations is shown to increase with the phase speed. Implications for breakdown and mixing will be discussed.
practical laboratory experiments in which ZVI could be observed. We verify our analytically obtained leading order inner and outer layer solutions with numerical simulations. In addition, maps in ZVI, we analyze their structures with matched asymptotic expansions, assuming viscosity determines the magnitude and thickness of the vortices, resulting in “vortex self-replication” that fills the fluid with turbulent vortices. To understand the role of baroclinic critical layers create linearly unstable vortex layers, which roll-up into vortices. Those vortices excite new baroclinic critical layers, which form new generations of density and velocity fields. The flow, confined from the top by a horizontal boundary, is a lighter alcohol-water mixture injected from a nozzle into quiescent heavier salt-water fluid. The injected flow is turbulent with Taylor Reynolds number about 75. We compare a set of length scales that characterize the mixing properties of our turbulent stratified shear flow including Thorpe Length $L_T$, Ozmidov Length $L_O$, and Ellison Length $L_E$.

1Supports from DGA and Labex LaSiPS are acknowledged

6:15PM L29.00011 Instability and mixing of stratified shear layers forced by internal wave strain. ALEXIS KAMINSKI, JOHN TAYLOR, DAMTP, University of Cambridge — Mixing of the stably-stratified ocean interior plays an important role in determining the vertical stratification and the transport of key biological and geochemical tracers. Shear instabilities are thought to be a key mechanism in triggering small-scale mixing in the ocean, and a large literature is devoted to examining the stability properties of steady, parallel stratified shear flows. However, geophysical flows are frequently complicated by additional processes, such as internal waves, leading to variation in space and in time. Not only is the breaking of internal waves an important source of mixing, but the vertical strain caused by these waves may also impact the stability of the flows through which they propagate. Here, we idealize this process by imposing a standing wave which is spatially and temporally periodic onto a stably-stratified shear flow. We use a direct-adjoint looping method to examine the linear stability of this complicated base flow over a range of parameters in order to identify and quantify the effect of the wave strain on the overall flow stability. Direct numerical simulations are then used to examine the nonlinear evolution and subsequent mixing.

6:28PM L29.00012 Internal Wave Apparatus for Copepod Behavior Assays. S. JUNG, K.A. HAAS, D.R. WEBSTER, Georgia Tech — Internal waves are ubiquitous features in coastal marine environments and have been observed to mediate vertical distributions of zooplankton in situ. Internal waves are generated through oscillations of the pycnocline in stratified waters and thereby create fine-scale hydrodynamic cues that copepods and other zooplankton are known to sense, such as fluid density gradients and velocity gradients (quantified as shear deformation rate). The role of copepod behavior in response to cues associated with internal waves is largely unknown. Thus, a coupled quantification of copepod behavior and hydrodynamic cues will provide insight to the bio-physical interaction and the role of biological versus physical forcing in mediating organism distributions. We constructed a laboratory-scale internal wave apparatus to facilitate fine-scale observations of copepod behavior in flows that replicate in situ conditions of internal waves in a two-layer stratification. Three cases are chosen with density jump ranging between 0.75 – 1.5 kg/m$^3$. Analytical analysis of the two-layer system provides guidance of the target forcing frequency to generate a standing internal wave with a single dominate frequency of oscillation. Flow visualization and signal processing of the interface location are used to quantify the wave characteristics. A copepod behavior assay is conducted, and sample trajectories are analyzed to identify copepod response to internal wave structure.

Monday, November 23, 2015 4:05PM - 6:41PM –
Session L30 Geophysical Fluid Dynamics: Stratified Turbulence 311 - Colm-Cille Caulfield, University of Cambridge

4:05PM L30.00001 Instability of Stratified Shear Flow: Intermittency and Length Scales. ROBERT ECKE, Los Alamos National Laboratory, PHILIPPE ODIER, ENS Lyon — The stability of stratified shear flows which occur in oceanic overflows, wind-driven thermoclines, and atmospheric inversion layers is governed by the Richardson Number $Ri$, a non-dimensional balance between stabilizing stratification and destabilizing shear. For a shear flow with velocity difference $U$, density difference $\Delta \rho$ and characteristic length $H$, one has $Ri = g (\Delta \rho/H)H/U^2$. A more precise definition is the gradient Richardson Number $Ri_g = N^2/S^2$ where the buoyancy frequency $N = \sqrt{(g/\partial \rho/\partial z)}$, the mean strain $S = \partial U/\partial z$ with $\partial$ parallel to gravity and with ensemble or time averages defining the gradients. We explore the stability and mixing properties of a wall-bounded shear flow for $0.1 < Ri_g < 1$ using simultaneous measurements of density and velocity fields. The flow, confined from the top by a horizontal boundary, is a lighter alcohol-water mixture injected from a nozzle into quiescent heavier salt-water fluid. The injected flow is turbulent with Taylor Reynolds number about 75. We compare a set of length scales that characterize the mixing properties of our turbulent stratified shear flow including Thorpe Length $L_T$, Ozmidov Length $L_O$, and Ellison Length $L_E$.

4:18PM L30.00002 Baroclinic Critical Layers and the Zombie Vortex Instability (ZVI) in Stratified, Rotating Shear Flows: Where They Form and Why. MENG WANG, PATRICK HUERRE, CHUNG-HSIANG JIANG, SUYANG PEI, MARYANN RUI, PHILIP MARCUS, University of California, Berkeley — It has been found recently that baroclinic critical layers are responsible for a new finite-amplitude instability, called the Zombie Vortex Instability (ZVI), in stratified (with Brunt–Väisälä frequency $N$) flows, rotating with angular velocity $\Omega$ and shear $\sigma$. ZVI occurs via baroclinic critical layers that create linearly unstable vortex layers, which roll-up into vortices. Those vortices excite new baroclinic critical layers, which form new generations of vortices, resulting in “vortex self-replication” that fills the fluid with turbulent vortices. To understand the role of baroclinic critical layers in ZVI, we analyze their structures with matched asymptotic expansions, assuming viscosity determines the magnitude and thickness of the critical layer. We verify our analytically obtained leading order inner and outer layer solutions with numerical simulations. In addition, maps of the control parameter space (Reynolds number, $N/\Omega$ and $\sigma/\Omega$) are presented that show two regimes where ZVI occurs, and the physics that determines the boundaries of the two regimes is interpreted. The parameter map and its underlying physics provide guidance for designing practical laboratory experiments in which ZVI could be observed.
4:31PM L30.00003 Spontaneous layer formation dynamics in stratified Taylor–Couette flow\textsuperscript{1}, COLIN LECLERCQ, University of Bristol, JAMIE L. PARTRIDGE, DAMTP, University of Cambridge, PIERRE AUGIER, LEGI (Grenoble), CNRS, C.P. CAULFIELD, BP Institute and DAMTP, University of Cambridge, PAUL F. LINDEN, STUART B. DALZIEL, DAMTP, University of Cambridge, MUST COLLABORATION — The spontaneous formation of horizontal layers is a common feature of strongly and stably stratified flows and plays a major role in the dynamics of geophysical flows. However, little is known about the physical mechanism setting the depth of the layers spontaneously emerging in “stratified Taylor–Couette flow” in the annulus between a rotating inner cylinder and a fixed outer cylinder, initially filled with stably, axially and linearly stratified fluid. Using linear stability analysis, direct numerical simulations and experiments, we investigate the relative importance of primary linear instability and secondary nonlinear processes in the transient dynamics leading to the experimentally and numerically observed step-like density profile in this flow. We explore the effects of the particular form of the spin-up of the inner cylinder and initial conditions on the transient dynamics and nonlinear attractor of the flow. By better understanding the dynamics of layer formation, we are able to identify the appropriate scaling laws relating layer depth to rotation rate, initial stratification, gap width and radius ratio.

\textsuperscript{1}EPSRC programme grant EP/K034529/1

4:44PM L30.00004 Energy and water vapor transport in a turbulent stratified environment, LUCA GALLANA, Politecnico di Torino, FRANCESCA DE SANTI, MICHELE IOVIO, Politecnico di Torino, DIMEAS, RENZO RICHIARDONE, Universita' degli Studi di Torino, DANIELA TORDELLA, Politecnico di Torino, DIMEAS — We present direct numerical simulations about the transport of kinetic energy and unsaturated water vapor across a thin layer which separates two decaying turbulent flows with different energy. This interface lies in a shearless stratified environment modeled by means of Boussinesq’s approximation. Water vapor is treated as a passive scalar (Kumar et al. 2014). Initial conditions have \( F r^2 \) between 0.64 and 64 (stable case) and between -3.2 and -19 (unstable case) and \( Re_c = 250 \). Dry air is in the lower half of the domain and has higher turbulent energy, seven times higher than the energy of moist air in the upper half. In the early stage of evolution, as long as \( |Fr|^2 > 1 \), stratification plays a minor role and the flows follows closely neutral stratification mixing. As the buoyancy terms grows, \( F r^2 \sim O(1) \), the mixing process deeply changes. A stable stratification generates a separation layer which blocks the entrainment of dry air into the moist one, characterized by a relative increment of the turbulent dissipation rate compared to the local turbulent energy. On the contrary, an unstable stratification slightly enhances the entrainment. Growth-decay of energy and mixing layer thickness are discussed and compared with laboratory and numerical experiments.

5:57PM L30.00005 Layering from anticyclonic vortices in a rotating stratified medium with combined salinity and temperature effects\textsuperscript{1}, JOEL SOMMERIA, MICHAEL BURNIN\textsuperscript{2}, SAMUEL VIBOUD, LEGI/CNRS — We generate anticyclonic vortices by a fluid source in a rotating and uniformly stratified medium, a laboratory model of long lived vortex lenses in the ocean. Experiments are performed in the large Coriolis rotating platform at Grenoble, 13 m in diameter, providing previously unaccessible turbulent regimes. The other novelty is to combine temperature and salinity effects, like in meddies, vortices formed by intrusion in the Atlantic ocean of warm and salty water from the Mediterranean Sea. For both heated and unheated cases, we observe shear driven instability at the vortex periphery, leading to the emission of material filament from a large-scale m=2 instability. Heated vortices behave much the same way but with two key additions. One, prominent at early times, is that the vortex edge appears serrated around most of its circumference in the upper part of the lens. Two, clearer for later times, a staircase density profile develops above the eddy. We explain this small scale turbulence as thermal convection in the statically unstable density profile resulting from selective vertical diffusion of temperature (while salinity is less diffusive). The resulting turbulent mixing generates horizontal intrusions at the upper part of the vortex, unlike the double-diffusive instability.

\textsuperscript{1}This work has been funded by Agence Nationale de la Recherche (ANR), project 'OLA'

\textsuperscript{2}present affiliation: CSU San Marcos

5:10PM L30.00006 Turbulent mixing due to Holmboe wave instability in stratified shear flows at high Reynolds numbers\textsuperscript{1}, HESAM SALEHIPOUR, University of Toronto, COLM-CILLE CAULFIELD, BP Institute & DAMTP, University of Cambridge, W. RICHARD PELTIER, University of Toronto — We consider numerically the transition to turbulence and associated mixing in parallel stratified shear flows with hyperbolic tangent initial velocity and density distributions. When the characteristic length scale of density variation is sufficiently sharper than that of the velocity variation, this flow is primarily susceptible to Holmboe wave instability (HWI) which perturbs the interface to exhibit characteristic cusped interfacial waves. Unlike previous low-Re experimental and numerical studies, in the high-Re regime in which our DNS analyses are performed, the primary HWI triggers a vigorous yet markedly more long-lived turbulent event compared to its better known relative, the Kelvin-Helmholtz instability (KHI). HWI 'scours' the primary density interface, leading to substantial irreversible mixing and vertical transport of density displaced above and below the (robust) primary density interface which is comparable in both absolute terms and relative efficiency to the mixing associated with an equivalent KHI. Our results establish categorically that, provided the Reynolds number is high enough, shear layers with sharp density interfaces and associated locally high values of the gradient Richardson number are sites of substantial and efficient irreversible mixing.

\textsuperscript{1}H.S. is grateful to the David Crighton Fellowship from DAMTP, University of Cambridge.

5:23PM L30.00007 Mixing efficiency dependence on overturning and turbulence intensity in stratified shear flows\textsuperscript{1}, C. P. CAULFIELD, BP Institute & DAMTP, U. of Cambridge, ALI MASHAYEK, EAPS, MIT, W. R. PELTIER, Physics, U. of Toronto — It is well-known that both the total amount of irreversible mixing and its efficiency in stratified shear flows are strongly time-dependent. We consider shear layers that are susceptible to primary Kelvin-Helmholtz instabilities, developing relatively large billow overturnings that in turn are subject to various secondary instabilities which trigger turbulence transition. Valuable insights can be gained by considering the time-dependence of three characteristic length scales of the flows; the overturning Thorpe scale \( L_T \); the largest turbulence scale unaffected by stratification known as the Ozmidov scale \( L_O = \sqrt{\nu/\epsilon} \); and the Kolmogorov scale \( L_K = (\nu^3/\epsilon)^{1/4} \), where \( \epsilon \) is the kinetic energy dissipation rate, \( \nu \) is the kinematic viscosity, and \( N \) is the buoyancy frequency. Provided \( L_O/L_K \) is sufficiently large, we show that \( L_T \) first grows as the primary billow develops, but then falls rapidly as the turbulence onsets and \( L_O \) increases in turn and then decays more slowly, leading to a typical monotonic increase in the ratio \( L_O/L_T \) with time. Both the most efficient and the most vigorous mixing occurs when \( L_T \approx L_O \), which has important implications for the interpretation and modelling of real oceanic mixing events.

\textsuperscript{1}This work has been funded by the British Council grant EP/K034529/1.
5:36PM L30.00008 Biases in Thorpe scale estimates of turbulence dissipation. ALBERTO SCOTTI. Dept. of Marine Sciences, UNC-CH — The Thorpe-scale method is widely used to estimate dissipation and mixing rates in environmental stratified turbulent flows from density measurements along vertical profiles. We show that the relevant displacement scale in general is not the rms value of the Thorpe displacement, rather, the displacement field must be Reynolds decomposed to separate the mean from the turbulent component, and it is the turbulent component than ought to be used to diagnose mixing and dissipation. In shear-driven flows, the rms of the Thorpe displacement, known as the Thorpe scale is shown to be equivalent to the turbulent component of the displacements, and we show that the Thorpe scale approximates the Ozmidov scale, or, which is the same, the Thorpe scale is the appropriate scale to diagnose mixing and dissipation. However, when mixing is driven by the available potential energy of the mean flow (convective-driven mixing), we show that the Thorpe scale is (much) larger than the Ozmidov scale.

1Work supported by ONR under grant N00014-09-1-0288

5:49PM L30.00009 An analysis of diapycnal mixing efficiency in stably stratified turbulent flows. AMRAPALLI GARANAIK, SUBHAS KARAN VENAYAGAMOORTHY, Colorado State University, DEREK STRETCH, University of KwaZulu-Natal — In order to estimate turbulent diapycnal mixing in stably stratified flows such as in oceanic flows, two key quantities are required namely the diapycnal mixing efficiency $R_f$ and the dissipation rate of turbulent kinetic energy $\epsilon$. The focus of this study is to investigate the variability of $R_f$ by considering oceanic turbulence data obtained from microstructure profiles in conjunction with data from laboratory experiments and DNS. The analysis of the field data was performed on turbulent patches which were identified using the Thorpe sorting method for potential temperature. The turbulent kinetic energy $K$ contained within a turbulent patch was inferred based on the flow regime following the methodology proposed by Mater and Venayagamoorthy (Physics of Fluids, 26, 036601, 2014). The analysis shows that high mixing efficiency can persist at high buoyancy Reynolds numbers ($Re_b = \epsilon/\nu^2$, where $N$ is buoyancy frequency and $\nu$ is the kinematic viscosity), contrary to the notion that mixing efficiency decreases in a universal manner beyond $Re_b > 100$. These findings clearly show that $Re_b$ based parameterizations that are obtained from low-Reynolds number experimental/DNS studies are not universal and/or appropriate for geophysical flows.

6:02PM L30.00010 The Efficiency of Deep and Abyssal Ocean Turbulent Mixing. ALI MASHAYEK, MIT, COLM CAULFIELD, University of Cambridge, RAFFAELE FERRARI, MIT, MAXIM NIHURASHIN, University of Tasmania, RICHARD PELTIER, University of Toronto — Turbulent mixing produced by breaking of internal waves in the deep ocean plays a primary role in the climate through exerting a control upon the upwelling of deep dense waters formed at high latitudes, thereby driving the global ocean overturning circulation. A key parameter used to characterize turbulent mixing in observations, climate models, and global energy budgets is the ‘efficiency’ of mixing, here defined as the ratio of the portion of the tide and wind energy input into the deep ocean that is invested in mixing, to the portion viscously dissipated into heat. Efficiency is conventionally assumed to be a constant of approximately twenty percent. Here we show that it varies significantly in the abyssal ocean, and that mixing is predicted to be most efficient, reaching values as high as fifty percent near topographic features which host vigorous wave generation and breaking. This result suggests a more accurate closure of the bulk ocean energy budget, a goal lying at the heart of understanding the role of the ocean circulation in climate and one towards which the oceanographic community has been striving for decades.

6:15PM L30.00011 Plankton dynamics in thermally-stratified free-surface turbulence. SALVATORE LOVECCHIO, ALFREDO SOLDATI, University of Udine — Thermal stratification induced by solar heating near the ocean-atmosphere interface influences the transfer fluxes of heat, momentum and chemical species across the interface. Due to thermal stratification, a region of large temperature gradients (thermocline) may form with strong consequences for the marine ecosystem. In particular, the thermocline is believed to prevent phytoplankton from reaching the well-lit surface layer, where they can grow through the process of photosynthesis. In this paper, we use a DNS-based Eulerian-Lagrangian approach to examine the role of stratification on phytoplankton dynamics in thermally-stratified free-surface turbulence. We focus on gyrotactic self-propelled phytoplankton cells, considering different stratification levels (quantified by the Richardson number) and different gyro tactic re-orientation times. We show that the modulation of turbulent fluctuations induced by stable stratification has a strong effect on the orientation and distribution of phytoplankton, possibly leading to trapping of some species within the thermocline. Specifically, we observe the appearance of a depletion layer just below the free-surface as stratification increases, accompanied by a reduction in the vertical stability of phytoplankton cells.

6:28PM L30.00012 An affordable and accurate conductivity probe for density measurements in stratified flows. MARCO CARMINATI, Dipartimento di Elettronica, Informazione e Bioingegneria, Politecnico di Milano, PAOLO LUZZATTO-FEGIZ, Department of Mechanical Engineering, UC Santa Barbara — In stratified flow experiments, conductivity (combined with temperature) is often used to measure density. The probes typically used can provide very fine spatial scales, but can be fragile, expensive to replace, and sensitive to environmental noise. A complementary instrument, comprising a low-cost conductivity probe, would prove valuable in a wide range of applications where resolving extremely small spatial scales is not needed. We propose using micro-USB cables as the actual conductivity sensors. By removing the metallic shield from a micro-B connector, 5 gold-plated microelectrodes are exposed and available for 4-wire measurements. These have a cell constant $\sim 550 \text{m}^{-1}$, an intrinsic thermal noise of at most $30 \text{pA/Hz}^{1/2}$, as well as sub-millisecond time response, making them highly suitable for many stratified flow measurements. In addition, we present the design of a custom electronic board (Arduino-based and Matlab-controlled) for simultaneous acquisition from 4 sensors, with resolution (in conductivity, and resulting density) exceeding the performance of typical existing probes. We illustrate the use of our conductivity-measuring system through stratified flow experiments, and describe plans to release simple instructions to construct our complete system for around $200.

Monday, November 23, 2015 4:05PM - 6:41PM —
Session L31 Waves: Nonlinear Waves and Turbulence 312 - Eric Falcon, CNRS, Paris, France
4:05PM L31.00001 Observation of resonant interactions among gravity surface waves1, ERIC FALCON, Université Paris Diderot, MSC, CNRS, Paris, France, FELICIEN BONNEFOY, Ecole Centrale de Nantes, LHEEA, CNRS, Nantes, France, FLORENCE HAUDIN, Université Paris Diderot, MSC, CNRS, Paris, France, GUILLAUME MICHEL, BENOIT SEMIN, Ecole Normale Supérieure, LPS, CNRS, Paris, France, THOMAS HUMBERT, SEBASTIEN AUMAITRE, CEA-Saclay, Sphynx, CNRS, GIF-sur-Yvette, France, MICHAEL BERHANU, Université Paris Diderot, MSC, CNRS, Paris, France — We experimentally study resonant interactions of gravity surface waves in a large basin. We generate two oblique sinusoidal swells of tunable angle, steepness and frequency ratio. These waves interact each other and give birth to a resonant wave whose properties (growth rate and resonant response curve) are fully characterized. A phase locking between waves is also evidenced. All our experimental results are found in good quantitative agreement with 4-wave interaction theory of gravity waves with no fitting parameter. Slightly off-resonance experiments are also reported. For stronger wave steepness, departures from the weakly nonlinear theory are observed. Our results thus strongly extend previous experimental results performed more than 50 years ago.

1This work was supported by ANR Turbulon 12-BS04-0005.

4:18PM L31.00002 Role of the basin boundary conditions in gravity wave turbulence, MICHAEL BERHANU, MSC, CNRS, Université Paris Diderot, LUC DEKE, Scripps Institution of Oceanography, University of California San Diego, BENJAMIN MIQUEL, University of Colorado at Boulder, PABLO GUTIERREZ, DFI-FCFM-Universidad de Chile, TIMOTHEE JAMIN, MSC, CNRS, Université Paris Diderot, BENOIT SEMIN, LPS, Ecole Normale Supérieure, ERIC FALCON, MSC, CNRS, Université Paris Diderot, FELICIEN BONNEFOY, LHEEA, Ecole Centrale de Nantes — Gravity wave turbulence is studied in a large wave basin where irregular waves are generated unidirectionally. The role of the basin boundary conditions (absorbing or reflecting) are investigated. To that purpose, an absorbing sloping beach opposite to the wavemaker can be replaced by a reflecting vertical wall. The wave field properties depend strongly on these boundary conditions. Unidirectional waves propagate before to be damped by the beach whereas a multi directional wave field is observed with the wall. In both cases, the wave spectrum scales as a frequency-power law with an exponent that increases continuously with the forcing amplitude up to a value close to -4. We have also studied freely decaying gravity wave turbulence in the closed basin. No self-similar decay of the spectrum is observed, whereas its Fourier modes decay first as a time power law due to nonlinear mechanisms, and then exponentially due to linear viscous damping. We estimate the linear, nonlinear and dissipative time scales to test the time scale separation. Using the mean energy flux from the initial decay of wave energy, the Kolmogorov-Zakharov constant of the weak turbulence theory is evaluated experimentally for the first time.

4:31PM L31.00003 Experiments on linear waves propagating over a turbulent background, PABLO GUTIERREZ, Univ de Chile, SEBASTIEN AUMAITRE, CEA Saclay, France, CLAUDIO FALCON, Univ de Chile — We are interested in what happens to a linear wave propagating on the surface of a turbulent flow. This problem is studied with two experimental procedures. First, we excite surface-resonant-modes by means of the periodic motion of a container with water. When we impose turbulent motion in the bulk of the water, we observe a clear reduction in the resonance peaks. This represent a simple way to identify turbulent fluid motion as a source of dissipation for surface waves. The second procedure is to locally excite a wave at a given frequency, and to study its propagation along the container. Here again, when there is an underlying turbulent flow, we observe the enhancement of wave dissipation. Also, we observe a shift in the wavenumber through larger values, which can be understand as a random scattering of the wave on the turbulent structures.

4:44PM L31.00004 Non local resonances in weak turbulence of gravity-capillary water waves, NICOLAS MORDANT, QUENTIN AUBOURG, LEGI, Université Grenoble Alpes, France — We investigate experimentally the statistical properties of wave turbulence of surface waves on water. In the limit of weak non linear energy an energy cascade in scale is predicted by the Weak Turbulence Theory. Energy transfers are predicted to occur among resonant waves. We use a Fourier Transform Profilometry technique that provides a 2D measurement of the water surface deformation that is resolved in time and scale. The principle is to project a pattern on the surface of water which diffuses light thanks to the addition of a Titanium oxyde powder. The pattern can then be inverted to provide the elevation of the water surface. Our wave tank is 70 cm long and we investigate waves that lie in the vicinity of the capillary-gravity crossover with frequencies between 1Hz and 100 Hz. We compute 3-wave correlations so that to study the non linear coupling and the energy transfers among resonant waves. We observe a 3-wave non linear coupling which is dominantly unidirectional and non local in scale: a low frequency gravity wave can be coupled to 2 high frequency capillary waves. We will also discuss the importance of approximate resonances in the wave coupling.

4:57PM L31.00005 Faraday waves on time-dependent domains, MAHDI GHADIRI, ROUSLAN KRECHETNIKOV, University of Alberta — Faraday wave patterns — standing waves which form on the free fluid surface due to its vertical vibration – have been frequently used as a testbed for new theories and ideas. As part of the recent effort to understand dynamics and evolution on time-dependent spatial domains, in this talk we will present experimental investigation on how Faraday wave patterns respond to the domain deformation. In our experimental setup of a vibrating water container with controlled moving walls, the characteristics of the free surface patterns are measured using the Fourier transform profilometry technique, which allows us to get accurate time history of patterns three-dimensional landscape. Our study reveals, at the experimental level, how patterns transform in response to the domain dynamics on various length- and time-scales.

5:10PM L31.00006 Propagation of nonlinear waves over submerged step: wave separation and subharmonic generation, EDUARDO MONSALVE, Laboratoire de Physique et Mécanique des Milieux Hétérogènes, ESPCI - ParisTech, AGNES MAUREL, Institut Langevin, ESPCI - ParisTech, VINCENT PAGNEUX, Laboratoire d’Acoustique de l’Université de Paris, PHILIPPE PETITJEANS, Laboratoire de Physique et Mécanique des Milieux Hétérogènes, ESPCI - ParisTech — Water waves can be described in simplified cases by the Helmholtz equation. However, even in these cases, they present a high complexity, among which their dispersive character and their nonlinearities are the subject of the present study. Using Fourier Transform Profilometry, we study experimentally the propagation of waves passing over a submerged step. Because of the small water depth after the step, the wave enters in a nonlinear regime. In the shallow water region, the second harmonic leads to two types of waves: bound waves which are slaves of the fundamental frequency with wavenumber $2k(\omega)$, and free waves which propagate according to the usual dispersion relation with wavenumber $k(\omega)$. Because of the presence of these two waves, beats are produced at the second harmonic with characteristic beat length. In this work, for the first time we extended this analysis to the third and higher harmonics. Next, the region after the step is limited to a finite size L with a reflecting wall. For certain frequencies and L-values, the spectral component becomes involved, with the appearance of sub harmonics. This regime is analyzed in more details, suggesting a transition to a chaotic and quasi-periodic wave behavior.
5:23PM L31.00007 Vortex kinematics and dynamics in deep-water breaking waves†, KEN MELVILLE, NICHOLAS PIZZO, LUC DEIKE, Scripps Institution of Oceanography, UC San Diego — Surface wave breaking can be modeled as a transitional process from irrotational to turbulent flow. Thus the introduction of vorticity across the range of inertial to dissipative scales is of great significance for the kinematics and dynamics of breaking. In this presentation, we review laboratory experimental data showing the introduction of coherent vortices at breaking and present an impulsive force model (just half of the smoke ring problem) that predicts the coherent structure in terms of the energy dissipated by breaking. We then apply this model, supported by DNS of breaking, to predict the distribution of the energy lost from the wave field between turbulence and the coherent vorticity. The models and available experimental and numerical data are consistent with inertial scaling of the wave energy dissipated by breaking.

†Funded by NSF and ONR.

5:36PM L31.00008 Experimental observation of steady inertial wave turbulence in deep rotating flows†, EHUD YAROM, ERAN SHARON, Hebrew Univ of Jerusalem — We present experimental evidence of inertial wave turbulence in deep rotating fluid. Experiments were performed in a rotating cylindrical water tank, where previous work showed statistics similar to 2D turbulence (specifically an inverse energy cascade). Using Fourier analysis of high resolution data in both space (3D) and time we show that most of the energy of a steady state flow is contained around the inertial wave dispersion relation. The nonlinear interaction between the waves is manifested by the widening of the time spectrum around the dispersion relation. We show that as the Rossby number increases so does the spectrum width, with a strong dependence on wave number. Our results suggest that in some parameters range, rotating turbulence velocity field can be represented as a field of interacting waves (wave turbulence). Such formalism may provide a better understanding of the flow statistics.

†This work was supported by the Israel Science Foundation, Grant No. 81/12

5:49PM L31.00009 On the structure of turbulence dissipation rate under unsteady breaking waves†, MORTEZA DERAKHTI, JAMES KIRBY, Univ of Delaware — During the last decade, extensive laboratory and field measurements have been conducted for the estimation and parameterization of the turbulence dissipation rate under unsteady breaking waves, showing a large amount of scatter depending on the selected estimation, type and scale of the considered breaking waves. To further elucidate the physical processes involved in turbulence generation and dissipational processes, Deraghti & Kirby, JFM, (2014) examined shear- and bubble-induced dissipation. They used a 3D VOF-based Navier-Stokes solver extended to incorporate entrained bubble populations using an Eulerian-Eulerian formulation for a poly-disperse bubble phase, and found that the total bubble-induced dissipation accounts for more than 50% of the total dissipation in the breaking region (the results were presented at DFD13, Abstract 001799). In this presentation, we will examine the 3D distribution of breaking-induced turbulent kinetic energy and dissipation rate during the active breaking period. The role of breaking-induced vortical structures in the transport of turbulent motions will be addressed as well. Finally, the accuracy of the available analytic scaling relations of the intensity and depth dependence of wave breaking turbulence dissipation rate will be discussed.

†NSF, Physical Oceanography Program, grant OCE-1435147

6:02PM L31.00010 Interactions of steep and breaking waves with winds and solid bodies, ZIXUAN YANG, LIAN SHEN, University of Minnesota — The interactions of steep and breaking waves with winds and solid bodies at sea surface is important to many problems in ocean science and engineering. In this study, we perform large-eddy simulations using a finite-difference code with high-performance parallel computing. The air-water interface is captured using a coupled level set and volume of fluid method. A sharp interface immersed boundary method is applied to capture the effect due to the presence of solid bodies. A wall layer model is employed to address high Reynolds numbers. A numerical wave generator is utilized to accurately produce waves with specified parameters. The results are validated for a number of canonical problems, and the performances of different wall-layer model schemes are evaluated using a priori and a posteriori tests. Based on the simulation data, the flow details and interaction mechanisms are analyzed.

6:15PM L31.00011 Effect of progressive surface waves on near-surface transport of scalars by turbulent wind, LIAN SHEN, University of Minnesota, DI YANG, University of Houston — The presence of progressive water surface waves plays a vital role in the air-sea exchange of scalar quantities, such as water vapor and heat. The periodic surface curvature and motions of the waves impose considerable disturbance to the turbulence boundary layer flow over the wave surface, affecting the transport of both momentum and scalars. In this study, the effect of surface waves on scalar transport is investigated using direct numerical simulation (DNS). The DNS solver uses pseudo-spectral and finite-difference schemes for the flow and scalar fields, with spatial discretization carried out on a moving wave-fitted computational grid to capture the surface wave effect. The results show considerable variations in the statistics of the scalar transport for different phase speeds of the waves that correspond to different development stages of wind-generated ocean waves. Based on the DNS data, several turbulent closure models for RANS modeling of scalar transport are evaluated using a priori test.

6:28PM L31.00012 Rogue waves for a system of coupled derivative nonlinear Schrödinger equations†, HIU NING CHAN, The University of Hong Kong, BORIS MALOMED, Tel Aviv University, KWOK WING CHOW, The University of Hong Kong — Previous works in the literature on water waves have demonstrated that the fourth-order evolution of gravity waves in deep water will be governed by a higher order nonlinear Schrödinger equation. In the presence of two wave trains, the system is described by a higher order coupled nonlinear Schrödinger system. Through a gauge transformation, these evolution equations are reduced to a coupled derivative nonlinear Schrödinger system. The goal here is to study rogue waves, unexpectedly large displacements from an equilibrium position, through the Hirota bilinear transformation theoretically. The connections between the onset of rogue waves and modulation instability are investigated. The range of cubic nonlinearity allowing rogue wave formation is elucidated. Under a finite group velocity mismatch between the two components, the existence regime for rogue waves is extended as compared to the case with a single wave train. The amplification ratio of the amplitude can be higher than that of the single component nonlinear Schrödinger equation.

†Partial financial support has been provided by the Research Grants Council through contracts HKU7117/13E and HKU17200815.
4:05PM L32.00001 Rotating parallel ray omni-directional integration for instantaneous pressure reconstruction from measured pressure gradient\textsuperscript{1}, XIAOFENG LIU, SETH SIDDLE-MITCHELL, San Diego State University — This paper presents a novel pressure reconstruction method featuring rotating parallel ray omni-directional integration, as an improvement over the circular virtual boundary integration method introduced by Liu and Katz (2003, 2006, 2008 and 2013) for non-intrusive instantaneous pressure measurement in incompressible flow field. Unlike the virtual boundary omni-directional integration, where the integration path is originated from a virtual circular boundary at a finite distance from the real boundary of the integration domain, the new method utilizes parallel rays, which can be viewed as being originated from a distance of infinity, as guidance for integration paths. By rotating the parallel rays, omni-directional paths with equal weights coming from all directions toward the point of interest at any location within the computation domain will be generated. In this way, the location dependence of the integration weight inherent in the old algorithm will be eliminated. By implementing this new algorithm, the accuracy of the reconstructed pressure for a synthetic rotational flow in terms of r.m.s. error from theoretical values is reduced from 1.03% to 0.30%. Improvement is further demonstrated from the comparison of the reconstructed pressure with that from the Johns Hopkins University isotropic turbulence database (JHTDB).

\textsuperscript{1}This project is funded by the San Diego State University.

4:18PM L32.00002 Wire-cooling based synthetic experiment to evaluate multi-sensor hotwire performance\textsuperscript{1}, SPENCER ZIMMERMAN, CALEB MORRILL-WINTER, University of Melbourne, JOSEPH KLEWICKI, University of Oldenburg — The 2d-Laser Cantilever Anemometer (2d-LCA) is an innovative sensor for two-dimensional velocity measurements in fluids. It uses a microstructured cantilever made of silicon and SU-8 as a sensing element and thus capable of performing measurements with extremely high temporal resolutions up to 150kHz. The size of the cantilever defines its spatial performance. Measurements performed with NASA MSFC.

\textsuperscript{1}The support of the Australian Research Council is gratefully acknowledged.

4:31PM L32.00003 Large Field of View PIV Measurements of Air Entrainment by SLS SMAT Water Sound Suppression System\textsuperscript{1}, MATTHEW STEGMEIR, STAMATIOS POTHOS, DAN BISSELL, TSI, Inc. Shoreview MN — Water-based sound suppressions systems have been used to reduce the acoustic impact of space vehicle launches. Water flows at a high rate during launch in order to suppress Engine Generated Acoustics and other potentially damaging sources of noise. For the Space Shuttle, peak flow rates exceeded 900,000 gallons per minute. Such large water flow rates have the potential to induce substantial entrainment of the surrounding air, affecting the launch conditions and generating airflow around the launch vehicle. Validation testing is necessary to quantify this impact for future space launch systems. In this study, PIV measurements were performed to map the flow field above the SMAT sub-scale launch vehicle scaled launch stand. Air entrainment effects generated by a water-based sound suppression system were studied. Mean and fluctuating fluid velocities were mapped up to 1m above the test stand deck and compared to simulation results.

\textsuperscript{1}Measurements performed with NASA MSFC.

4:44PM L32.00004 The effect of freestream turbulence on the wake of a 2D square prism\textsuperscript{1}, DANIEL LANDER, CHRIS LETCHFORD, Department of Civil and Environmental Engineering, Rensselaer Polytechnic Institute, Troy, NY, USA, MICHAEL AMITAY, Center for Flow Physics and Control, Department of Mechanical, Nuclear and Aerospace Engineering, Rensselaer Polytechnic Institute, Troy, NY, USA, GREGORY KOPP, Boundary Layer Wind Tunnel Laboratory, University of Western Ontario, London, CA. — The effect of freestream turbulence (FST) on a 2D square prism is investigated at $Re_D = 5.0 \times 10^4$ using long duration Time Resolved Particle Image Velocimetry (TR-PIV). Increasing the FST results in alterations to the flow field in the shear-layer and base regions and the origins of the apparent differences are discussed. The triple decomposition technique is employed to disintegrate changes attributable to the coherent and random components of the global wake stresses. In the presence of FST the vortex formation process is altered due to an increase reattachment time of the separating shear-layers on the trailing edge of the prism. This is accompanied by a transposition of the von-Kármán vortices observed in the phase averaged flow field; a feature complementary to the narrowing and lengthening of the steady wake commonly observed in the literature.

\textsuperscript{1}We wish to acknowledge financial support provided by the NSF under grant CMMI-1200987

4:57PM L32.00005 The 2d-LCA as an alternative to x-wires , JAROSLAW PUCZYLOWSKI, MICHAEL HOLLING, JOACHIM PEINKE, University Oldenburg — The 2d-Laser Cantilever Anemometer (2d-LCA) is an innovative sensor for two-dimensional velocity measurements in fluids. It uses a microstructured cantilever made of silicon and SU-8 as a sensing element and is capable of performing measurements with extremely high temporal resolutions up to 150kHz. The size of the cantilever defines its spatial resolution, which is in the order of 150 µm only. Another big feature is a large angular range of 180° in total. The 2d-LCA has been developed as an alternative measurement method to x-wires with the motivation to create a sensor that can operate in areas where the use of hot-wire anemometry is difficult. These areas include measurements in liquids and in near-wall or particle-laden flows. Unlike hot-wires, the resolution power of the 2d-LCA does not decrease with increasing flow velocity, making it particularly suitable for measurements in high speed flows. Comparative measurements with the 2d-LCA and hot-wires have been carried out in order to assess the performance of the new anemometer. The data of both measurement techniques were analyzed using the same stochastic methods including a spectral analysis as well as an inspection of increment statistics and structure functions. Furthermore, key parameters, such as mean values of both velocity components, angles of attack and the characteristic length scales were determined from both data sets. The analysis reveals a great agreement between both anemometers and thus confirms the new approach.
5:10PM L32.00006 Accidental Turbulent Discharge Rate Estimation from Videos¹
, ERIC IBARRA², University of California at Berkeley, FRANKLIN SHAFFER, NETL, Department of Energy, ÖMER SAVAŞ, University of California at Berkeley — A technique to estimate the volumetric discharge rate in accidental oil releases using high speed video streams is described. The essence of the method is similar to PIV processing, however the cross correlation is carried out on the visible features of the efflux, which are usually turbulent, opaque and immiscible. The key step in the process is to perform a pixelwise time filtering on the video stream, in which the parameters are commensurate with the scales of the large eddies. The velocity field extracted from the shell of visible features is then used to construct an approximate velocity profile within the discharge. The technique has been tested on laboratory experiments using both water and oil jets at $Re \approx 10^5$. The technique is accurate to 20%, which is sufficient for initial responders to deploy adequate resources for containment. The software package requires minimal user input and is intended for deployment on an ROV in the field.

¹Supported by DOI via NETL
²Undergraduate Researcher

5:23PM L32.00007 Thermal conductivity measurements using hot-wires at small Peclet number , GILAD ARWATZ, YUYANG FAN, MARCUS HULTMARK, Princeton University — The feasibility of using hot-wires to measure gas thermal conductivity is investigated. When the local Peclet number of a hot-wire is small ($Pe<1$), molecular diffusion dominates the heat transport, and the wire becomes less sensitive to velocity. This phenomenon can be utilized to measure the thermal conductivity of the gas. To investigate the viability of the principle of operation, a lumped capacitance model is proposed, capturing the effects of both convection and conduction on heat transfer from the wire. By investigating the sensitivity of the model to velocity, temperature and conduction, it is shown that as wire dimension decreases, the sensor becomes less sensitive to both velocity and temperature and more sensitive to conduction. The model also captures the effect of varying wire dimension as well as overheat ratio.

5:36PM L32.00008 Toward the measurement of differentiable high resolution profile data in wall turbulence¹, JOHN ELNSAB, JASON MONTY, University of Melbourne, CHRISTOPHER WHITE, University of New Hampshire, MANOOCHEHR KOOCHESFAHANI, Michigan State University, JOSEPH KLEWICKI, University of Melbourne and University of New Hampshire — High resolution streamwise velocity profiles are obtained in fully developed channel flow using molecular tagging velocimetry (MTV) over a Reynolds number based upon friction velocity and channel half-height from 200 to 2000. Due to the spatial resolution afforded from the MTV technique (800 points per profile), the velocity profile is differentiable in the wall-normal direction. This, along with pressure drop measurements, allows estimates of the mean viscous force and Reynolds stress (RS) via manipulation of the mean momentum equation. Differentiation of the RS profile allows for an investigation into the gradient of the RS. This quantity is central to the dynamics, as it acts as a net source or sink of mean momentum depending upon position relative to the RS maximum. The MTV technique and methods used to obtain the profiles are discussed. The main issue with obtaining smooth profiles is dealing with the inherent spatial pattern in the array of an intensified CCD camera. This pattern is significant in the outer region, where the profile variations and the data spacing relative to the inherent scales of motion are small. A dynamic flat-field image is used to remove a majority of this pattern. The MTV results are compared and evaluated relative DNS data.

¹The authors would like to acknowledge the Australian Research Council for funding this work

5:49PM L32.00009 Optical Properties of Inductively RF Discharge for Argon (Ar) , MURAT TANISLI, NESLIHAN SAHIN, SERCAN MERTADAM, Anadolu University — Different power supplies in laboratories can produce plasma, which is the fourth state of matter. In this study, the inductively radio frequency (RF) plasma of Ar at low pressure in the quartz glass reactor prepared for special design is obtained. Discharge properties of the generated plasma are examined with optical emission spectroscopy (OES). For RF power values, Ar was sent in certain periods and amounts to reactor in which were obtained the low pressure with vacuum pump. Plasma was generated and the data obtained from OES were composed for calculating of electron temperature. Pump. Plasma was generated and the data obtained from OES were composed for calculating of electron temperature. In this way, the electron transitions can also be investigated from data. In the presence of experimental data, the collisional radiative model can be used to obtain of the electron temperature. In addition, the various graphs of the plasma parameters are showed.

6:02PM L32.00010 Transient laminar opposing mixed convection in a symmetrically heated duct with a plane symmetric sudden contraction-expansion: Buoyancy an inclination effects¹, LORENZO MARTINEZ-SUASTEGUI, ENRIQUE BARRETO, ESIME Azcapotzalco, Instituto Politécnico Nacional, CESAR TREVIÑO, UMDI, Facultad de Ciencias, Universidad Nacional Autónoma de México, Sisal — Transient laminar opposing mixed convection is studied experimentally in an open vertical rectangular channel with two discrete protruded heat sources subjected to uniform heat flux simulating electronic components. Experiments are performed for a Reynolds number of $Re = 700$, Prandtl number of $Pr = 7$, inclination angles with respect to the horizontal of $\gamma = 0\degree$, $45\degree$ and $90\degree$, and different values of buoyancy strength or modified Richardson number, $Ri^* = Gr^*/Re^2$. From the experimental measurements, the space averaged surface temperatures, overall Nusselt number of each simulated electronic chip, phase-space plots of the self-oscillatory system, characteristic times of temperature oscillations and spectral distribution of the fluctuating energy have been obtained. Results show that when a threshold in the buoyancy parameter is reached, strong three-dimensional secondary flow oscillations develop in the axial and spanwise directions.

¹This research was supported by the Consejo Nacional de Ciencia y Tecnología (CONACyT), Grant number 167474 and by the Secretaría de Investigación y Posgrado del IPN, Grant number SIP 20141309.

6:15PM L32.00011 A Discretized Method for Deriving Vortex Impulse from Volumetric Datasets , NOAM BUCKMAN, MIT, LEAH MENDELSOHN, Massachusetts Inst of Tech-MIT, ALEXANDRA TECHET, MIT — Many biological and mechanical systems transfer momentum through a fluid by creating vortical structures. To study this mechanism, we derive a method for extracting impulse and its time derivative from flow fields observed in experiments and simulations. We begin by discretizing a thin-cored vortex filament, and extend the model to account for finite vortex core thickness and asymmetric distributions of vorticity. By solely using velocity fields to extract vortex cores and calculate circulation, this method is applicable to 3D PIV datasets, even with low spatial resolution flow fields and measurement noise. To assess the performance of this analysis method, we simulate vortex rings and arbitrary vortex structures using OpenFOAM computational fluid dynamics software and analyze the wake momentum using this model in order to validate this method. We further examine a piston-vortex experiment, using 3D synthetic particle image velocimetry (SAPIV) to capture velocity fields. Strengths, limitations, and improvements to the framework are discussed.
Monday, November 23, 2015 4:05PM - 6:41PM – Session L33 Experiments: PIV Techniques Ballroom A - Jiarong Hong, University of Minnesota

4:05PM L33.00001 Improvements on Digital Inline Holographic PIV for Turbulent Flow Measurement1. JIARONG HONG, MOSTAF A TOLOUI, KEVIN MALLERY, University of Minnesota — Among all the 3D PIV techniques used in wall-bounded turbulent flow measurements, digital inline holographic (DIH) PIV provides the highest spatial resolution for near-wall flow diagnostics with low-cost, simple and compact optical set-ups. Despite these advantages, DIH-PIV suffers from major limitations including poor longitudinal resolution, human intervention (i.e. requirement for manually determined tuning parameters during tracer field reconstruction and extraction), limited tracer concentration, and expensive computations. These limitations prevent this technique from being widely implemented for high resolution 3D flow measurements. In this study, we present our work on improving holographic particle extraction algorithm with the goal of overcoming some of abovementioned limitations. Our new DIH-PIV processing method has been successfully implemented on multiple experimental cases ranging from 3D flow measurement within a micro-channel to imaging near-wall coherent structures in smooth and rough wall turbulent channel flows.

1This work is supported by the startup package of Jiarong Hong and the MnDrive Fellowship of Mostafa Tolouei from University of Minnesota

4:18PM L33.00002 Volumetric Echocardiographic Particle Image Velocimetry (V-Echo-PIV), AHMAD FALAHATPISEH, ARASH KHERADVAR, Univ of California - Irvine — Measurement of 3D flow field inside the cardiac chambers has proven to be a challenging task. Current laser-based 3D PIV methods estimate the third component of the velocity rather than directly measuring it and also cannot be used to image the opaque heart chambers. Modern echocardiography systems are equipped with 3D probes that enable imaging the entire 3D opaque field. However, this feature has not yet been employed for 3D vector characterization of blood flow. For the first time, we introduce a method that generates velocity vector field in 4D based on volumetric echocardiographic images. By assuming the conservation of brightness in 3D, blood speckles are tracked. A hierarchical 3D PIV method is used to account for large particle displacement. The discretized brightness transport equation is solved in a least square sense in interrogation windows of size 163 voxels. We successfully validate the method in analytical and experimental cases. Volumetric echo data of a left ventricle is then processed in the systolic phase. The expected velocity fields were successfully predicted by V-Echo-PIV. In this work, we showed a method to image blood flow in 3D based on volumetric images of human heart using no contrast agent.

4:31PM L33.00003 Effect of random errors in planar PIV data on pressure estimation in vortex dominated flows, JEFFREY MCCLURE, SERHIY YARUSEVYCH, University of Waterloo — The sensitivity of pressure estimation techniques from Particle Image Velocimetry (PIV) measurements to random errors in measured velocity data is investigated using the flow over a circular cylinder as a test case. Direct numerical simulations are performed for ReD = 100, 300 and 1575, spanning laminar, transitional, and turbulent wake regimes, respectively. A range of random errors typical for PIV measurements is applied to synthetic PIV data extracted from numerical results. A parametric study is then performed using a number of common pressure estimation techniques. Optimal temporal and spatial resolutions are derived based on the sensitivity of the estimated pressure fields to the simulated random error in velocity measurements, and the results are compared to an optimization model derived from error propagation theory. It is shown that the reductions in spatial and temporal scales at higher Reynolds numbers leads to notable changes in the optimal pressure evaluation parameters. The effect of smaller scale wake structures is also quantified. The errors in the estimated pressure fields are shown to depend significantly on the pressure estimation technique employed. The results are used to provide recommendations for the use of pressure and force estimation techniques from experimental PIV measurements in vortex dominated laminar and turbulent wake flows.

4:44PM L33.00004 Volumetric Real-time Wide Field Microscopy with Tunable Acoustic Lens: A New Tool for MicroPIV, TING HSUAN CHEN, CRAIG ARNOLD, Princeton Univ — Obtaining volumetric images with high frame rate is a fundamental challenge for 3D micro Particle Image Velocimetry (PIV) used in characterizing the dynamics of fluid systems. In this presentation, we propose a new method based on a tunable acoustic lens integrated in a simple optical system. By synchronizing a pulsed LED with a high-speed camera, we are able to resolve a volume of 2mm by 2mm with depth 1mm in 7us. The ability to resolve a volume of fluid in microseconds opens the door to exploring the fundamental dynamics in small-scale fluid systems.

4:57PM L33.00005 Characterization of Flow Bench Engine Testing1. ALEX VORIS, Grove City College, LAUREN RILEY, PAUL PUZNAUSKAS, The University of Alabama — This project was an attempt at characterizing particle image velocimetry (PIV) and swirl-meter test procedures. The flow direction and PIV seeding were evaluated for in-cylinder steady state flow of a spark ignition engine. For PIV seeding, both wet and dry options were tested. The dry particles tested were baby powder, glass particulate, and titanium dioxide. The wet particles tested were fogs created with olive oil, vegetable oil, DEHS, and silicon oil. The seeding was evaluated at 0.1 and 0.25 Lift/Diameter and at cylinder pressures of 10, 25 and 40 inches of H2O. PIV results were evaluated through visual and fluid momentum comparisons. Seeding particles were also evaluated based on particle size and cost. It was found that baby powder and glass particulate were the most effective seeding options for the current setup. The oil fogs and titanium dioxide were found to deposit very quickly on the mock cylinder and obscure the motion of the particles. Based on initial calculations and flow measurements, the flow direction should have a negligible impact on PIV and swirl-meter results. The characterizations found in this project will be used in future engine research examining the effects of intake port geometry on in-cylinder fluid motion and exhaust gas recirculation tolerances.

1Thanks to NSF site grant #1358991.

5:10PM L33.00006 Instantaneous, phase-averaged, and time-averaged pressure from particle image velocimetry1. ROELAND DE KAT, University of Southampton — Recent work on pressure determination using velocity data from particle image velocimetry (PIV) resulted in approaches that allow for instantaneous and volumetric pressure determination. However, applying these approaches is not always feasible (e.g. due to resolution, access, or other constraints) or desired. In those cases pressure determination approaches using phase-averaged or time-averaged velocity provide an alternative. To assess the performance of these different pressure determination approaches against one another, they are applied to a single data set and their results are compared with each other and with surface pressure measurements. For this assessment, the data set of a flow around a square cylinder (de Kat & van Oudheusden, 2012, Exp. Fluids 52:1089–1106) is used.

1RdK is supported by a Leverhulme Trust Early Career Fellowship
5:36PM L33.00008 PIV-based estimation of unsteady loads on a flat plate at high angle of attack using momentum equation approximations. ZHAO PAN, JARED WHITEHEAD, Brigham Young University, SCOTT THOMSON, Brigham Young University Idaho, TADD TRUSCOTT, Utah State University — After more than 20 years of development, PIV has become a standard non-invasive velocity field measurement technique, and promises to make PIV-based pressure calculations possible. However, the errors inherent in PIV velocity fields propagate through integration and contaminate the calculated pressure field. We propose an analysis that shows how the uncertainties in the velocity field propagate to the propagation of error in the Poisson equation. First we note that dynamics of error propagation using boundary value problems (BVPs). Next, L_2-norm and/or L_\infty-norm are utilized as the measure of error in the velocity and pressure field. Finally, using analysis techniques including the maximum principle, the Poincare inequality pressure field can be bounded by the error level of the data by considering the well-posedness of the BVPs. Specifically, we exam if and how the error in the pressure field depend continually on the BVP data. Factors such as flow field geometry, boundary conditions, and velocity field noise levels will be discussed analytically.

5:36PM L33.00009 Combined PIV, PLIF, and laser focal displacement measurements (LFDM) to quantify gas-liquid interfacial shear stress. AMANDINE GUISSART, University of Liége, LUIS BERNAL, University of Michigan, GREGORIOS DIMITRIADIS, VINCENT TERRAPON, University of Liége — The direct measurement of loads with force balance can become challenging when the forces are small or when the body is moving. An alternative is the use of Particle Image Velocimetry (PIV) velocity fields to indirectly obtain the aerodynamic coefficients. This can be done by the use of control volume approaches which lead to the integration of velocities, and other fields deriving from them, on a contour surrounding the studied body and its supporting surface. This work exposes and discusses results obtained with two different methods: the direct use of the integral formulation of the Navier-Stokes equations and the so-called Noca’s method. The latter is a reformulation of the integral Navier-Stokes equations in order to get rid of the pressure. Results obtained using the two methods are compared and the influence of different parameters is discussed. The methods are applied to PIV data obtained from water channel testing for the flow around a 16:1 plate. Two cases are considered: a static plate at high angle of attack and a large amplitude imposed pitching motion. Two-dimensional PIV velocity fields are used to compute the aerodynamic forces. Direct measurements of dynamic loads are also carried out in order to assess the quality of the indirectly calculated coefficients.

5:49PM L33.00009 Combined PIV, PLIF, and laser focal displacement measurements (LFDM) to quantify gas-liquid interfacial shear stress. IAN MCCARTHY, DAVID HANN, BUDDHIKA HEWAKANDAMBY, BARRY AZZOPARDI, University of Nottingham — Simultaneous Particle image velocimetry (PIV) and Planar Laser Induced Fluorescence imaging (PLIF), using a pulsed Nd:YAG laser alongside a specially design optical system to produce a pair of very fine light sheets. This equipment, coupled a dual set of high speed synchronized camera, and a combination of reflective seeding particles, fluorescent dye and tracers were used to calculate the shear stress at the gas liquid interface by determining the velocity vectors in both phases. These quantities, along with the position and profile of the interface were found at a number of different inlet conditions. These conditions related to various flow patterns commonly discussed within the literature. These regimes; stratified, stratified- wavy, 2-D and 3-D waves are seen at various liquid and gas Reynolds values, with increasing complexity occurring as higher Reynolds numbers. Validation of the results was done via control of the shear stress in a number of different ways, and also compared with result of temporal film thickness taken using the LFDM. Results from these tests show good agreement with one another and those found in literature, with determination of gas-liquid shear stress found for regimes not previously investigated in this manner.

6:02PM L33.00010 DeepPIV: Particle image velocimetry measurements using deep-sea, remotely operated vehicles. KAKANI KATIJA, ALANA SHERMAN, DALE GRAVES, DENIS KLIMOV, CHAD KECY, BRUCE ROBISON, Research and Development, Monterey Bay Aquarium Research Institute — The midwater region of the ocean (below the euphotic zone and above the benthos) is one of the largest ecosystems on our planet, yet remains one of the least explored. Little-known marine organisms that inhabit midwater have developed life strategies that contribute to their evolutionary success, and may inspire engineering solutions for society relevant challenges. Although significant advances in underwater vehicle technologies have improved access to midwater, small-scale, in situ fluid mechanics measurement methods that seek to quantify the interactions that midwater organisms have with their physical environment are lacking. Here we present DeepPIV, an instrumentation package affixed to remotely operated vehicles that quantifies fluid motions from the surface of the ocean down to 4000 m depths. Utilizing ambient suspended particulate, fluid-structure interactions are evaluated on a range of marine organisms in midwater. Initial science targets include larvaceans, biological equivalents of flapping flexible foils, that create mucus houses to filter food. The structure of these mucus houses and the function they play in selectively filtering particles, and these dynamics can serve as particle-mucus models for human health. Using DeepPIV, we reveal the complex structures and flows generated within larvacean mucus houses, and elucidate how these structures function.

6:15PM L33.00011 A method to resolve low velocities in a PIV system. SUNIL BHARAD-WAJ, MEHEBOOB ALAM, Jawaharlal Nehru Centre for Advanced Scientific Research, Jakkur PO, Bangalore 560064 — A method is proposed to improve the velocity-dynamic range (VDR) of particle-image velocimetry (PIV) technique. This method uses two different timings of a pulsed laser and an outlier detection technique that helped to measure very low velocities, bypassing the limits set by the VDR of the PIV-system. The lower limit of the resolvable velocity is not set by the algorithm but by the laser-timings. The reliability of the method is verified by carrying our planar measurements of the mean and fluctuation velocities in an axisymmetric jet at a Reynolds number of about 3500. The radial velocity, which is usually an order-of-magnitude lower than the axial velocity, is successfully resolved in the ambient region of the jet as compared to results obtained by employing the post-processing techniques of the standard PIV-system. Overall, the proposed method seems to increase the velocity-dynamic range of PIV-algorithm to capture low-velocities in an otherwise fast flow.

6:28PM L33.00012 Image-processing method for near-wall PIV measurement around a moving interface. LICHAO JIA, YIDING ZHU, HUJING YUAN, CUNBIAO LEE, Peking University — This paper presents a PIV (particle image velocimetry) image processing method for the near-wall measurement when the interface is moving. Based on the successful interface tracking and precise determination of the velocity of the interface, the optimal synthetic particles with the kinetic information of the interface are added into the original particle image. The performance of the velocity estimation near the wall is then improved by the effective restriction of the particles from both sides of the interface. Quantitative evaluations of this method have been performed by applying it to Monte Carlo simulations and experimental tests. The improved method could help to provide more reliable results for the measurement of the flows around a rotating blade.
4:05PM L35.00001 Binary Raindrop Collisions

FIRAT TESTIK, KALIMUR RAHMAN, University of Texas at San Antonio — In this talk, we will present first-time observations of binary raindrop collisions in natural rainfall and discuss the observed raindrop collision outcomes (i.e. breakup, coalescence, and bounce). Binary raindrop collisions have long been hypothesized as a key process in shaping the raindrop size distribution, an important quantity for a number of meteorological and hydrological applications. Testik (2009) developed a regime diagram to determine the outcomes of a raindrop collision based upon the collision kinetic energies and surface energies of the colliding raindrops. This regime diagram has been validated previously using two different laboratory datasets for the collision of simulated raindrops. A new instrument that we have developed for precipitation microphysical observations, called High-speed Optical Disdrometer (HOD), made raindrop collision observations possible in natural rainfall and provided a valuable small dataset. In the light of these first time field observations of raindrop collisions, we will discuss Testik's diagram in this talk. Testik, F. Y., 2009. Outcome regimes of binary raindrop collisions. Atmos. Res., 94, 389–399.

1This material is based upon work supported by the National Science Foundation under Grant No. AGS-1144846.

4:18PM L35.00002 ABSTRACT WITHDRAWN —

4:31PM L35.00003 Scaling vs simulations in the head-on collision of viscous drops with insoluble surfactants

CAROLINA VANNOZZI, University of California Santa Barbara — Scaling arguments are presented to show the effect of the surface diffusivity Ds on the head-on collision of two equal-sized viscous drops in a viscous matrix with insoluble surfactants. The scaling arguments are compared to simulations [1] of the experimental system studied by Yoon et al [2] where the drops are Polybutadiene(PBD) in PDMS, stabilized by block copolymers surfactants. Overall, the scaling could predict the effect of the different parameters on the drainage time (the surface Peclet and the pushing force due to the external flow), but could not predict the experimental or simulated values. We tested our simulations against the scaling argument of [3], that claimed that emulsions stabilized by small molecule surfactants can be described with the assumption of non-diffusing surfactants. Here, however, following the same arguments, but without using the Stokes-Einstein expression for the surfactant surface mobility employed in Ref. [3] and by simply substituting the parameters for different emulsion systems, we show that Ds can be neglected only for oil in water emulsions, not for water in oil emulsion [1].


4:44PM L35.00004 Oscillations of a liquid bridge resulting from the coalescence of two droplets

VERONIQUE CHIREUX, DAVID FABRE, FREDERIC RISSO, PHILIPPE TORDJEMAN, SEBASTIEN CAZIN, Institut de Mecanique des Fluides de Toulouse, Universite de Toulouse, CNRS, IMFT-GROUPE INTERFACES TEAM — We study the inertial oscillations of a bridge of liquid maintained between two disks, both experimentally and theoretically. In the experiment, the bridge is formed by the coalescence of two droplets. After coalescence, the bridge performs weakly damped oscillations until it reaches its equilibrium shape. Four modes of oscillations can be extracted from digital processing of images recorded by means of a high-speed camera. Their frequency and damping rate have been determined and found to be independent of the initial conditions. Concurrently, the eigen modes of oscillations of a non-cylindrical bridge have been computed by assuming inviscid flow and small amplitude oscillations, and their characteristics turn out to be significantly different from that of a cylindrical bridge. The agreement between theoretical and measured frequencies confirms that the experimental modes correspond to the eigenmodes of the linear inviscid theory.

4:57PM L35.00005 Dynamics of drop coalescence on under-liquid substrates

SURJYASISH MITRA, SUSHANTA MITRA, York University — Theoretical understanding of drop coalescence on under-liquid substrates is a challenging problem due to the presence of a surrounding viscous medium. Though, most work till date have focused on coalescence in air medium, the presence of a surrounding viscous medium is a significant extension to this classical coalescence problem. Such instances are often found in physical systems such as oil-spills, wetting of marine ecosystem, etc. In the present work, a modified one-dimensional lubrication equation has been developed to describe the early coalescence behavior of two symmetric sessile drops for under-liquid substrates, which takes into account the viscosities of both the drop and the surrounding medium. We found a new time scale which governs the process and there exist a cross-over time between the universal scaling of the bridge height growth $h \sim t$ (valid for both under-liquid and air) and a much slower bridge growth $h \sim t^{1/2}$ occurring at a later time. It is also found that the evolving bridge profile has a self-similarity, which breaks up much earlier for under-liquid substrates as opposed to symmetric coalescence in air.

5:10PM L35.00006 Partial coalescence of soap bubbles

DANIEL M. HARRIS, UNC Chapel Hill, GIUSEPPE PUCCI, University of Calabria, JOHN W. M. BUSH, MIT — We present the results of an experimental investigation of the merger of a soap bubble with a planar soap film. When gently deposited onto a horizontal film, a bubble may interact with the underlying film in such a way as to decrease in size, leaving behind a smaller daughter bubble with approximately half the radius of its progenitor. The process repeats up to three times, with each partial coalescence event occurring over a time scale comparable to the inertial-capillary time. Our results are compared to the recent numerical simulations of Martin and Blanchette [Phys. Fluids 27, 012103 (2015)] and to the coalescence cascade of droplets on a fluid bath.

5:23PM L35.00007 Coalescence of surfactant-laden drops in liquids

EMILIA NOWAK, MARK SIMMONS, University of Birmingham — Whilst coalescence of droplets in air is much studied, the mechanism of merging surfactant-laden droplets in other liquids is less well understood. The dynamics of the coalescence of droplets in presence of surfactants was investigated focusing on the curvature and progression of the width of the neck that bridges the drops (up to millimetres) as well as the mixing patterns and surface flows driven by Marangoni stresses. Coalescence of different composition droplets revealed difference in the curvature of the meniscus on either side of the growing bridge which was more pronounced for the lower viscosities of the surrounding oils and related to the different local values of the surface tension. With the aid of a dye present in one of the drops, the visualisation of bulk flow was possible and different patterns were observed with increasing viscosity of the surrounding oil that led to formation of mushroom-like structures inside the droplets.

1EPSRC Programme Grant, MEMPHIS, EP/K0039761/1
In-flight surface tension and viscosity measurements of inkjet printed droplets. HENDRIK STAAT, University of Twente, ARJAN VAN DER BOS, MARC VAN DEN BERG, HANS REINTEN, HERMAN WUSHOFF, Océ Technologies B.V., MICHEL VERSLUIJS, DETLEF LOHSE, University of Twente. In modern drop-on-demand inkjet printing, the jetted liquid is a mixture of solvents, pigments and surfactants. In order to predict the droplet formation process, it is of importance to know the liquid properties. Surface tension is not constant at the timescale of droplet formation for a liquid that contains surfactants, making it non-trivial to determine the surface tension of the ink directly. Therefore we developed a technique to measure the surface tension of liquids during inkjet printing. We use high speed imaging to record the shape oscillation of a microdroplet within the first few hundred microseconds after droplet pinch-off. The frequency of oscillation depends on the surface tension, so by determining this frequency, we can measure the surface tension. The decay of oscillation amplitude is set by the viscosity, so we can also determine the viscosity with this technique. We use this technique to study the effect of surfactants on the surface tension of ink during the inkjet printing process.

Assessment of Droplet Collision Models in Pulsed Sprays. GILES BRERETON, FARID ROSHANGHALB, Michigan State University. The electronic control of fuel injectors allows multiple ensembles of the same pulsed spray event to be measured at different locations in the spray, and to be ensemble averaged for data analysis. In this talk, we present experimental laser-diffraction measurements of droplet size distributions in planes through the pulsed spray, at different locations downstream of the regions of primary and secondary breakup. The measured size distribution closest to the injector serves as an initial condition for droplet collision simulations whereas the measured downstream distributions serve as target data. Lagrangian simulations of a population of spherical fuel droplets matching the measured near-injector size distribution and velocity are then carried out using different collision and satellite-droplet-generation models and compared with downstream size-distribution measurements. Collision models which account rationally for the relative vector velocities of colliding droplet pairs yield downstream size distributions in good agreement with measurements whereas ‘ad hoc’ models and those which assume random collision angles fair poorly.

Stability and motion of liquid bridges between non-parallel surfaces. MOHAMMADMEHDI ATAEI, York University, HUANCHEN CHEN, TIAN TANG, University of Alberta, ALIDAD AMIR-FAZLI, York University. Squeezing and stretching liquid bridges formed by approaching upper surface to a sessile drop deposited on a lower surface, is frequently observed in nature and industry, e.g. printing. However, most literature focuses on liquid bridges between two parallel surfaces. In practice, bridges can also be formed between surfaces with an angle between them. Here, the effect of α on the stability and motion of the bridge was studied experimentally. Different pairs of surfaces from hydrophilic to hydrophobic, along with different contact angle hysteresis (CAH) values, were used to study the effect of surface contact angle (SCA) and CAH on the bridge stability and motion. Unlike bridges between parallel surfaces, a stable bridge may not be formed when α is larger than a threshold value αc. Instead, when bridge forms, it can undergo unstable movement towards the ends of surfaces. Shown in this study, αc is governed by both SCA and CAH (typically missed in literature). Also, during the squeezing and stretching cycles, because of α, bulk motion of the liquid bridge along the surfaces can be observed. The direction and magnitude of the bulk motion is found to be related to SCA, CAH and α.

Surfactant effect on drop coalescence and film drainage hydrodynamics. WEHELIYE WEHELIYE, MAXIME CHINAUD, VICTOR VOULGAROPOULOS, PANAGIOTA ANGELI, Department of Chemical Engineering, University College London. Coalescence of a drop on an aqueous-organic interface is studied in two test geometries: a rectangular acrylic vessel and a Hele-Shaw cell (two parallel plates placed 2mm apart) are investigated for the experiments. Time resolved Particle Image Velocimetry (PIV) measurements provide information on the hydrodynamics during the bouncing stage of the droplet and on the vortices generated at the bulk fluid after the droplet has coalesced. The velocity field inside the droplet during its coalescence is presented. By localizing the rupture point of the coalescence in the quasi two dimensional cell, the film drainage dynamics are discussed by acquiring its flow velocity by PIV measurements with a straddling camera. The effect of surface tension forces in the coalescence of the droplet is investigated by introducing surface active agents at various concentrations extending on both sides of the critical micelle concentration.

Small drops from large nozzles. ALFONSO ARTURO CASTREJON-PITA, University of Oxford. We report experimental and numerical results of the generation of drops which are significantly smaller than the nozzle from which they are generated. The system consists of a cylindrical reservoir and two endplates. One plate is a thin metal sheet with a small orifice in its centre which acts as the nozzle. The other end consists of a piston which moves by the action of an electromechanical actuator which in turn is driven by sine-shape pull-mode pulses. The meniscus (formed at the nozzle) is thus first overturned, forming a cavity. This cavity collapses and a thin and fast jet emerges from its centre. Under appropriate conditions the tip of this jet breaks up and produces a single diminutive drop. A good agreement between the experimental and numerical results was found. Also, a series of experiments were performed in order to study the effects that the pulse amplitude and width, together with variations in the liquid properties, have over the final size of the droplet. Based on these experiments, a predictive law for the droplet size has been derived.

This work was funded by the Royal Society (University Research Fellowship and Research Grant), the John Fell Fund (Oxford University Press), the Ministry of Science and Education (DPI2013-46485 Spain), and the Junta de Andalucia (P08-TEP-31704128 Spain).

Monday, November 23, 2015 4:05PM - 6:41PM —
4:05PM L36.00001 A Numerical Investigation of a Gaseous Jet Interacting with a Supercavity. MICHAEL KINZEL, MICHAEL MOENY, MICHAEL KRANE, The Pennsylvania State University - Applied Research Laboratory, IVAN KIRSCHNER, Applied Physical Sciences — In this work, the interaction between a ventilated supercavity and a jet are examined using computational fluid dynamics (CFD). In this context, supercavities are large gaseous cavities that surround a vehicle for drag reduction. Its interaction with a gaseous jet is not well understood, and CFD is used to help understand the physical interactions. A validated CFD model is used, indicating that the CFD qualitatively captures a wide range of interaction regimes. More importantly, for the context of developing physical insight, the CFD seems to capture the correct qualitative trend in the bulk cavity behavior. Using these validated models, a number of novel insights into the physical characteristics of the interaction are developed. These interactions are described by: (1) the jet gas and ventilation gas poorly mix within the cavity, (2) the jet appears to cause additional gas leakage by transitioning the cavity from a recirculating flow to an axial flow, (3) the jet has the ability to lengthen the cavity, and (4) the jet invokes wake instabilities that drive cavity pulsation. These phenomena are to be presented and discussed within the presentation.

4:18PM L36.00002 A cavitation bubble bursts near a particle. STEPHANE POULAIN, Universite de Toulouse, ISAE-Supaero, Departement Aerodynamique, Energetique et Propulsion, Toulouse, France, GABRIEL GUENOUN, Department of Physics, ENS Cachan, Cachan, France, SEAN GART, WILLIAM CROWE, SUNGHWAN JUNG, Department of Biomedical Engineering and Mechanics, Virginia Tech, Blacksburg, Virginia, USA — Cavitation bubbles induce impulsive forces on surrounding substrates, particles, or surfaces. Even though cavitation is a traditional topic in fluid mechanics, current understanding and studies do not capture the effect of cavitation on suspended objects in fluids. In the present work, the dynamics of a spherical particle due to a cavitation bubble is experimentally characterized and compared with an analytical model. Three phases are observed: the growth of the bubble where the particle is pushed away, its collapse where the particle approaches the bubble, and a longer time scale postcollapse where the particle continues to move toward the collapsed bubble. The particle motion in the longer time scale presumably results from the asymmetric cavitation evolution at an earlier time. Our theory considering the asymmetric bubble dynamics shows that the particle velocity strongly depends on the distance from the bubble as an inverse-fourth power law, which is in good agreement with our experimentation. This study sheds light on how small free particles respond to cavitation bubbles in fluids.

4:31PM L36.00003 The Dynamics of Partial Cavities and the Effect of Non-Condensable Gas. SIMO A. MAKIHARJU, HARISH GANESH, STEVEN L. CECCIO, University of Michigan. — Partial cavitation is encountered in a variety of common applications, from fuel injectors to lifting surfaces, and in general it has detrimental effects on the system wear and performance. Partial cavitation is also responsible for cavitation erosion and cavitation noise. In the present work, experiments are conducted focusing on the dynamics of shedding cavities forming in a canonical geometry (downstream of a wedge apex). The inlet cavitation number was fixed at 2.0 and the Reynolds number based on the hydraulic diameter was 6x10^6. The effects of dissolved gas content and of non-condensable gas injection into the cavity were carefully studied utilizing dynamic pressure transducers and x-ray radiography. Gas was injected either immediately downstream of the wedge’s apex or further downstream in mid-cavity. The gas injected over the wedge apex was found to end up in the separated shear layer, and did not affect the development of cavitation. The results suggest that non-condensable gas injection can cause the shedding mechanism to switch from one dominated by condensation shock to one dominated by re-entrant liquid jet.

4:44PM L36.00004 Cavitation structures formed during the collision of a sphere with an ultra-viscous wetted surface. MOHAMMAD MANSOOR, King Abdullah University of Science and Technology, JEREMY MARSTON, Texas Tech University, JAMAL UDDIN, University of Birmingham, SIGURDUR THORODDSEN, King Abdullah University of Science and Technology. — We investigate the characteristics of cavitation and its precursors when a sphere collides with a solid surface covered with a layer of non-Newtonian liquid having kinematic viscosities of up to ν₁ = 20,000,000 cSt. Liquids with high visco-elastic properties are shown to enable sphere rebound without any prior contact with the solid wall. Cavitation by depressurization (i.e. during rebound) in such non-contact cases is observed to occur after a noticeable delay from when the minimum gap distance is reached and originate from remnant bubbles (remains of the obliterated primary bubble entrapped initially by the lubrication pressure of air during film entry). Contact-cases produced a cylindrical structure attached to the wall having undulations along the cavity interface which were further investigated using high-speed particle image velocimetry (PIV) techniques. We show the existence of shear-stress-induced cavitation during sphere approach towards the base wall (i.e. the pressurization stage) in ultra-viscous films. A theoretical model based on the lubrication assumption is solved for the squeeze flow in the regime identified for shear-induced cavity events to investigate the criterion for cavity inception in further detail.

4:57PM L36.00005 Bubble coalescence at any Reynolds number. JAMES MUNRO, University of Cambridge, CHRISTOPHER ANTHONY, OSMAN BASARAN, Purdue University, JOHN LISTER, University of Cambridge. — When two bubbles touch, a hole is formed in the fluid sheet between them, and surface tension drives a radial flow which quickly pulls the hole wider. The singular shape and velocity of the initial configuration make experimental imaging or numerical simulation of the very early stages of coalescence challenging. Here we present detailed similarity solutions for the thickness of the fluid sheet and the velocity profile, and show that the radius of the hole increases as $r_{\text{h}} \propto t^{1/2}$ for any Reynolds (Ohnesorge) number. Remarkably, the initial quadratic profile of the sheet allows for an exact solution in which inertia and viscosity have the same scalings with time and remain in fixed proportion. Solution of a third-order set of ordinary differential equations determines the prefactors and profiles. In addition, asymptotic analysis of the compressional boundary layer structure in the inviscid limit formally justifies and brings new insight to earlier ad hoc ‘blob’ models. Comparison can be made between our similarity solutions, full Navier–Stokes simulations and experimental data from Paulsen et al., Nat. Commun., vol. 5, 2014.

5:10PM L36.00006 Coalescence of Bubbles in a Newtonian Fluid. CHRISTOPHER ANTHONY, SUMEET THETE, Purdue University, JAMES MUNRO, JOHN LISTER, University of Cambridge, MICHAEL HARRIS, OSMAN BASARAN, Purdue University. — Bubble coalescence plays a central role in industry and nature. While considerable work has been done in the past decades to analyze the coalescence of drops in a passive medium, it is only quite recently that the problem of bubble coalescence has begun to receive comparable interest. During bubble coalescence, two bubbles touch and create a gas bridge that grows from microscopic to macroscopic scales. We use high-accuracy simulation to analyze the dynamics in the vicinity of the space-time singularity created by the merging of two bubbles immersed in an outer Newtonian fluid of non-negligible density and viscosity while treating the inner gas as dynamically passive. This problem has recently been studied experimentally by Nagel and coworkers (2014) and theoretically by Munro and coworkers (2015) by asymptotic analysis. While both studies agree on power law scaling of the variation of the minimum neck radius with time, there is a discrepancy in the proposed/observed prefactors. In order to reconcile these differences, simulations are used to access earlier times than it has been possible in experiments. Extremely small length scales are also attained in the simulations through the use of a truncated domain approach.
5:23PM L36.00007 Bubble coalescence in a power-law fluid, PRITISH KAMAT, SUMEET THETE, OSMAN BASARAN, Purdue University — As two spherical gas bubbles in a liquid are slowly brought together, the liquid film or sheet between them drains and ultimately ruptures, forming a circular hole that connects them. The high curvature near the edge of the liquid sheet drives flow radially outward, causing the film to retract and the radius of the hole to increase with time. Recent experimental and theoretical work in this area has uncovered self-similarity and universal scaling regimes when two bubbles coalesce in a Newtonian fluid. Motivated by applications such as polymer processes, food and drug manufacture, and aeration/deaeration systems where the liquids often exhibit deformation-rate thinning rheology, we extend the recent Newtonian studies to bubble coalescence in power-law fluids. In our work, we use a combination of thin-film theory and full 3D, axisymmetric computations to probe the dynamics in the aftermath of the singularity.

5:36PM L36.00008 Numerical simulations of the translation of collapsing bubbles, ELENA IGUALADA-VILLODRE, Universidad Carlos III de Madrid, DANIEL FUSTER, CNRS-UPMC UMR 7190 Institut d’Alembert, France, JAVIER RODRIGUEZ-RODRIGUEZ, Universidad Carlos III de Madrid — In this work we present a numerical method developed to solve the collapse of single non-spherical bubbles in an incompressible liquid. The Gerris software is used to solve for the 3D conservation equations in both phases in a system where the total volume changes in the gas are imposed. The numerical results are used to discriminate various bubble collapse regimes as a function of the collapse intensity and the strength of a non-symmetrical force (e.g. gravity). At low Weber numbers and non-zero Froude numbers, the bubble remains approximately spherical. In this regime the solution numerically obtained is shown to converge in the inviscid case to the theoretical solution. For large Weber numbers, a fast jet breaks the bubble dissipating an important part of energy during the collapse. Interestingly, if it is possible to identify regimes for moderate Weber numbers where the initiation of jet formation influences its translational motion without breaking the bubble. In accordance with numerical results, experiments with bubbles generated by water electrolysis subjected to shock waves show that bubbles suffer non-spherical interface deformations. The results of this study may help to further develop medical applications using bubbles as drug-carriers.

5:49PM L36.00009 Planing forces and cavity shapes on cylindrical afterbodies, AREN HELLUM, JESSE BELDEN, DAVID BEAL, STEPHEN HUYER, CHARLES HENOCHE, DANA HRUBES, NUWC-Newport — Supercavitation is a drag reduction technique by which an underwater body is enclosed over a significant portion of its length in a bubble of gas. Hydrodynamic forces act on the body only through contact with the nose and a planing section at the rear. Models of the planing forces typically assume that the body is placed into a cavity which is unchanged by the presence of the body, and the present study was designed to test the validity of this assumption. Measurements were taken of the planing forces for five afterbody lengths over a range of angles concurrently with photographs showing the size and shape of the cavity produced. These observations reveal that the cavity form and growth rate are significantly affected by both the length and angle of attack of the body; the length of the cavity shrinks at the same angle of attack as the body length is reduced past a critical threshold, suggesting a hydrodynamic interaction between the afterbody trailing edge and the cavity. Additionally, the planing forces demonstrate a non-monotonic dependence on attack angle that is not readily explained by existing models, specifically a lift crisis for short bodies in which the planing lift goes to zero over a range from -1 to -3 degrees.

6:02PM L36.00010 Experimental study on the onset of cavitation induced by an impact1, AKIHITO KIYAMA, CHIHIRO KURIHARA, YOSHIYUKI TAGAWA, Tokyo Univ of Agr & Tech — We study reasonable expression for predicting the onset of cavitation induced by an impact experimentally. A liquid-filled test tube is dropped and impacts a floor, followed by the emergence of cavitation bubbles inside a liquid. As floor materials, a metal and a resin are chosen. As a wetting liquid, gas-saturated silicone oil was used. Experiments are conducted at room temperature. The condition for cavitation occurrence for a resin floor cannot be described by the typical velocity measured by high-speed imaging. Temporal resolution and spatial resolution of which are respectively O(10) μs and O(100) μm. We investigate sudden acceleration at the impact using an accelerometer. Its temporal resolution is O(1) ns, much smaller than that of high-speed imaging. The time history of acceleration for the resin floor is more moderate and peak acceleration is smaller than that for the metal floor. Based on these findings, we discuss the reasonable description of the criterion for the onset of cavitation bubbles, applicable for various floors.

1JSPS KAKENHI Grant Number 26709007

6:15PM L36.00011 High temperatures produced by bubble collapse near a rigid wall, SHAHABODIN ALAHYARI BEIG, BAHMAN ABOULHASANZADEH, ERIC JOHNSEN, Mechanical Engineering Department, University of Michigan — The collapse of a cavitation bubble is known to have damaging effects on its surroundings. Although numerous investigations have been conducted to predict the pressures produced by this process, fewer have been devoted to determine the heating produced by the bubble collapse. Such heating of the surrounding medium may be important for materials whose mechanical properties depend on temperature (e.g., polymeric coatings). A newly developed computational method to solve the compressible Navier-Stokes equations for gas/liquid flows is used. The high curvature near the edge of the liquid sheet drives flow radially outward, causing the film to retract and the radius of the hole to increase with time. Recent experimental and theoretical work in this area has uncovered self-similarity and universal scaling regimes when two bubbles coalesce in a Newtonian fluid. Motivated by applications such as polymer processes, food and drug manufacture, and aeration/deaeration systems where the liquids often exhibit deformation-rate thinning rheology, we extend the recent Newtonian studies to bubble coalescence in power-law fluids. In our work, we use a combination of thin-film theory and full 3D, axisymmetric computations to probe the dynamics in the aftermath of the singularity.

6:28PM L36.00012 Investigation of cavitating flows by X-ray and optical imaging, OLIVIER COUTIER-DELGOSHA1, SYLVIE FUZIER2, ILYASS KHLIFA3, Arts et Metiers ParisTech / LML laboratory, KAMEL FEZZAA, APS - Argonne National Laboratory — Hydrodynamic cavitation is the partial vaporization of high speed liquid flows. The turbulent, compressible and unsteady character of these flows makes their study unusually complex and challenging. Instabilities generated by the occurrence of cavitation have been investigated in the last years in the LML laboratory by various non-intrusive measurements including X-ray imaging (to obtain the fields of void fraction and velocity in both phases), and PIV with fluorescent particles (to obtain the velocity fields in both phases). It has been shown that cavitation is characterized by significant slip velocities between liquid and vapor, especially in the re-entrant jet area and the cavity wake. This results suggests some possible improvements in the numerical models currently used for CFD of cavitating flows.

1Professor at Arts et Metiers ParisTech, Director of the LML laboratory
2Professor assistant at Arts et Metiers ParisTech
3PhD, research Engineer

Monday, November 23, 2015 4:05PM - 6:41PM
Session L37 Minisymposium: Cavitation in Soft Tissue Sheraton Back Bay A - E. Johnsen, University of Michigan
4:05PM L37.00001 Bioeffects due to acoustic droplet vaporization1, JOSEPH BULL, University of Michigan — Encapsulated micro- and nano-droplets can be vaporized via ultrasound, a process termed acoustic droplet vaporization. Our interest is primarily motivated by a developmental gas embolism therapy technique for cancer treatment. In this methodology, infusion of tumors is induced by selectively formed vascular gas bubbles that arise from the acoustic vaporization of vascular microdroplets. Additionally, the microdroplets may be used as vehicles for localized drug delivery, with or without flow occlusion. In this talk, we examine the dynamics of acoustic droplet vaporization through experiments and theoretical/computational fluid mechanics models, and investigate the bioeffects of acoustic droplet vaporization on endothelial cells and in vivo. Early timescale vaporization events, including phase change, are directly visualized using ultra-high speed imaging, and the influence of acoustic parameters on droplet/bubble dynamics is discussed. Acoustic and fluid mechanics parameters affecting the severity of endothelial cell bioeffects are explored. These findings suggest parameter spaces for which bioeffects may be reduced or enhanced, depending on the objective of the therapy. This work was supported by NIH grant R01EB006476.

1Supported by JSPS KAKENHI Grant Number 15K13865.

4:31PM L37.00002 Design and Control of Functional Microbubbles for Medical Applications of Ultrasound1, SHU TAKAGI, TAIKI OSAKI, TAKUYA ARIOYOSHI, TAKASHI AZUMA, The University of Tokyo, MITSUHISA ICHIYANAGI, Sophia University, IKUYA KINFUJI, The University of Tokyo — Microbubbles are used as a contrast agent for ultrasound diagnosis. It is also expected to be use for the treatment. One of the possible applications is microbubble DDS. For that purpose, microbubbles need to be well-controlled for the generating process and manipulation. In this talk, for the design and control of the functional microbubbles, an experimental study on generation and surface modification of microbubbles are explained. Using a T-junction type microchannel, small bubbles about 5µm size are successfully generated. For the surface modification, Biotin-coated microbubbles are tried to adhere the Avidin-coated wall. Furthermore, the manipulation of the microbubbles using ultrasound is also discussed. Plane-wave and focused ultrasound is used to manipulate a microbubble and bubble clusters. The experimental results are shown in the presentation.

4:57PM L37.00003 Stretching cells and delivering drugs with bubbles1, CLAUS-DIETER OHL, FENFANG LI, CHAN CHON U, YU GAO, CHENJIE XI, Nanyang Technological University — In this talk we’ll review our work on impulsive cell stretching using cavitation bubbles and magnetic microbubbles for drug delivery. For sufficient short time cells can sustain a much larger areal strain than the yield strain obtained from quasi-static stretching. Experiments with red blood cells show that even then the rupture of the cell is slow process; it is caused by diffusive swelling rather than mechanical violation of the plasma membrane. In the second part we’ll discuss bubbles coated with magnetic and drug loaded particles. These bubbles offer an interesting vector for on demand delivery of drugs using mild ultrasound and magnetic fields. We report on basic experiments in microfluidic channels revealing the release of the agent during bubble oscillations and first in vivo validation with a mouse tumor model.

5:23PM L37.00004 Bubble-cell interactions with laser-activated polymeric microcapsules, MICHELE VERSLUIS1, GUILLAUME LAJOINIE, University of Twente, TOM VAN ROOIJ, ILYA SKACHKOV, KLAZINA KOOIMAN, NICO DE JONG, Erasmus MC Rotterdam, PHYSICS OF FLUIDS GROUP, UNIVERSITY OF TWENTE TEAM, BIOMEDICAL ENGINEERING, ERASMUS MC TEAM — Polymeric microcapsules that are made light-absorbing by the addition of a dye in their shell can generate cavitation microbubbles with spatiotemporal control when irradiated by a pulsed laser. These particles less than 3 µm in size can circulate through the body, bind to tissues and are expected to be readily detected, even if a single cavitation bubble is produced. In this paper, we study the impact of such cavitation bubbles on a cell monolayer and quantify it in terms of cell poration and cell viability. Two capsules formulations were used; the first one encapsulates a low boiling point oil and induced less cell damage than the second that was loaded with a high boiling point oil. We also report the generation of stable bubbles by the first capsule formulation that completely absorb the cells in their close vicinity.

1Singapore National Research Foundations Competitive Research Program funding (NRF-CRP9-2011-04).

5:49PM L37.00005 Microcavitation as a Neuronal Damage Mechanism in Blast Traumatic Brain Injury1, CHRISTIAN FRANCK, JONATHAN ESTRADA, Brown University — Blast traumatic brain injury (bTBI) is a leading cause of injury in the armed forces. Diffuse axonal injury, the hallmark feature of blunt TBI, has been investigated in direct mechanical loading conditions. However, recent evidence suggests inertial cavitation as a possible bTBI mechanism, particularly in the case of exposure to blasts. Cavitation damage to free surfaces has been well-studied, but bubble interactions within confined 3D environments, in particular their stress and strain signatures are not well understood. The structural damage due to cavitation in living tissues particularly at the cellular level are incompletely understood, in part due to the rapid bubble formation and deformation strain rates of up to ~105-106 s⁻¹. This project aims to characterize material damage in 2D and 3D cell culture environments by utilizing a novel high-speed red-blue diffraction assisted image correlation method at speeds of up to 10⁶ frames per second.

1We gratefully acknowledge funding from the Office of Naval Research (POC: Dr. Tim Bentley)

6:15PM L37.00006 Bubble dynamics in high-amplitude ultrasound therapies1, ERIC JOHNSEN, LAUREN MANCIA, University of Michigan — Cavitation plays an important role in certain therapeutic ultrasound procedures, such as histortripsy in which megahertz pressure pulses are used to destroy tissue. The large tensions (>25 MPa) nucleate bubbles in the tissue, which rapidly grow to radii on the order of hundreds of microns and subsequently collapse. To better understand potential cavitation-induced damage, we developed a numerical framework for spherical bubble dynamics in soft tissue that includes liquid compressibility and full thermal effects, as well as a comprehensive viscoelastic model with elasticity, relaxation, viscosity and various nonlinearities. This framework has enabled us to understand the effects of the viscoelastic and thermal properties of the tissue on the bubble dynamics, and compute stress and temperature fields in the surroundings. Results indicate that different viscoelastic properties affect the bubble dynamics differently, but that overall the viscoelastic nature of tissue produces larger stresses and increased heating on the surroundings, compared to bubble dynamics in purely viscous liquids.

1This work was supported by NSF grant number CBET 1253157 and NIH grant number 1R01HL110990-01A1.
Science Policy Office.

inlet control (flow rate vs. pressure) and surfactants on the formation dynamics. This law involves a minimum of fitting parameters. We finally discuss the influence of several insights about the fluid flows in both phases. From there we infer the scaling law that relates droplet volume and frequency to the Capillary number associated to each inlet flow rate. These variations in a microfluidic cross-junction with a minimum number of geometrical parameters. We mostly focus on the dripping regime. The formation sequence is decomposed in two steps, inflation and squeezing, that vary differently according to both water and oil flow rates. These variations reveal several insights about the fluid flows in both phases. From there we infer the scaling law that relates droplet volume and frequency to the Capillary number associated to each inlet flow rate. This law involves a minimum of fitting parameters. We finally discuss the influence of inlet control (flow rate vs. pressure) and surfactants on the formation dynamics.

1This project is currently being supported by an NSF CAREER Award grant CBET-1151091.

4:05PM L38.00001 Generation of Monodisperse Liquid Droplets in a Microfluidic Chip Using a High-Speed Gaseous Microflow1, POOYAN TIRANDAZI, CARLOS HIDROVO, Northeastern Univ — Over the last few years, microfluidic systems known as Lab-on-a-Chip (LOC) and micro total analysis systems (µTAS) have been increasingly developed as essential components for numerous biochemical applications. Droplet microfluidics, however, provides a distinctive attribute for delivering and processing discrete as well as ultrasmall volumes of fluid, which make droplet-based systems more beneficial over their continuous-phase counterparts. Droplet generation in its conventional scheme usually incorporates the injection of a liquid (water) into a continuous immiscible liquid (oil) medium. In this study we demonstrate a novel scheme for controlled generation of monodisperse droplets in confined gas-liquid microflows. We experimentally investigate the manipulation of water droplets in flow-focusing configurations using a high inertial air stream. Different flow regimes are observed by varying the gas and liquid flow rates, among which, the “dripping regime” where monodisperse droplets are generated is of great importance. The controlled size and generation rate of droplets in this region provide the capability for precise and contaminant-free delivery of microliter to nanoliter volumes of fluid. Furthermore, the high speed droplets generated in this method represent the basis for a new approach based on droplet pair collisions for fast efficient micromixing which provides a significant development in modern LOC and µTAS devices.

4:18PM L38.00002 Planar Microfluidic Drop Splitting and Merging, JAMES FRIEND, UCSD, LESLIE YEO, RMIT University, MAD-LAB TEAM — Open drop microfluidic platforms offer attractive alternatives to closed microchannel devices, however, to be effective they require efficient schemes for planar drop transport and manipulation. While there are many methods that have been reported for drop transport, it is far more difficult to carry out drop operations of dispensing, merging and splitting. In this work, we introduce a novel alternative to merge and split droplets using laterally-offset modulated surface acoustic waves (SAWs). To verify the feasibility of this approach we use delivery into and out of a capillary channel. Upon removal of the modulated SAW energy, capillary forces at the center of the elongated drop drain the capillary bridge region towards both ends, resulting in its collapse and consequential splitting of the drop. This occurs only below a critical Ohnesorge number, a balance between the viscous forces that retard the drainage and the sufficiently large capillary forces that cause the liquid bridge to pinch. By this scheme we show the possibility of both reliable symmetric splitting of a drop with an average deviation in droplet volumes of only around 4%, and no greater than 10%, as well as asymmetric splitting, by tuning the input energy to the device—thus presenting a comparable alternative to electrowetting.

4:31PM L38.00003 Droplet velocity in a micrometric Hele-Shaw Cell, BENJAMIN REICHERT, AXEL HUERRE, ESPCI/CNRS, OLIVIER THEODOLY, Aix-Marseille Universite, ISABELLE CANTAT, IPR-Rennes, MARIE-CAROLINE JULIEN, ESPCI/CNRS — Droplet-based microfluidics is a growing field often requiring an accurate synchronization for automated systems. The question we address is the prediction of a viscous droplet velocity pushed by a surrounding liquid set at a fixed mean velocity. In a previous work, we showed that the level of confinement plays a crucial role by investigating the lubrication film thickness. Two regimes have been observed [Huere et al., PRL accepted 2015]: at low capillary number the film is so thin that intermolecular forces come into play setting the film thickness at a constant value whatever the capillary number. At higher capillary number a scaling law is observed following Hodges et al. model [Hodges et al. JFM 2004]. As the properties of the lubrication film impact the dissipation mechanisms, we expect that the level of confinement also plays a crucial role in setting the droplet velocity. We have performed rational experiments (investigating viscosity ratio, droplet confinement). We show that two regimes of droplet velocity as a function of capillary number are also observed and, in the capillary regime the droplets go faster than the one estimated from models. We propose a refined model taking into account a modified droplet dissipation that should be useful for the community.

4:44PM L38.00004 Interaction between microfluidic droplets in a Hele-Shaw cell, ITAI SARIG, YULI STAROSVETSKY, AMIR GAT, Technion - Israel Institute of Technology — Various fluidic systems, such as chemical and biofluidic devices, involve droplets moving within an immersing fluid in narrow micro-channels. Modeling the dynamics of such systems requires calculation of the forces of interaction between the moving droplets. These forces are commonly approximated by superposition of dipoles solutions, which requires an assumption of sufficiently large distance between the droplets. In this work we obtain exact solutions for two droplets, and a droplet within a droplet, located within a moving immersing fluid and without limitation on the distance between the droplets. This is achieved by solution of the Laplace equation for the pressure in a bi-polar coordinate system. Fourier method and transformation and calculation of the force in a Cartesian coordinate system. Our results are validated with numerical computations, experimental data and with the existing dipole-based models. We utilize the results to calculate the dynamics of a droplet within a droplet, and of two close droplets, located within an immersing fluid with oscillating speed. The obtained results may be used to study the dynamics of dense droplet lattices, common to many current micro-fluidic systems.

4:57PM L38.00005 Nonlinear Dynamics of Droplets in a Hele-Shaw Cell: Short-Lived Solitary Waves in a 1D Lattice, AMIR GAT, DANILA MEIMUKHIN, YULI STAROSVETSKY, Technion - Israel Institute of Technology—We study the nonlinear dynamics of a one-dimensional lattice consisting of shallow droplets, immersed in an immiscible liquid flowing within a Hele-Shaw cell. Such configurations are commonly used in micro-fluidic devices for chemical and biological applications. We apply regular multi-scale expansions constructed for the asymptotic limit of low energy excitations. The expansions yield Korteweg de Vries and linear Schrodinger equations governing the system dynamics, which is remarkable for configurations without inertial effects. Solutions of the governing equation are shown to include a special class of short-living solitary waves. The analytical findings are validated by the numerical computations.

5:10PM L38.00006 The mysterious droplet birth in a microfluidic cross junction1, STEPHANIE VAN LOO, TRISTAN GILET, University of Liege — In microfluidics flow focusing is widely used to produce water-in-oil droplets in microchannels at high frequency. Nevertheless, the scaling laws associated to droplet length, speed and frequency could not be identified yet, owing to the large number of parameters involved (incl. complex geometry). We here present an experimental study of droplet formation in a microfluidic cross-junction with a minimum number of geometrical parameters. We mostly focus on the dripping regime. The formation sequence is decomposed in two steps, inflation and squeezing, that vary differently according to both water and oil flow rates. These variations reveal several insights about the fluid flows in both phases. From there we infer the scaling law that relates droplet volume and frequency to the Capillary number associated to each inlet flow rate. This law involves a minimum of fitting parameters. We finally discuss the influence of inlet control (flow rate vs. pressure) and surfactants on the formation dynamics.

1Supported by the FRIA/FNRS and the Interuniversity Attraction Poles Programme (IAP7/38 MicroMAST) initiated by the Belgian Science Policy Office.
5:23PM L38.00007 Flow regimes in a T-mixer operating with a binary mixture. SIMONE CAMARRI, LORENZO SICONOLFI, CHIARA GALLETTI, MARIA VITTORIA SALVETTI, University of Pisa — Efficient mixing in small volumes is a key target in many processes. Among the most common micro-devices, passive T-shaped micro-mixers are widely used. For this reason, T-mixers have been studied in the literature and its working flow regimes have been identified. However, in most of the available theoretical studies it is assumed that only one working fluid is used, i.e. that the same fluid at the same thermodynamic conditions is entering the two inlets channels of the mixer. The use of micro-devices often involves the mixing of two different fluids or of the same fluid at different thermodynamic conditions. In this case flow regimes significantly different than those observed for a single working fluid may occur. The present work aims at investigating the flow regimes in a T-mixers when water at two different temperatures, i.e. having different viscosity and density, is entering the mixer. The effect of the temperature difference on the flow regimes in a 3D T-mixer is investigated by DNS and stability analysis and the results are compared to the case in which a single working fluid is employed.

5:36PM L38.00008 Droplet migration toward and away from wall in micro-flow. YENG-LONG CHEN, SHIH-HAO WANG, WEI-TING YEH, Academia Sinica. The hydrodynamically-induced particle migration phenomenon in microfluidics has been applied for cell isolation and particle separation. First-order analysis has been able to predict the migration velocity due to particle surface inertial stress and particle deformation, for small Reynolds Re and Capillary (Ca) numbers [1]. However, at moderate flow rates, non-linear dependences of particle migration on flow rate are found [2]. We employed lattice Boltzmann-immersed boundary method to examine the dependence of droplet migration on Re, Ca, and the droplet inner/outer viscosity ratio λ. We found that whether a droplet migrates towards or away from the wall at steady state depends strongly on λ. At high flow rates, droplets with lower inner viscosity migrate towards the center. At low flow rates, there is an optimal λ at which the droplet steady state position is closest to the channel center. This result agrees with prior experimental observations for oil in water droplets [3]. The consequences for particle separation will be discussed.

5:49PM L38.00009 Droplet Trajectory Control Using Light-Induced Thermocapillary Effects in a Microchannel. JUNE WON, SEUNGMIN KANG, SIMON DONG, Department of Mechanical Convergence Engineering, Hanyang University. Controlling droplets is one of the important functions on a microfluidic chip. Marangoni effects induced by interfacial tension gradient has been paid attention due to its strong driving force on a droplet by means of droplet control. Solutalcapillary effects occurs when the interfacial tension induced gradient is due to the transport of surfactant molecules. We aim to investigate light-induced solutalcapillary effects on a droplet trajectory. Unlike few previous studies, we illuminate a continuous phase with a laser beam, in order to minimize possible damage or property change to target molecules contained in droplets. A mixture solution of black metallic ink and oleic acid is used for the continuous phase fluid. DI-water is the disperse phase. As a result, we found that the trajectory shifting increases with increasing flow rate, non-linear dependences of particle migration on flow rate are found [2]. We employed lattice Boltzmann-immersed boundary method to examine the dependence of droplet migration on Re, Ca, and the droplet inner/outer viscosity ratio λ. We found that whether a droplet migrates towards or away from the wall at steady state depends strongly on λ. At high flow rates, droplets with lower inner viscosity migrate towards the center. At low flow rates, there is an optimal λ at which the droplet steady state position is closest to the channel center. This result agrees with prior experimental observations for oil in water droplets [3]. The consequences for particle separation will be discussed.

6:02PM L38.00010 Droplets in microchannels: dynamical properties of the lubrication film. AXEL HUERRE, MMN, UMR CNRS 7083, ESPCI ParisTech, 75005 Paris, France, OLIVIER THEODOLY, LAI, INSERM U6000, CNRS UMR 6212, Case 937, 35000 Rennes, France, ALEXANDRE LEHANGUY, Department of Chemical Engineering, Technion-IIT, Haifa, 32000, Israel, MARIE-PIERRE VALIGNAT, LAI, INSERM U6000, CNRS UMR 6212, Case 937, 35000 Rennes, France, ISABELLE CANTAT, IPR, UMR CNRS 6251, Université de Rennes 1, 35000 Rennes, France, MARIE-CAROLINE JULLIEN, MMN, UMR CNRS 7083, ESPCI ParisTech, 75005 Paris, France — The motion of droplets or bubbles in confined geometries has been extensively studied; showing an intrinsic relationship between the lubrication film thickness and the droplet velocity. When capillary forces dominate, the lubrication film thickness evolves non linearly with the capillary number due to viscous dissipation both in the droplet and between meniscus and wall. However, this film may become thin enough (tens of nanometers) that intermolecular forces come into play and affect classical scalings. Our experiments yield highly resolved topographies of the shape of the interface and allow us to bring new insights into droplet dynamics in microfluidics. We find and characterize two distinct dynamical regimes, dominated respectively by capillary and intermolecular forces. In the first regime, we also identify a new interfacial boundary condition considering only viscous stress continuity that agrees well with film thickness dynamics and interfacial velocity measurement.

6:15PM L38.00011 Counter-current thermocapillary migration of bubbles in microchannels using self-rewetting liquids. ROBSON NAZARETH, The University of Edinburgh, PEDRO SAENZ, MIT, PRASHANT VALLURI, KHELLIL SIFIANE, The University of Edinburgh. The study of bubble transport in microchannels is of great interest in evaporative cooling of microdevices technologies. This is because bubble transport under heat-transfer or phase-change causes several flow instabilities that are less understood and hinder informed design of microcooling devices. Bubble motion in microchannels under temperature gradients is highly influenced by thermocapillary forces due surface tension gradients. Most studies until now so far are mainly based on pure liquids which present a linear temperature (inverse) dependence of surface tension. In this work, we consider motion of a bubble (formed of inert gas) in the so-called self-rewetting fluid that presents a parabolic (quadratic) dependence of surface tension on temperature. At high temperature ranges that includes a surface tension minimum, we particularly investigate the counter-current thermocapillary migration of bubbles in these liquids, as experimentally depicted by Shanahan and Seifane (2014), by means of direct numerical simulations. We present a model that solves the 3D governing equations of mass, momentum, interface and energy for the two-phase system composed by incompressible, Newtonian and immiscible fluids. We resolve the deformable interface by means of a Volume-of-Fluid method. Our results indicate that there exists a pressure drop limit beyond which there would be no counter-current migration of bubbles.

6:28PM L38.00012 Numerical Simulations of Droplet Dynamics in PEM Fuel Cell Microchannels. ERIC CAUBLE, MARK OWKES, Montana State University. Proton exchange membrane (PEM) fuel cells are of beneficial interest due to their capability of producing clean energy with zero emissions. An important design challenge hindering the performance of fuel cells is controlling water removal to maintain a hydrated membrane while avoiding excess water that may lead to channel blockage. Fuel cell water management requires a detailed knowledge of multiphase flow dynamics within microchannels. Direct observation of gas-liquid flows is difficult due to the small scale and viewing obstructions of the channels within the fuel cell. Instead, this work uses a CFD approach to compute the formation and dynamics of droplets in fuel cell channels. The method leverages a conservative volume-of-fluid (VOF) formulation coupled with a novel methodology to track dynamic contact angles. We present details of the numerical approach and simulation results relevant to water management in PEM fuel cells. In particular, it is shown that variation of the contact hysteresis angle influences the wetting properties of the droplet and significantly impacts water transport throughout the a fuel cell channel.
Session L39 Microscale Flows: Particles, Orientation, Active Matter and Self-Assembly
Sheraton Back Bay C - Enkeleida Lushi, Brown University

4:05PM L39.00001 Transition to collective motion and mixing in suspensions of micro-rotors\textsuperscript{1}, PETIA VLAHOVSKA, ENKELEIDA LUSHI, Brown University — Self-organization of active objects has attracted considerable attention recently, in particular in the context of living matter. Hydrodynamic interactions play a crucial role in the emerging behavior when the objects are immersed in fluid, especially in the low Reynolds number regime. While self-propelled active objects have been extensively investigated, the collective behavior of rotating active particle has received limited attention. To elucidate the transition to collective behavior and especially the role of multi-body hydrodynamic interactions, we numerically study systems of co- and counter-rotating spheres by varying the mixture ratio as well as the total volume fraction. We show that while macroscopic patterns emerge with increasing volume fraction in all the mixtures, the organization of the 100-0 and 50-50 mixtures are different in nature. The 50-50 rotor mixtures generate macroscopic fluid flows that are larger in magnitude and more chaotic, due to the propensity of rotors of opposite spins to pair up and co-swim. The properties of these generated fluid flows are investigated, and in particular we show that the mixing of a passive dye field is more efficiently done by 50-50 rotor mixtures.

\textsuperscript{1}We acknowledge support from NSF CBET 1437545 and NSF CBET 1544196.

4:18PM L39.00002 Phase transition of active rotors due to passive particles, KYONGMIN YEO, IBM T J Watson Res Ctr, ENKELEIDA LUSHI, PETIA VLAHOVSKA, Brown University — We study the emergent collective dynamics of active rotors and microparticles. The micro-rotors consist of two active rotors doped to passive, tracer particles. The active rotors are actuated by an external magnetic field gradient, and can either align themselves with the field or counter-rotate. At low rotor densities, the active rotors form a ordered lattice, while at high rotor densities, the active rotors form a disordered system. The passive particles are immobile and do not influence the collective behavior of the active rotors. However, at high rotor densities ($\phi \geq 0.10$), the passive particles are able to organize the active rotors, and the collective behavior of the active rotors changes from ordered to disordered. The passive particles are immobile and do not influence the collective behavior of the active rotors.

4:31PM L39.00003 Electro-Orientiation of Boron-Nitride Nanotubes in Aqueous Solution, SEMIH CETINDAG, Rutgers University, SANGIL KIM, Lawrence Livermore National Laboratory, BISHNU TIWARI, SHIVA BHANDARI, DONGYAN ZHANG, YOKE KHIN YAP, Michigan Technological University, JERRY SHAN, Rutgers University — Boron-nitride nanotubes (BNNTs), which have similar structure to carbon nanotubes but very different electronic properties, are of interest for a variety of applications, including hydrogen storage, water desalination, mechanical reinforcement and improving the thermal conductivity of composites. Many potential active suspensions of micro-rotors for mixing were found because of the potential for alignment of the active rotors. However, at higher rotor densities, the active rotors form a disordered system. The passive particles are immobile and do not influence the collective behavior of the active rotors. However, at high rotor densities ($\phi \geq 0.10$), the passive particles are able to organize the active rotors, and the collective behavior of the active rotors changes from ordered to disordered. The passive particles are immobile and do not influence the collective behavior of the active rotors.

4:44PM L39.00004 Magnetic self-assembly of microparticle clusters in an aqueous two-phase microfluidic cross-flow, NIKI ABBASI, STEVEN G. JONES, BYEONG-Ul MOON, SCOTT S.H. TSAI, Ryerson University — We present a technique that self-assembles paramagnetic microparticles on the interface of aqueous two-phase system (ATPS) fluids in a microfluidic cross-flow. A co-flow of the ATPS is formed in the microfluidic cross channel as the flows of a dilute dextran (DEX) phase, along with a flow-focused particle suspension, converge with a dilute polyethylene glycol (PEG) phase. The microparticles arrive at the liquid-liquid interface and self-assemble into particle clusters due to forces on the particles from an applied external magnetic field gradient, and the interfacial tension of the ATPS. The microparticles form clusters at the interface, and once the cluster size grows to a critical value, the cluster passes through the interface. We control the size of the self-assembled clusters, as they pass through the interface, by varying the strength of the applied magnetic field gradient and the ATPS interfacial tension. We observe rich assembly dynamics, from the formation of Pickering emulsions to clusters that are completely encapsulated inside DEX phase droplets. We anticipate that this microparticle self-assembly method may have important biotechnological applications that require the controlled assembly of cells into clusters.

4:57PM L39.00005 Quantum dots deposition in a capillary tube, YONG LIN KONG, FRANCOIS BOULOGNE, HYOUNGSOO KIM, JANINE NUNES, JIE FENG, HOWARD STONE, Princeton University — The ability to assemble nanomaterials, such as quantum dots, enables the creation of functional devices that present unique optical and electronic properties. For instance, light-emitting diodes with exceptional color purity can be printed via the evaporative-driven assembly of quantum dots. Nevertheless, current studies of the colloidal deposition of quantum dots have been limited to the flat substrates of a planar geometry. Here, we investigate the evaporation-driven assembly of quantum dots inside a confined cylindrical geometry. Specifically, we observe distinct deposition or coating patterns of quantum dots at different positions along the length of a capillary tube. Such changes of coating behavior could be influenced by the evaporation speed as well as the concentration of quantum dots. Understanding the factors governing the coating process can provide a means to control the assembly of quantum dots inside a capillary tube, ultimately enabling the creation of novel photonic devices.

5:10PM L39.00006 Oscillatory flow and induced steady streaming flow around two spheres, DAVID FABRE, IMPT, University of Toulouse, JAVIERA JALAL, JUSTIN LEONTINI, RICHARD MANASSEH, Swinburne University of Technology — We investigate the flow around two fixed spheres of identical radius, subject to an oscillating flow at frequency $\omega$ and weak amplitude $u_0$. Expanding the flow in series of $u_0$, the leading order corresponds to an oscillating flow with zero mean, while the second-order correction contains a steady streaming component. Thanks to a modal decomposition in the azimuthal direction, we are able to reduce the problem to a few linear problems in 2D domain corresponding to the meridional ($r,z$) plane. Investigation of the streamlines of the steady component of the flow shows intricate patterns due to the interaction between the streaming flows induced by both spheres. The analysis also allows to compute the mean forces felt by both spheres. If the spheres are aligned obliquely with respect to the oscillating flow, the steady component of the flow shows intricate patterns due to the interaction between the streaming flows induced by both spheres. In this transverse configuration, they experience an axial force which can be either attractive or repulsive. At high frequencies the force is always attractive. At low frequencies, it is repulsive. At intermediate frequencies, the force is attractive at large distances and repulsive at small distances, leading to the existence of an equilibrium configuration.
Particle trajectory entanglement in microfluidic channels, ALVARO MARIN, MASSIMILIANO ROSSI, CHRISTIAN KÄHLER, Bundeswehr University Munich. Suspensions in motion can show very complex and counterintuitive behavior, particularly at high concentrations. In this talk we show an overlooked phenomenon occurring when a dilute particle solution is forced to travel in a narrow channel (only a few times the particle size). At critical interparticle distances, particles tend to interlace their trajectories forming a sort of hydroclusters only bonded by hydrodynamic interactions. While classical studies on non-Brownian self-diffusivity report average particle displacements of fractions of the particle diameter, the trajectories observed in our system show displacements of several particle diameters. Indeed, such a behavior resembles the deterministic trajectories found by Uspal et al. (Nat. Comm. 4, 2013) with engineered particle doublets. Trajectory statistics are obtained for different shear rates and particle sizes. The results are compared with particle dynamics simulations and analyzed under the light of recent studies on the irreversibility of non-Brownian suspensions (Metzger et al., Phys. Rev. E, 2013) to elucidate the nature of the hydrodynamic interactions entering into play. The reported phenomenon could be applied to promote advective mixing in micro-channels or particle/droplet self-assembly.

Dynamic self-assembly of microscale rotors and swimmers, MEGAN DAVIES-WYKES, New York University, JEREMIE PALACCI, University of California San Diego, TAKUJI ADACHI, LEIF RISTROPH, YANPENG LIU, XIAO ZHONG, JUN ZHANG, MICHAEL WARD, MICHAEL SHELLEY, New York University. Self-assembly is a process found throughout nature and is often dynamic, requiring fuel to occur. Artificial examples are valuable both as aids to understanding biological systems and for developing manufacturing techniques for micron-scale machines. We will describe the behaviour of micron-scale rods, constructed of three equal length segments of gold, platinum and gold (Au-Pt-Au). When placed in a solution of hydrogen peroxide fuel, these are expected to create an extensile-like flow in the surrounding fluid. These immotile rods self-assemble into structures that exhibit the two fundamental types of motion: rotation and translation, in the form of steadily rotating stacks and T-shaped swimmers. This is a rare example of an artificial system where dynamic and reversible self-assembly results in ordered structures which exhibit emergent motility.

Forming particle chains in inertial microfluidic devices, KAITLYN HOOD, University of California Los Angeles, LAWRENCE LIU, University of Southern California, MARCUS ROPER, University of California Los Angeles. Particles in microfluidic devices at finite Reynolds number self-assemble into evenly-spaced chains, which can be exploited in inertial microfluidic devices for flow cytometry, high speed imaging, and entrapment. While the location and number of chains can be manipulated by changing the channel geometry, the particle interactions are not understood well enough to manipulate the spacing between particles. We present a mathematical model of particle interactions and the formation of particle chains. We will address the following questions: Is there a preferred particle spacing? What are the conditions needed for chain formation?

Quantifying colloidal particle bands and their formation in combined electroosmotic and Poiseuille flow, ANDREW YEE, NECTMINTIN CEVHERI, MINAMI YODA, Georgia Institute of Technology. Recently, we have shown that suspended radii \( a = 245 \text{ nm} \) particles flowing through a microchannel driven by the combination of a dc electric field and pressure gradient (where the resulting electroosmotic and shear flows are in opposite directions) are attracted to the wall at low electric field magnitude \( |E| \), then assemble into concentrated bands that only exist within a few \( \mu \text{m} \) of the wall above a threshold value of \( |E| \). The \( 6 \mu \text{m} \) wide bands are aligned with the flow direction and are roughly periodic along the cross-stream direction. This talk focuses on quantitative characterization of these bands, for example how \( |E_{c1}| \) the time required for bands to form after applying the electric field \( T_c \), and the number of bands depend upon parameters such as particle volume fraction \( \phi \), shear rate \( \gamma \), and \( a \). The dynamics of the particles within the bands are visualized by imaging a mixture of particles with different fluorescent labels. The visualizations show that the particles are in a liquid state within these bands, and suggest that the particles nearest the wall move in the direction of the electroosmotic flow, while those farther from the wall move in the direction of the shear flow.

Hydrodynamic alignment and assembly of nano-fibrillated cellulose in the laminar extensional flow: Effects of solidifying agents, NITESH MITTAL, FREDRIK LUNDELL, DANIEL SODERBERG, Royal Inst of Tech. There are several fiber production technologies that are based on wet-spinning processes. Many such processes rely on the transformation of a liquid solution into a solid filament. The kinetics of solidification depends largely on the diffusion of the solvents, additives and polymer molecules, which make such systems quite complex and differ from a system to another as a function of the specific chemical, physical and structural features of the used material components. Moreover, tuning the orientation of the polymers in the liquid suspensions makes it further possible to control their structure, which in turn can lead to materials having improved properties. By keeping in mind the facts mentioned above, the aim of the current study is to utilize benefits of a flow focusing approach to align carboxymethylated cellulose nanofibrils (CNF), as a colloidal dispersion, with the help of a laminar elongational flow-field followed by the solidification using different solidifying agents or molecules (with dissimilar diffusion behavior based on their size and charges) to synthesize fibers with enhanced mechanical properties. CNF are charged elongated particles obtained from woods with diameter of 4-10 nm and length of 1-1.5 \( \mu \text{m} \), and they are completely biodegradable.

Effect of flow on Janus rods organization in polymer blends, SHAGHAYEGH KHANI, Case Western Reserve University, SAFA JAMALI, Massachusetts Institute of Technology, ARMAN BOROMAND, JOAO MAIA, Case Western Reserve University. In the past decade, Janus particles have attracted a lot of attention due to their amphiphilic nature. Directed assembly of these particles in polymer matrices can provide a tool for fabricating new functional materials. For example, the strong affinity of Janus particles to interfaces, could allow the control of the interface of phase separating polymer blends by controlling the Janus particles assembly. In this work, using mesoscale computational methods, we show that the spatial organization of Janus rods can be exploited for tuning mixtures of immiscible polymer blends. In particular, we explore the effect of different parameters that influence the rods alignment and orientation at the interface under equilibrium condition. Flow can dramatically alter the localization of these particles within the polymer blend. Therefore, we not only monitor the microstructures formed by these systems at rest, but we also do so under flow conditions and upon relaxation after flow cessation. The results of this study can be used for designing new approaches for directing nano-particles into desired morphologies, which will subsequently tune the final characteristics and properties of nano-composites.

Monday, November 23, 2015 4:05PM - 6:41PM –
Session L40 Microscale Flows: Microfluidic Devices I
Sheraton Back Bay D - Sangjin Ryu, University of Nebraska-Lincoln
4:18PM L40.00002 Flow fraction in charged rectangular microchannel to optimally design hydrodynamic filtration chip for cell sorting\textsuperscript{1}. MYUNG-SUK CHUN\textsuperscript{2}, SOHYUN JEONG, JAE HUN KIM, Korea Institute of Science and Technology (KIST), National Agenda Research Div., Seoul, TAE SEOK LEE, Harper International Corp., NY14225 — Among the passive separations, hydrodynamic filtration (HDF) can perform the fractionation of cells or particles by selective extraction of streamline controlled by the flow fraction at each branch. Only the stream near the sidewall enters the branches as the focusing, with the amount of fluid leaving the main channel being determined by the flow distribution related to the hydraulic flow resistances. Its understanding is important, but in-depth consideration has not been treated until now. The virtual boundary of the fluid layer should be first specified, and the parabolic velocity profile starts to form from the steady state flow with high Péclet numbers. We computed the 3-dimensional flow profile at the rectangular cross-section with any aspect ratios, by considering electrokinetic transport coupled with the Poisson-Boltzmann and Navier-Stokes equations. The chip was designed with the parameters rigorously determined by the complete analysis of laminar flow for flow fraction and complicated networks of main and multi-branched channels for cell sorting into the finite number of subpopulations. For potential applications to the precise sorting, our designed microfluidic chip can be validated by applying model cells consisting of heterogeneous subpopulations.

\textsuperscript{1}Supported by the KIST Institutional Program (No. 2E25382)

\textsuperscript{2}Prof. Dr.

4:31PM L40.00003 Deformability-based capsule sorting . ANNE LE GOFF, NADEGE MUNIER, PAULINE MAIRE, BMBI, Université de Technologie de Compiegne, FLORENCE EDWARDS-LEVY, ICMR, Université de Reims, ANNE-VIRGINIE SALSAC, BMBI, Université de Technologie de Compiegne — Many microfluidic devices have been developed for cancer diagnosis applications, most of which rely on costly antibodies. Since some cancer cells display abnormal mechanical properties, new sorting tools based on mechanical sensing are of particular interest. We present a simple, passive pinched flow microfluidic system for capsule sorting. The device consists of a straight microchannel containing a cylindrical obstacle. Thanks to a flow-focusing module placed at the channel entrance, capsules arrive well-centered in the vicinity of the obstacle. Pure size-sorting can be achieved at low shear rate. When increasing the shear rate, capsules are deformed in the narrow space between the pillar and the wall. The softer the capsule, the more tightly it wraps around the obstacle. After the obstacle, streamlines diverge, allowing for the separation between soft capsules, that follow central streamlines, and stiff capsules, that drift away from the obstacle with a wider angle. This proves that we have developed a flexible multipurpose sorting microsystem based on a simple design.

4:44PM L40.00004 Multiphase ferrofluid flows for micro-particle sorting , RAN ZHOU, CHENG WANG, Missouri University of Science and Technology — Utilizing negative magnetophoresis, ferrofluids have demonstrated great potential for sorting nonmagnetic micro-particles by size. Most of the existing techniques use single phase ferrofluids by pushing micro-particles to channel walls, the sorting speed is thus hindered. We demonstrate a novel sorting strategy by co-flowing a ferrofluid and a non-magnetic fluid in a droplet. Due to the magnetic field, the particles migrate across the ferrofluid stream at size-dependent velocities as they travel downstream. The laminar interface between the two fluids functions as a virtual boundary to accumulate particles, resulting in effective separation of particles. A stable and sharp interface is important to the success of this sorting technique. We investigate several factors that affect sorting efficiency, including magnetic field, susceptibility difference of the fluids, flow velocity, and channel geometry.

4:57PM L40.00005 Electrohydrodynamic manipulation of particles adsorbed on the surface of a drop\textsuperscript{1}, EDISON AMAH, KINNARI SHAH, IAN FISCHER, PUSHPENDRA SINGH, NJIT — In our previous studies we have shown that particles adsorbed on the surface of a drop can be concentrated at its poles or equator by applying a uniform electric field. This happens because even when the applied electric field is uniform the electric field on the surface of the drop is nonuniform, and so particles adsorbed on the surface are subjected to dielectrophoretic (DEP) forces. In this paper, we study the behavior of adsorbed particles at low electric field frequencies when the drop and ambient liquids are weakly conducting dielectric liquids, and model it using a leaky dielectric model. The electrohydrodynamic (EHD) flow which arises because of the accumulation of charge on the surface of the drop can be from pole-to-equator or equator-to-pole depending on the properties of the drop and ambient liquids. The flow however diminishes with increasing frequency and there is a critical frequency at which the drag force on a particle due to the EHD flow becomes equal to the DEP force, and above the critical frequency the DEP force dominates. When the fluid and particles properties are such that the EHD and DEP forces are in the opposite directions, particles can be can be collected at the poles or the equator, and also can be moved from the poles to the equator, or vice versa, by varying the frequency. Also, it is possible to separate the particles of a binary mixture when the critical frequencies of the two types of particles are different.

\textsuperscript{1}The work was supported by National Science Foundation

5:10PM L40.00006 Magnetophoretic control of water droplets in bulk ferrofluid . GEORGIOS KATSIKIS, Stanford University, ALEXANDRE BRANT, École Polytechnique, Paris-Saclay, MANU PRAKASH, Stanford University — We present a microfluidic platform for 2-D manipulation of water droplets immersed in bulk oil-based ferrofluid. Although non-magnetic, the droplets are exclusively controlled by magnetic fields, without any pressure-driven flow. The diphasic fluid layer is trapped in a submillimeter Hele-Shaw chamber that includes permanent track on its substrate. An in-plane rotating magnetic field magnetizes the peramloy tracks, thus producing local magnetic gradients, while an orthogonal magnetic field magnetizes the bulk ferrofluid. To minimize the magnetostatic energy of the system, droplets are attracted towards the locations of the tracks where ferrofluid is repelled. Using this technique, we demonstrate synchronous propagation of water droplets, analyze PIV data of the bulk ferrofluid flow and study the kinematics of propagation. In addition, we show droplet break-up, merging and derive relevant scaling laws. Finally, we discuss future applications owing to the biocompatibility of the droplets.
channel width-to-height ratio decreases. On the other hand, for the 180 degree case, the nozzle temperature determines the length and shape of the Leidenfrost point, a thin vapor layer is formed between the heated substrate and the liquid above it. The vapor pressure due to the presence of the vapor layer, together with the effect of surface tension of the liquid, exerted on the liquid-vapor interface, preventing the flow of the liquid. Furthermore, fluid mixing is substantially different for 60 degree and 180 degree between the two inlet channels and the 60 degree case, the nozzle temperature needs to increase to 160 degree Celsius in order to prevent the continuous flowing of the liquid. When nozzle temperature below 160 degree Celsius, intermittent ejection of microdroplets, whose size is a function of nozzle temperature, is observed.

We would like to acknowledge DTRA for their funding and support of our research.

5:23PM L40.00007 Contactless, high-throughput determination of electrical conductivity of one-dimensional nanomaterials by solution-based electro-orientation spectroscopy . CEVAT AKIN, JINGANG YI, LEONARD FELDMAN, JERRY SHAN, Rutgers University, CORENTIN DURAND, SABAN HUIS, AN-HUING LI, Oak Ridge National Laboratory. Oak Ridge National Laboratory, MIHAIL FILLER, Georgia Institute of Technology. The electrical-transport properties of nanowires of the same composition (and even fabricated within the same batch) often vary by orders of magnitude. Existing characterization methods are slow, making the large number of measurements needed to statistically characterize highly variable samples essentially impossible. Here, we demonstrate a contactless, solution-based method to efficiently determine the electrical conductivity of individual 1D nanomaterials. This new method, electro-orientation spectroscopy, is based on the transient alignment behavior of fluid-suspended nanowires in AC electric fields of different frequencies. Comparison with direct transport measurements by probe-based scanning tunneling electron microscopy shows that electro-orientation spectroscopy can quantitatively measure nanowire conductivity over a 6-order-of-magnitude range, 10^{-5} – 10 S/m. We demonstrate an automated microfluidic device capable of measuring and sorting hundreds of nanowires per hour. With this device, we statistically characterize the conductivity of a variety of nanowires and find significant variability in Si nanowires grown from the same wafer by metal-assisted chemical etching. Finally, we discuss the potential of the electro-orientation approach to be integrated with other solution-based methods for scalable positioning of nanowires for post-growth device assembly.

5:36PM L40.00008 A modular and lowcost 3D-printed microfluidic device with assembly of capillaries for droplet mass production . A. A. AGUIRRE-PABLO, J. M. ZHANG, E. Q. LI, S. T. THORODDSSON, K. K. Abdullah University of Science and Technology. A 3D-printed microfluidic system with assembly of capillaries for droplet generation. The system consists of the following parts: 3Dprinted Droplet Generation Units (DGUs) with embedded capillaries and two 3D-printed pyramid distributors for supplying two different fluid phases into every DGU. A single DGU consists of four independent parts: a top channel, a bottom channel, a capillary and a sealing gasket. All components are produced by 3dprinting except the capillaries, which are formed in a glass-puller. DGUs are independent of the distributor and from each other; they can easily be assembled, replaced and modified due to its modular design which is an advantage in case of a faulty part or clogging, eliminating the need to fabricate a complete new system which is cost and time demanding. We assessed the feasibility of producing droplets in this device varying different fluid parameters, such as liquid viscosity and flow rate, which affect droplet size and generation frequency. The design and fabrication of this device is simple and low-cost with the 3D printing technology. Due to the modular design of independent parts, low-cost fabrication and easy parallelization of multiple DGUs, this system provides great flexibility for industrial applications.

5:49PM L40.00009 Hydrodynamics and Mass Transfer Characteristics of Laminar Bioelectrochemical Systems, a Summary . WAY LEE CHENG, REZA SADR, Texas A&M University. Hydrodynamics and diffusion characteristics of laminar bioelectrochemical systems (BES) with common micro-channel configuration are summarized. Computational fluid dynamics (CFD) simulations are performed to supplement literature results and to provide a comprehensive summary for the flow and diffusion characteristics in these systems in terms of dimensionless parameters. The results show that decreasing the fluid velocity enhances mixing between the two parallel flow streams with a stronger mixing in the near wall region. Reducing the ratio of channel width to channel height enhances mixing. Changing the angle between the inlet channels, in general, does not have a strong effect on the flow field, except when the angle is larger than about 135 degrees. Furthermore, fluid mixing is substantially different for 60 degree and 180 degree between the two inlet channels. For the 60 degree case, the nozzle temperature determines the length and shape of the Leidenfrost point. The mixing enhancements include flow turning regions, flow splitters, and vortex shedding. The relative effectiveness of these different approaches for rapid micro-mixing is discussed. Simulations found that flow turning regions provided the best mixing profile. Experimental validation of the optimal design is verified through laser confocal microscopy experiments.

6:02PM L40.00010 Valve-less microdispenser . MING KWANG TAN, WANG XIN, WENG KENT LEE, Monash University Malaysia, Jalan Lagoon Selatan, 47500 Bandar Sunway. We demonstrate the concept of valve-less microdispenser to control of the liquid flow through the nozzle, by inducing Leidenfrost effect into the nozzle design. When the nozzle is heated above the Leidenfrost point, a thin vapor layer is formed between the heated substrate and the liquid above it. The vapor pressure due to the presence of the vapor layer, together with the effect of surface tension of the liquid, exerts on the liquid-vapor interface, preventing the flow of the liquid through the nozzle. The experimental results shown that nozzles of diameter 400 micrometer and below, the nozzle temperature of 150 degree Celsius is sufficient to prevent the continuous flowing of the liquid, as for nozzles of diameter between 400 to 500 micrometer, the nozzle temperature needs to increase to 160 degree Celsius in order to prevent the continuous flowing of the liquid. When nozzle temperature below 160 degree Celsius, intermittent ejection of microdroplets, whose size is a function of nozzle temperature, is observed.

6:15PM L40.00011 Design of an Efficient Turbulent Micro-Mixer for Protein Folding Experiments . VENKATESH INGUA, MSME Student, BLAIR PEROT, Advising Professor, Protein folding studies require the development of micro-mixers that require less sample, mix at faster rates, and still provide a high signal to noise ratio. Chaotic to marginally turbulent micro-mixers are promising candidates for this application. In this study, various turbulence and unsteadiness generation concepts are explored that avoid cavitation. The mixing enhancements include flow turning regions, flow splitters, and vortex shedding. The relative effectiveness of these different approaches for rapid micro-mixing is discussed. Simulations found that flow turning regions provided the best mixing profile. Experimental validation of the optimal design is verified through laser confocal microscopy experiments.

6:28PM L40.00012 Fabrication and characterization of vertically aligned carbon-nanotube membranes . RICHARD CASTELLANO, CEVAT AKIN, MATT PURRI, JERRY SHAN, Rutgers University, SANGIL KIM, FRANCESCO FORNASIERO, Livernmore National Labs. Membranes having vertically-aligned carbon-nanotube (VACNT) pores offer promise as highly efficient and permeable membranes for use as breathable thin films, or in filtration and separation applications, among others. However, current membrane-fabrication techniques utilizing chemical-vapor-deposition-grown VACNT arrays are costly and difficult to scale up. We have developed a solution-based, electro-field-assisted approach as a cost-effective and scalable method to produce large-area VACNT membranes. Nanotubes are dispersed in a liquid polymer, and aligned and electrodeposited with the aid of an electric field prior to crosslinking the polymer to create VACNT membranes. We experimentally examine the electrodeposition process, focusing on parameters including the electric field, composition of the solution, and CNT functionalization that can affect the nanotube number density in the resulting membrane. We characterize the CNT pore size and number density and investigate the transport properties of the membrane. Size-exclusion tests are used to check for defects and infer the pore size of the VACNT membranes. Dry-gas membrane permeability is measured with a pressurized nitrogen-flow system, while moisture-vapor-transfer rate is measured with the ASTM-E96 upright-cup test. We discuss the measured transport properties of the solution-based, electric-field-fabricated VACNT membranes in reference to their application as breathable thin films.

1 This work is support by the National Science Foundation.

1 We would like to acknowledge DTRA for their funding and support of our research.
Session L41 Russell Donnelly Minisymposium
Sheraton Constitution A - K.R. Sreenivasan, New York University

4:05PM L41.00001 Russell Donnelly at Chicago , LEO KADANOFF, Retired — The period (1956-1965) in which Russ served as a faculty member at Chicago was one in which he set the main topics that occupied his subsequent professional life. These included the few-degree-Helium work that formed the low temperature physics of that period. This work included studies of motion of vortex lines and ions in Helium. During that period, he had seven doctoral students mostly devoted in work in these areas. In addition he served as a good citizen of both his department and his field of science, by publishing a book on “Experimental Superfluidity” based on course lecture notes and also serving as an editor of a conference volume on “Non-equilibrium Thermodynamics.” Because of the University of Chicago’s nepotism rules, Russ’s wife could not follow her academic interests here. They both left for the University of Oregon.

4:18PM L41.00002 An Instability in Stratified Taylor-Couette Flow1, HARRY SWINNEY, University of Texas at Austin — In the late 1950s Russell Donnelly began conducting experiments at the University of Chicago on flow between concentric rotating cylinders, and his experiments together with complementary theory by his collaborator S. Chandrasekhar (Hydrodynamic and Hydromagnetic Stability, Clarendon Press, 1961) did much to rekindle interest in the flow instability discovered and studied by G.I. Taylor (1923). The present study concerns an instability in a concentric cylinder system containing a fluid with an axial density gradient. In 2005 Dubrulle et al. suggested that a ‘stratorotational instability’ (SRI) in this system could provide insight into instability and angular momentum transport in astrophysical accretion disks (Astron. Astrophys. 429, 429). In 2007 the stratorotational instability was observed in experiments by Le Bars and Le Gal (Phys. Rev. Lett. 99, 064502). We have conducted an experiment on the SRI in a concentric cylinder system (radius ratio \( \eta = 0.876 \)) with buoyancy frequency \( N/2\pi = 0.25, 0.50, \) or \( 0.75 \) Hz. For \( N = 0.75 \) Hz we observe the SRI onset to occur for \( \Omega_{\text{outer}}/\Omega_{\text{inner}} > \eta \), contrary to the prediction of Shalybkov and Rüdiger (Astron. Astrophys. 438, 411, 2005).

1Research conducted with Bruce Rodenborn and Ruy Ibanez

4:31PM L41.00003 Expanding Participation in Fluid Dynamics Research, RANDALL TAGG, University of Colorado Denver — Two legacies provided by great scientists are scientific discoveries and more scientists. Is there a way that these impacts can be magnified? Examples using the Taylor-Couette experiment and other fluid dynamics problems will demonstrate that indeed more people can fruitfully engage in open and even bold investigation. Participants include high school students, teachers, undergraduates, artists, business developers and interested laypersons. With imagination, good training, and a suitable lab space, a special tribute can be given to those who mentor us by scaling up the breadth of their influence.

4:44PM L41.00004 Russ Donnelly’s research at the University of Oregon, JOSEPH NIEMELA, The Abdus Salam ICTP — Coming to the University of Oregon in 1966, Russ Donnelly built up a strong research activity having two threads within hydrodynamics: the flow of ordinary fluids and that of superfluids. Vortices quantized and classical were at the heart of his research. His 1991 book Quantized Vortices in Helium II, by now a standard reference for researchers and students, elucidated some of it. To produce vortices Russ brought from Chicago two enormous rotating tables, based on 1-m diameter industrial lathe chucks obtained from General Motors. They were also used for classical systems such as Taylor-Couette flow (to generate strong Coriolis forces) and thermal convection, where the properties of rotation including early experimental investigations of the Kuppers-Lortz instability were studied. Another common thread in his research was the modulation of control parameters leading to Stokes layer effects, both thermal and viscous. In the early 90s, Russ and his group turned their attention to cryogenic turbulence in normal and superfluid systems, creating what has now become a small industry and a well-established sub-field within low temperature physics.

4:57PM L41.00005 Turbulent convection at high Rayleigh numbers1, GUENTER AHLERS, Univ of California - Santa Barbara — Russ Donnelly had a vision of building a ten-meter tall Rayleigh-Bénard convection cell for use at helium temperatures at one of the high-energy physics facilities with very large helium liquefaction capacity. It would have reached Rayleigh numbers in the \( 10^{20} \) range and had the promise of yielding detailed information about the so-called ultimate state of turbulent convection which is highly relevant to many geophysical and astrophysical problems as well as to onoceanography and climate physics. Although this was not to happen for reasons beyond his control, a laboratory-sized precursor of this venture yielded data for \( \text{Ra} \) up to \( 10^{12} \). \(^3\) The results were interpreted to yield no definitive indication of a transition to the ultimate state. This talk will review some of these data and compare them with more recent measurements using SF\(_6\) at ambient temperatures and high pressure. \(^4\) This comparison suggests that the Donnelly group actually entered a transition range to the ultimate state near \( \text{Ra}^* \approx 6 \times 10^{12} \), but re-entered the classical state at larger \( \text{Ra} \) because with increasing \( \text{Ra} \) the Prandtl number (which affects \( \text{Ra}^* \)) also increased in those experiments. In view of the above, one can estimate that, for the same parameter values, the originally envisioned ten-meter cell could have yielded a range of a couple of decades for \( \text{Ra} \) in the ultimate state.


1Supported by NSF Grant DMR11-58514.

5:10PM L41.00006 Czech cryogenic fluid dynamics inspired by Russ Donnelly1, LADISLAV SKRBEK, Faculty of Mathematics and Physics, Charles University in Prague — Following nearly five years of work along with Russ in Eugene on cryogenic turbulent convection and quantum grid turbulence, two laboratories in Prague and in Brno have been established to continue experimental research in cryogenic fluid dynamics using all three forms of cryogenic 4He - cold helium gas, normal liquid He I and superfluid He - as excellent multi-purpose working fluids. We review some of our investigations of very high Rayleigh number cryogenic thermal convection and classical and quantum turbulence in liquid helium. In particular, we discuss heat transfer efficiency of turbulent Rayleigh-Bénard convection and the role of non-Oberbeck-Boussinesq conditions on possible transition to its ultimate regime; our second sound attenuation experiments probing both steady state and decaying coflow, counterflow and pure superflow of He II through channels of square cross-section including the concept of effective kinematic viscosity. We then introduce visualization experiments of classical and quantum flows of liquid helium using micron-size hydrogen/deuterium particles and our recent results on transition to quantum turbulence based on the revisited experiments with a torsionally oscillating disc.

1Supported by GACR P203/11/0442 and 203/14/02005S
5:23PM L41.00007 Quantised vortices in polariton lattices, NATALIA BERLOFF, Skoltech (Russia) and University of Cambridge (UK) — The first comprehensive treatment of quantised vorticity in the light of research on vortices in modern fluid mechanics appeared in Russell Donnelly’s seminal research papers and summarized in his 1991 book “Quantized Vortices in Helium II.” Recently quantised vortices have been studied in polariton condensates. Polaritons are the mixed light-matter quasi-particles that are formed in the strong exciton-photon coupling regime. Under non-resonant optical excitation rapid relaxation of carriers and bosonic stimulation result in the formation of a non-equilibrium polariton condensate characterized by a single many-body wave-function, therefore, naturally possessing quantized vortices. Polariton condensates can be imprinted into any two-dimensional lattice by spatial modulation of the pumping laser and form vortices via interacting outflows from the pumping sites. Optically pumped polariton condensates can be injected in lattice configurations with arbitrary density profiles offering the possibility to control the kinetics of the condensate and therefore the number and location of vortices. I will present some new developments in theoretical and experimental studies of quantized vortices in polariton condensates and discuss possible practical implementations of polariton lattices.

5:36PM L41.00008 Visualization in quantum fluids, DANIEL LATHROP, University of Maryland — The motion of quantized vortices, which are topological phase defects analogous to crystalline dislocations, substantially controls the dynamics of quantum fluids. Quantized vortices have been observed in superfluid 4He and trapped atom systems, and have been inferred in superfluid 3He and neutron stars. Long-range quantum order parameter a number of related physical phenomena, including superfluidity, trapped-atom Bose-Einstein condensates, superconductivity, ferromagnetism, anti-ferromagnetism, lasers, and the Higgs mechanism. While superfluidity in 4He is one of the first discovered of these phenomena, it is one of the least understood, given that the strongly interacting nature of helium makes theory difficult, and that development of local experimental probes is lagging. The advent of flow visualization of particles that trace quantized vortices has led to many advances. That progress was caused by repeated suggestions from Russ Donnelly, Joe Niemela, and Joe Vinen. Those suggestions led the team, including Gregory P. Bewley, K.R. Sreenivasan and myself, to venture into the quantum fluid realm.

5:49PM L41.00009 Direct and inverse energy transfers in superfluid turbulence, CARLO BARENGHI, Newcastle University — Three dimensional isotropic homogeneous turbulence is characterized by a direct cascade of energy from large to small length scales. A reversed flux of energy from small to large length scales is observed in two dimensional turbulence, and, in the presence of a strong anisotropy, the vorticity is constrained to thin, quantum vortex filaments of atomic thickness which interact with each other by a distance of the order of the vortex core thickness. Recent experimental, theoretical, and numerical studies have demonstrated evidence of the direct energy cascade in superfluid turbulence. In this work we show that in superfluid turbulence the more subtle inverse energy transfer described by Biferale et al. can be directly understood in physical space from the geometry of reconnecting vortex filaments, and argue that the effect has been detected in experiments with liquid helium.

6:02PM L41.00010 Russell Donnelly’s last legacy: Pursuing grid turbulence in superfluid 3He, GARY IHAS, JHIEE YANG, University of Florida — Quantum turbulence, a tangle of quantized vortex lines in a superfluid, may hold significant keys to understanding all types of turbulence. Russell Donnelly pioneered this line of research, beginning with studies of grid turbulence probed by second sound. The apparatus built by Russell and his students, with significant up-grades, is now being used at the University of Florida to continue his work on decaying grid turbulence in superfluid 3He. The Oregon work used a 1cm wide circular channel, while the Florida work has been in both 1 cm and 5 cm square channels. The larger channel allows detailed study of the increase in eddy size before saturation at the channel walls during the decay process. Power law fits of the turbulence decay in time allow comparison with theory, work intended to be Russell Donnelly’s last experiment.

6:15PM L41.00011 Decaying turbulence at the laminar-turbulence transition in a pipe, NIGEL GOLDENFELD, TSUNG-LIN HSIEH, HONG-YAN SHIH, Loomis Laboratory of Physics, University of Illinois at Urbana-Champaign — As a follow-up to Donnelly’s pioneering research on the decay of superfluid turbulence in a pipe, we have studied a different regime: transition to turbulence. Near the onset to turbulence in a pipe, turbulent puffs decay either directly or through splitting, with characteristic time-scales that exhibit a super-exponential dependence on Reynolds number. Using direct numerical simulations of transitional pipe flow, we show that a collective mode, a so-called zonal flow emerges at large scales, activated by anisotropic turbulent fluctuations, as measured in terms of Reynolds stress. This zonal flow imposes a shear on the turbulent fluctuations that tends to suppress their anisotropy, leading to stochastic oscillatory dynamics. These results motivate the proposal that the laminar-turbulence non-equilibrium phase transition can be modeled by an effective theory, usefully thought of as predator-prey dynamics, leading to a predicted universality class of directed percolation.

6:28PM L41.00012 Granular materials and their connection to Russell Donnelly, BOB BEHRINGER, Duke University — I have known Russell Donnelly for most of my professional career. Our interests in liquid helium, in fluid dynamics and instabilities, the use of helium to study convection, are all points of significant overlap. Trying to decide when I first met Russ is hard, so let me focus on one year, 1996. That year Russ came to Duke to give the Fritz London Memorial Lecture. It was also a year that I remember because the DFD meeting at Syracuse had a large number of talks dealing with granular materials. I first became interested in this field as a system to test for hydrodynamic-like instabilities. Russ had been a strong supporter of including granular flows in DFD meetings, and the field was well on its way in 1996. In fact, the predicted instability was not there, but many striking and novel phenomena were: interest in the physics of granular materials has grown dramatically since then. In this talk, I will explore some of the roots of granular physics and the connections to fluid flows. In particular, flowing grains show surprising fluctuations in forces that are tied to novel structures known as force chains. These structures also play a key role in how granular materials become “solids,” i.e. jam. The idea of jamming arose in early work by M. Cates et al. and by A. Liu and S. Nagel. We have recently shown that the nature of jamming is fundamentally changed when the grains have friction or shape, which are general properties of grains that form many everyday materials. In general, understanding how granular materials transition between jammed or unjammed draws substantially on statistical physics—something that would have strongly appealed to Russ.

1Supported in part by US NSF #1007937.

1This work was partially supported by the National Science Foundation through grant NSF-DMR-1044901.

1Work supported by NSF-DMR1206351, DMS1248071, NASA NNX15AD38G, and the W.M. Keck Foundation.
8:00AM M1.00001 Thermal striping in nuclear reactors: POD analysis of LES simulations and experiment1, ELIA MERZARI, ANL, ANDRES ALVAREZ, MIT, OANA MARIN, ALEKSANDR OBABKO, STEVE LOMPERSKI, SHASHI S. VARTHAL, ANL — Thermal fatigue caused due to thermal striping impacts design and analyses of a wide-range of industrial apparatus. This phenomena is of particular significance in nuclear reactor applications, primarily in sodium cooled fast reactors. In order to conduct systematic analyses of the thermal striping phenomena a simplified experimental set-up was designed and built at Argonne National Laboratory. In this set-up two turbulent jets with a temperature difference of about 20K were mixed in a rectangular tank. The jets entered the tank via 2 hexagonal inlets. Two different inlet geometries were studied, both experimentally and via high-fidelity LES simulations. Proper Orthogonal Decomposition (POD) was performed on the turbulent velocity field in the tank to identify the most dominant energetic modes. The POD analyses of the experimental data in both inlet geometrical configurations were compared with LES simulations. Detailed POD analyses are presented to highlight the impact of geometry on the velocity and thermal fields. These can be correlated with experimental and numerical data to assess the impact of thermal striping on the design of the upper plenum of sodium-cooled nuclear reactors.

1ALCF

8:13AM M1.00002 Planar Flow Casting: Crystalline and Non-crystalline Ribbon Formation1, JOSEPH MATTSON, Cornell University, ERIC THEISEN, Metglas Inc., PAUL STEEN, Cornell University — Planar flow casting (PFC) is a single-stage continuous casting process used in the production of thin metallic sheets. Molten metal is ejected from a reservoir and forced through a small gap to freeze against a translating substrate. The process is typically ‘feed limited’ which means that an imposed pressure drop determines the flow rate of metal to the solidification front, and thus the ultimate thickness of the solid sheet. Depending on the molten alloy, the substrate heat sink can provide sufficient cooling rates to produce a glassy (amorphous) metal. Otherwise, a crystalline solid is the result. In this talk, by relating ribbon thickness to residence time for both amorphous and crystalline products, we address the question: to what extent is processing ‘blind’ to the solidification mechanism?

1Support from National Science Foundation ( Awards 1400964 and 0960045); Metglas Inc

8:26AM M1.00003 ABSTRACT WITHDRAWN –

8:39AM M1.00004 Squeeze flow with capillary effect in Nano Imprint Lithography (NIL) process, BHARATH BABU NUNNA, SHIQIANG ZHUANG, EON SOO LEE1, New Jersey Inst of Tech — In the Nano imprinting process, the resist forms the required nano structures upon the squeeze effect, between the polymer mold and substrate. Due to this squeeze effect the resist will experience the squeeze force, which leads the fluid (resist) to fill the cavity of the mold. But the fluid due to its natural phenomenon undergoes a capillary effect that contributes to the fluid movement. In this presentation the fluid dynamics of the resist in the cavity upon the squeeze force and capillary effect are examined in detail. The study of the resist flow in the nano imprint lithography (NIL) process helps to define the exact required squeeze force to obtain the enhanced quality of nano structures.

1Prof. Eon Soo Lee is the “Principal Investigator” of the lab.

8:52AM M1.00005 A comparative study of SU-8 and wax based paper-fluidic device with respect to channel geometry, JINKEE LEE, ALI TURAB JAFRY, HOSUB LIM, Sungkyunkwan University — Although many fabrication techniques of paper fluidic devices have evolved as a result of its broad application spectrum and ease of use, the technology has still barely scratched the surface of its potential in terms of its underlying fundamental principle i.e. fluid flow analysis. In this paper we have studied the comparison of flow profile attained by using two of the most promising techniques of photolithography and wax printing from a hydrodynamic point of view. A modified protocol for synthesizing an SU-8 based channel and wax based channel is created by optimizing few process parameters to our equipment. Water and oil (oleic acid) are chosen as hydrophilic and hydrophobic fluids respectively and their flow is analyzed in straight channels within paper device. A new approach to vary flow velocity is described in detail involving dots as resistance inside the paper channel. Observing the length-time curve for the two fluids, it becomes evident that both follow the Lucas-Washburn equation if the width of channel is large enough. Various configurations of dots reveal different longitudinal flow velocity implying its application in simultaneous addition of chemicals without the need to change channel width or length

9:05AM M1.00006 Fluid Mechanics Optimising Organic Synthesis, EVGENIA LEIVADAROU1, STUART DALZIEL2, University of Cambridge — The Vortex Fluidic Device (VFD) is a new “green” approach in the synthesis of organic chemicals with many industrial applications in biodiesel generation, cosmetics, protein folding and pharmaceutical production. The VFD is a rapidly rotating tube that can operate with a jet feeding drops of liquid reactants to the base of the tube. The aim of this project is to explain the fluid mechanics of the VFD that influence the rate of reactions. The reaction rate is intimately related to the intense shearing that promotes collision between reactant molecules. In the VFD, the highest shear rates are found at the bottom of the tube in the Rayleigh and the Ekman layer and at the walls in the Stewardson layers. As a step towards optimising the performance of the VFD we present experiments conducted in order to establish the minimum drop volume and maximum rotation rate for maximum axisymmetric spreading without fingering instability.

1PhD candidate, Department of Applied Mathematics and Theoretical Physics
2Reader in Fluid Mechanics, Department of Applied Mathematics and Theoretical Physics, Director of GK Batchelor Laboratory

9:18AM M1.00007 Microfluidic IEF technique for sequential phosphorylation analysis of protein kinases, NAKCHUL CHOI, SIMON SONG, HOSEOK CHOI, BU-TAEK LIM, YOUNG-PIL KIM, Hanyang Univ. — Sequential phosphorylation of protein kinases play the important role in signal transduction, protein regulation, and metabolism in living cells. The analysis of these phosphorylation cascades will provide new insights into their physiological functions in many biological functions. Unfortunately, the existing methods are limited to analyze the cascade activity. Therefore, we suggest a microfluidic isoelectric focusing technique (µIEF) for the analysis of the cascade activity. Using the technique, we show that the sequential phosphorylation of a peptide by two different kinases can be successfully detected on a microfluidic chip. In addition, the inhibition assay for kinase activity and the analysis on a real sample have also been conducted. The results indicate that µIEF is an excellent means for studies on phosphorylation cascade activity.
9:31AM M1.00008 Capacitive Deionization: Performance Improvement Using Multistep Buffered Arrangement and Ordered Mesoporous Carbon Electrodes. YASAMIN SALAMAT, CARLOS RIOS PEREZ, ANVESH GURIJALA, RANDALL ERB, CARLOS HIDROVO, Northeastern University — Capacitive deionization (CDI) is an emerging novel technology for water treatment which uses an electrical field to adsorb ions to oppositely charged high porous media. The most distinguished feature of CDI is its ability to retrieve a fraction of the energy consumed for desalination during the regeneration cycle. Here, we propose a new architecture to improve the overall performance of CDI. In this method, an array of CDI cells are connected in series with solution buffers in between them. The buffer solution homogenizes the outlet concentration of the preceding cell and supplies a constant concentration reservoir for the next cell. The performance of the proposed CDI system with two CDI cells and one solution buffer was compared with a two-cascaded-cells array with no solution buffer. The obtained results demonstrated the superiority of the proposed buffered system, in terms of desalination percentage. In addition, a new method for fabricating ordered mesoporous carbon electrodes was introduced aimed at reducing the electrical resistance of the system and enhancing its adsorption capacity. Performance of the electrodes was evaluated using Electrochemical Impedance Spectroscopy (EIS) and Cyclic Voltammetry (CV). The proposed methods provide great potentials for CDI to be implemented in larger scales and industrial applications.

1The author would like to thank Jabulani Barber for all his help and support on manufacturing the carbon films.

9:44AM M1.00009 Characterization of desalination performance of CDI electrode materials using extended electroimpedance spectroscopy. CARLOS RIOS PEREZ, Northeastern University, ELLEN WILKES, The University of Texas at Austin, CARLOS HIDROVO, Northeastern University — A comprehensive characterization of porous materials developed for capacitive deionization (CDI) electrodes is very important for the future of this desalination technology. Traditional methods assess the adsorption performance of the electrodes using gas adsorption techniques and electrochemical tests. However, these results fail at comparing quantitatively the performance of different electrode materials. This presentation proposes using a combination of extended electroimpedance spectroscopy (EIS) tests and BET analysis to appraise the amount of salt adsorbed in a flow-by CDI system. The extended EIS experiments were analyzed using an equivalent circuit with three characteristic tiers that represent the dominant ionic migration processes with different time-scales: electro adsorption of ions in the micropores, migration of ion from bulk solution through macropores, adsorption of ions from the bulk solution. The results obtained show a very good agreement between characterization and desalination performance experiments for three commercial electrodes with different structure topology.

9:57AM M1.00010 Crackle noise from high-speed free-shear-flow turbulence. DAVID BUCHTA, JONATHAN FREUND, University of Illinois at Urbana-Champaign — High-thrust jet engines radiate a particularly intense and distinct sound that has become known as ‘crackle’. Its root mechanisms are not fully understood, though they are thought to involve nonlinear acoustics because the sound waves appear steepened. They also have a positive skewness, pressure maxima are stronger than minima, for unknown reasons. We use direct numerical simulations of free-shear-flow turbulence with Mach numbers ranging from $M = 0.9$ to $3.5$ to study the very near acoustic field and the turbulence interactions. Results indicate that crackle is insensitive to Reynolds number for the range considered, though DNS is restricted to modest Reynolds numbers. The very near field is teeming with weak, nonlinearly interacting Mach-like waves. Locally, these waves generate intense pressure fluctuations, especially as they merge. We observe that skewness changes are small over the propagation distances simulated, though more significant changes are to be expected over larger propagation distances. The source of the peculiar skewness is thus near or within the turbulence. Simulations modulating the underlying unstable linear modes reveal a sensitivity to crackle and are used to assess the role of large-scale structures in its source.

Tuesday, November 24, 2015 8:00AM - 9:57AM — Session M2 Flames: Non-premixed Flames 101 - Fabrizio Bisetti, King Abdullah University of Science and Technology

8:00AM M2.00001 Scaling of velocity and mixture fraction fields in laminar counterflow configurations. FABRIZIO BISSETTI, King Abdullah University of Science and Technology, Saudi Arabia, GIANFRANCO SCRIBANO, University of Nottingham Malaysia Campus, Malaysia — Counterflow configurations are widely used to characterize premixed, nonpremixed, and partially premixed laminar flames. We performed a systematic analysis of the velocity and mixture fraction fields in the counterflow configuration and obtained scaling laws, which depend on two suitable nondimensional numbers: (i) the Reynolds number based on the bulk velocity $U$ and half the separation distance between the nozzles $L$, and (ii) the ratio of the separation distance $H = 2L$ to the nozzle diameter $D$. Our study combines velocity measurements via Particle Image Velocimetry, detailed two-dimensional simulations including the nozzle geometry, and an exhaustive analysis of the data based on the nondimensional numbers. The flow field is shown to be moderately sensitive to the Reynolds number and strongly affected by the ratio $H/D$. By describing the self-similar behavior of the flow field in counterflow configurations comprehensively, our results provide a systematic explanation of existing burner designs as well as clear guidelines for the design of countercflows for pressurized nonpremixed flames. Finally, questions related to the limitations of one-dimensional models for counterflows are addressed conclusively.

8:13AM M2.00002 A model for the effective/turbulent Lewis numbers in turbulent non-premixed flames. NICHOLAS BURALI, GUILLAUME BLANQUART, Caltech — Turbulent mixing has a strong impact on the structure of turbulent premixed and non-premixed flames. Experimental results have highlighted that, with growing turbulence intensities, turbulent transport becomes gradually dominant over molecular mixing. As this occurs, the (average) turbulent flame structure transitions to that of a unity Lewis number unstretched flame. In the current work, this transition is characterized by developing an a priori model for the effective/turbulent species Lewis numbers in turbulent non-premixed flames. This model is developed from a Reynolds-averaged Navier-Stokes (RANS) formulation of the species and energy transport equations, and validated using existing experimental and numerical data. The results of this work provide a simple framework to estimate the Lewis numbers to be used in one-dimensional flame calculations for chemistry tabulation.
8:26AM M2.00003 Hydrodynamic and chemical effects of hydrogen dilution on soot evolution in turbulent nonpremixed bluff body ethylene flames , SILLI DENG, MICHAEL E. MUELLER, Princeton University, QING N. CHAN, The University of New South Wales, NADER H. QAMAR, FCT-Combustion, BASSAM B. DALLY, ZEYAD T. ALWAHABI, GRAHAM J. NATHAN, The University of Adelaide — A turbulent nonpremixed bluff body ethylene/hydrogen (volume ratio 2:1) flame is studied and compared with the ethylene counterpart [Mueller et al. Combust. Flame, 160, 2013]. Similar to the ethylene bluff body flame, a low-strain recirculation zone, a high-strain neck region, and a downstream jet-like region are observed. However, the maximum soot volume fraction in the recirculation zone of the hydrogen diluted case is significantly lower than the ethylene case. Large Eddy Simulation is used to further investigate soot evolution in the recirculation zone and to elucidate the role of hydrogen dilution. Since the central jet Reynolds numbers in both cases are the same (approximately 30,900), the jet velocity of the hydrogen diluted case is higher, resulting in a shorter and leaner recirculation zone. In addition, hydrogen dilution chemically suppresses soot formation due to the reduction of C/H ratio. Consequently, the reduction of the soot volume fraction for the hydrogen diluted ethylene flame is attributed to two major effects: hydrodynamic and chemical effects.

8:39AM M2.00004 In-Situ Analysis of Gradient Trajectories in a Reactive Turbulent Shear Flow , FELIX DIETZSCH, MICHAEL GAUDING, CHRISTIAN HASSE, TU Bergakademie Freiberg — Most understanding of turbulent fine-scale mixing has been gained from conditional statistics. Conditional statistics are examined along gradient trajectories, which constitute a natural, intrinsic coordinate system of the underlying scalar field. Statistics along gradient trajectories contain information about the temporal mechanism of turbulent mixing and combustion. Analyzing these statistics is an important step to understand the transient behavior of the interaction between turbulence and chemistry. The tracking of gradient trajectories is very challenging and has to be conducted in-situ in order to capture the smallest time-scales. The analysis is based on a direct numerical simulation of a turbulent diffusion flame exhibiting extinction and reignition.

8:52AM M2.00005 Local Velocity Field Measurements towards Understanding Flame Stabilization of Turbulent Non-premixed Jet Flames in Vitiated Coflow1, ARAVIN RAMACHANDRAN, ANIRUDH REDDY MOTHE, VENKATESWARAN NARAYANASWAMY, North Carolina State University — Turbulent combustion of a non-premixed methane jet issuing into a vitiated coflow is being studied in our lab. Flame luminosity studies demonstrated three dominant characteristic flame motions – a stable flame base (Mode A), complete blowout (Mode B), and partial blowout followed by re-anchoring of the flame by autoignition kernels (Mode C). The experiments presented in this work focused on Mode A, and were carried out over a range of oxidizer temperatures, oxygen molefractions, and fuel jet Reynolds numbers. Measurements of 2-D velocity fields near the base of the lifted jet flame were obtained using Particle Image Velocimetry (PIV) with the objective to delineate the dominant mechanisms involved in the flame stabilization. Statistical analysis of these instantaneous velocity fields will be presented, which shows non-trivial contributions from autoignition kernels as well as edge flame propagation towards flame stabilization. The effect of vortices and high local strain rates was observed to produce local extinctions and destabilize the flame, indicating their role as precursors to (unstable) Mode B and Mode C motions.

9:05AM M2.00006 Effects of local extinction on mixture fraction and scalar dissipation statistics in turbulent nonpremixed flames , ANTONIO ATTILI, FABRIZIO BISSETTI, King Abdullah Univ of Sci & Tech (KAUST) — Passive scalar and scalar dissipation statistics are investigated in a set of flames achieving a Taylor’s scale Reynolds number in the range 100 \( \leq Re_T \leq 150 \) [Attili et al. Comb. Flame 161, 2014; Attili et al. Proc. Comb. Inst. 35, 2015]. The three flames simulated show an increasing level of extinction due to the decrease of the Damköhler number. In the case of negligible extinction, the non-dimensional scalar dissipation is expected to be the same in the three cases. In the present case, the deviations from the aforementioned self-similarity manifests itself as a decrease of the non-dimensional scalar dissipation for increasing level of local extinction, in agreement with recent experiments [Karpetis and Barlow Comb. Inst. 30, 2005; Sutton and Driscoll Combust. Flame 160, 2013]. This is caused by the decrease of molecular diffusion due to the lower temperature in the low Damköhler number cases. Probability density functions of the scalar dissipation \( \chi \) show rather strong deviations from the log-normal distribution. The left tail of the pdf scales as \( \chi^{-1/2} \) while the right tail scales as \( e^{-\chi^{1/2}} \), in agreement with results for incompressible turbulence [Schumacher et al. J. Fluid Mech. 531, 2005].

9:18AM M2.00007 Turbulent non - premixed flames driven by the Richtmyer-Meshkov Instability , HILDA VARSHOCI, NITESH ATTAL, PRAVEEN RAMAPRABHU, University of North Carolina at Charlotte — We report on Direct Numerical Simulations of shock-induced mixing between fuel (H\(_2\)) and Oxidizer (O\(_2\)) streams separated by a sharp interface and driven by the Richtmyer-Meshkov instability (RMI). The resulting non-premixed flame is dominated by vigorous mixing that is a consequence of deposition of baroclinic vorticity at the interface. Such RMI-driven flames, when properly controlled, could play a decisive role in improving the performance of supersonic combustors such as scramjets. While the majority of past research efforts in this area have focused on the shock-bubble flame interaction, our configuration is fundamentally different and involves a planar shock interacting with a planar interface. This allows for the placement of well-defined, precisely controlled initial perturbations on the planar surface. Furthermore, the interface is statistically homogenous in all directions perpendicular to shock traverse, thus rendering the problem amenable to reduced-order 1D modeling of planar-averaged quantities. From detailed, high-resolution DNS [1], we describe flow and flame characteristics of a repeatedly reshocked turbulent RMI flame. We observe that with each reshock event, fresh deposition of vorticity on the already nonlinear interface greatly enhances mixing and combustion. [1] Attal, N., et al. Comput. Fluids 107 (2015): 59-76.

9:31AM M2.00008 Strain rate effects on soot evolution in turbulent nonpremixed flames , JEFFRY K. LEW, MICHAEL E. MUELLER, Princeton University, SALEH MAHMOUD, ZEYAD T. ALWAHABI, BAS-SAM B. DALLY, GRAHAM J. NATHAN, The University of Adelaide — Large Eddy Simulations (LES) of turbulent nonpremixed ethylene/hydrogen/nitrogen (2/2/1 by volume) jet flames are conducted to investigate the effects of global strain rate on soot evolution. The exit strain rate is varied by fixing the Reynolds number as the burner diameter and exit velocity are altered. A detailed integrated LES approach is employed that includes a nonpremixed flamelet model that accounts for heat losses from radiation, a transport equation model to account for unsteadiness in polycyclic aromatic hydrocarbon (PAH) evolution, a detailed soot model based on the Hybrid Method of Moments [Mueller et al. Combust. Flame 156 (2009)], and a novel presumed subfilter PDF model for soot-turbulence interactions. As the strain rate increases, the maximum soot volume fraction decreases due to the suppression of PAH formation. This trend with increasing strain rate is validated against experimental measurements conducted at The University of Adelaide.

1NSF Grant CBET-1511216
A complete description of the physical mechanism could provide insight into ways to design engine inlets for efficient mixing in combustion applications.

Tuesday, November 24, 2015 8:00AM - 10:10AM –
Session M3 Suspensions: General 102 - Themistoklis Sapsis, MIT

8:00AM M3.00001 Mobility of membrane-trapped particles, HASSAN MASOUD, University of Nevada, Reno, HOWARD STONE, Princeton University — The translation or diffusion of particles along membranes or interfaces is of interest because it is a model system for describing basic features of interfacial hydrodynamics. It is also important in cellular signalling in biology and biophysics, and it can be used to deduce the rheological properties of surface films. Here, we consider the translational mobility of spherical and oblate spheroidal particles protruding into the surrounding subphase liquid. Both the subphase and surface film contribute to the resistance experienced by the particle, which is calculated as a function of the degree of protrusion as well as the viscosity contrast between the surface film and the surrounding fluid. The calculations are based on a combination of a perturbation expansion involving the particle shape and the Lorentz reciprocal theorem. It appears that just considering one term of the expansions is in very good agreement with available analytical and numerical results.

8:13AM M3.00002 Eulerian flow modeling of suspensions containing interacting nano-particles: application to colloidal film drying, I. GERGIANAKIS, M. MEIRELES, P. BACCHIN, Y. HALLEZ, University of Toulouse - LGC — Nano-particles in suspension often experience strong non-hydrodynamic interactions (NHIs) such as electrostatic repulsions. In this work, we present and justify a flow modeling strategy adapted to such systems. Earlier works on colloidal transport in simple flows, were based on the solution of a transport equation for the colloidal volume fraction with a known fluid velocity field and a volume-fraction-dependent diffusion coefficient accounting for mass fluxes due to NHIs. Extension of this modelling to complex flows requires the coupled resolution of a momentum transport equation for the suspension velocity field. We use the framework of the Suspension Balance Model to show that in the \( \text{Pe} \ll 1 \) regime relevant here, the average suspension velocity field is independent of NHIs between nanoparticles, while the average fluid phase and solid phase velocity fields both always depend on the NHIs. Lastly, we apply this modelling strategy to the problem of the drying of a colloidal suspension in a micro-evaporator [Merlin et al. 2012, Soft Matter]. The influence of the effective Peclet number on the 1D/2D character of the flow is evaluated and the possible colloidal film patterning due to defaults of substrate topography is commented.

8:26AM M3.00003 Migration of rigid particles in two-phase shear flow of viscoelastic fluids, PATRICK ANDERSON, NICK JAENSSON, MARTIEN HULSEN, Eindhoven Universiy of Technology — In the Stokes regime, non-Brownian rigid particles in a slow flow will not migrate across streamlines if the fluid is Newtonian. In viscoelastic fluids, however, particles will migrate across streamlines away from areas of higher elastic stresses, e.g. towards the outer cylinder in a wide-gap Couette flow. This migration is believed to be due to a difference in normal stresses. We simulate the two-phase case where this difference in normal stresses is not due to the flow field, but rather due to the properties of the fluids. We apply the diffuse-interface model for the interface between the two fluids, which can naturally handle a changing topology of the interface, e.g. during particle adsorption. Furthermore, the diffuse-interface model includes an accurate description of surface tension and can be used for a moving contact line. A sharp interface is assumed between the particles and the fluids. Initially, a particle is placed close to an interface of two fluids with different viscoelastic properties in a shear flow. We show that based on the properties of the fluids and the interfacial tension, four regimes can be defined: 1) migration away from the interface, 2) halted migration towards the interface, 3) adsorption of the particle at the interface and 4) penetration of the particle into the other fluid. This research forms part of the research programme of the Dutch Polymer Institute (DPI), Project #746.

8:39AM M3.00004 Elliptical Particle Clustering in Cellular Flows, SEVERINE ATIS, THEMISTOKLIS SAPSIS, THOMAS PEACOCK, Massachusetts Inst of Tech-MIT — The transport of finite-sized objects by fluid flows is relevant to a wide variety of phenomena, such as debris transport on the ocean surface or bacteria advection in fluid environment. The shape of the advected objects can strongly alter their coupling with the surrounding flow field, and hence, greatly affecting their dispersion by the flow. We present the results of investigations of the behavior of neutrally buoyant, elliptical particles in two-dimensional cellular flows. We find that their trajectories, and overall organization, are markedly different than for spherical particles, with clear clustering for the elliptical particles associated with vortices.

8:52AM M3.00005 General and Rigorous Framework for Particle Adsorption on Fluid Interfaces, MARKUS SCHMUCK, Maxwell Institute for Mathematical Sciences and Heriot-Watt University, SERAFIM KALLIADASSIS, Imperial College London — Considers two arbitrary immiscible phases where one phase contains small and neutral particles of uniform size on the order of the interface. The wetting properties of the particles are accounted for by the contact angle formed at the interface between the two fluid phases and the particles. Under experimental observations that particles are adsorbed on the interface to lower the interfacial energy and hence the surface tension as well, we formulate a free-energy functional that accounts for these physical effects. By making use of variational methods and a consistent gradient flow formulation, we obtain partial differential equations that systematically describe the location of the interface and the density of the particles in the fluid phases and the interface. Our numerical experiments analyse the time evolution of the surface tension, the particle concentration, and the free energy over time and reflect the crucial property of a decreasing free energy under particle adsorption.

1Supported by EPSRC Grant No. EP/H034587, European Framework 7 via Grant No. 214919 (Multiflow) and ERC Advanced Grant No. 247031
9:05AM M3.00006 Gravity-Driven Particle-Laden Flow on an Incline, Sarah Burnett, University of North Carolina, Chapel Hill, Jesse Kreger, Occidental College, Hanna Kristensen, Pepperdine University, Andrew Stocker, University of San Francisco, Jeffrey Wong, Li Wang, Andrea Bertozzi, UCLA Math Department — We present experimental results of the height profile of particle-laden viscous thin films with finite volume on an incline. For high angles of inclination and high concentrations of mixtures, negatively buoyant particles undergo resuspension then accumulate at the front of the suspending fluid; this leads to the development of a particle-rich ‘ridge’. Theoretically, the ridge corresponds to the shocks which take on two characteristic shapes: singular and double shocks. We observe the presence of both formations experimentally by varying the volume of the slurry and compare our results to the theoretical model. Our research also investigates the dependence of the fingering instability as the inclination angle or particle to liquid concentration is changed. The slurries have similar dynamics to those used in coating flow techniques and other industrial applications.

9:18AM M3.00007 Particle-induced viscous fingering, Feng Xu, Dillon Strack, Texas A&M University, Grace Fomani, Tuskegee Institute, Celina Lopez, University of Texas at El Paso, Sungyon Lee, Texas A&M University — A novel fingering instability is experimentally observed when a mixture of particles and viscous oil is injected radially into a Hele-Shaw cell. According to the Saffman-Taylor theory, the equivalent configuration without particles exhibits no fingering. To characterize this particle-induced instability, a series of experiments are conducted with varying particle volume fractions, flow rates, and gap thicknesses. The experimental results show that the onset of fingering is most directly affected by the particle volume fraction: the interface is stable when the particle concentration is lower than 10% and becomes unstable with more pronounced fingering patterns with an increasing concentration. The interfacial instability is accompanied by regularized clusters of particles inside the displacing phase, each of which corresponds to a finger. Based on the key observations, we discuss the physical mechanism that drives the instability.

9:31AM M3.00008 Reverse drainage of a particle-laden thin film, Antonio Mastroberardino, Penn State University, Erie, Javed Siddique, Penn State University, York — Gravity driven flow of a thin film on a solid surface is a critical aspect of numerous industrial applications including the production of foams, wire and optical coating applications, and more recently, the coating of medicines. In particular, the system in consideration is influenced by competing forces, such as gravity, surface tension, and viscous forces, to name a few. Recently, several researchers have investigated the control of thin film flow by adding a controlling agent that allows for manipulation of the fluid via an external field. In this talk, we investigate the case in which the controlling agent is an aqueous suspension of magnetic nanoparticles and the external field is a non-uniform magnetic field. We formulate a mathematical model based on lubrication theory, present numerical solutions for the evolution of the film, and discuss the roles played by the key parameters of the system.

9:44AM M3.00009 ABSTRACT WITHDRAWN —

9:57AM M3.00010 Transport and Deposition of Electrosprayed Nanoparticles, Nicholas Brown, Paul Chiarot, Suny Binghamton — In an electrospray, high electric potentials are utilized to generate a fine aerosol of a conductive solvent. For this study, the solvent consisted of nanoparticles dispersed in alcohol. The nanoparticle suspensions act as printable nanoparticle inks. In this process, a glass capillary tube is held as a high electric potential relative to a grounded reference plate located below the tip. Droplets are ejected from the tube and directed towards the ground plate. If the solvent is sufficiently volatile, it will rapidly evaporate while the droplets are in flight (due to the high surface area to volume ratio) leaving behind dry, highly charged nanoparticles. The droplets/nanoparticles are deposited onto a target substrate that is placed on the ground plate. The transport of any individual droplet/nanoparticle from the emitter tip to the target substrate is a stochastic process. This transport can be modeled using a Monte Carlo simulation. The probability of an individual particle being deposited at a given location on the target substrate is directly related to the electric potential at that location. In other words, the probability function that determines the deposition is directly related to the electric potential at the substrate. The total potential is comprised of the applied electric potential required to generate the electrospray, the induced charge on the target dielectric, and the charge on the individual particles themselves. We report on the structure of droplet/nanoparticle deposits printed using electrospray. The evolution of the deposit is investigated over time using experimental studies and Monte Carlo simulations. The deposit structure passes through four distinct regimes that are characterized by repeatable bulk features.

Tuesday, November 24, 2015 8:00AM - 10:10AM —

Session M4 Particle-Laden Flows: Clustering and Dispersion II 103 - Rodney O. Fox, Iowa State University

8:00AM M4.00001 Strongly coupled turbulent gas-particle flows in vertical channels†, Rodney O. Fox, Department of Chemical and Biological Engineering, Iowa State University, Jesse CapeceLatro, Coordinated Science Laboratory, University of Illinois at Urbana-Champaign, Olivier DesJardins, Sibley School of Mechanical and Aerospace Engineering, Cornell University — Eulerian-Lagrangian (EL) simulations of strongly coupled (high mass loading) gas-particle flows in vertical channels are performed with the purpose of exploring the fundamental physics of fully developed, wall-bounded multiphase turbulence. An adaptive spatial filter is developed that accurately decomposes the total granular energy of the particles into correlated and uncorrelated components at each location in the wall-normal direction of the flow. In this manner, Reynolds- and phase-averaged (PA) two-phase turbulence statistics up to second order are reported for both phases and for three values of the PA mean fluid velocity. As expected due to the high mass loading, in all cases the turbulence production due to mean drag dominates production due to mean shear. A multiphase LRR-IP Reynolds-stress turbulence model is developed to predict the turbulent flow statistics as a function of the wall-normal distance. Using a correlation for the vertical drift velocity developed from the EL data, the turbulence model predictions agree satisfactorily with all of one-point EL statistics for the vertical channel flows, as well as for the homogeneous cluster-induced turbulence (CIT) statistics reported previously.

†Funded by U.S. National Science Foundation (CBET-1437865)
8:13AM M4.00002 Analysis and Comparison with DNS of a Stochastic Model for the Relative Motion of High-Stokes-Number Particles in Isotropic Turbulence. ROHIT DHARIWAL, SARMA RANI, Univ of Alabama - Huntsville, DONALD KOCH, Cornell University — In an earlier work, Rani, Dhariwal, and Koch (JFM, Vol. 756, 2014) developed an analytical closure for the diffusion current in the PDF transport equation describing the relative motion of high-Stokes-number particle pairs in isotropic turbulence. In this study, an improved closure was developed for the diffusion coefficient, such that the motion of the particle-pair center of mass is taken into account. Using the earlier and the new analytical closures, Langevin simulations of pair relative motion were performed for four particle Stokes numbers, $St_n = 10, 20, 40, 80$ and at two Taylor micro-scale Reynolds numbers $Re_\lambda = 70, 131$. Detailed comparisons of the analytical model predictions with those of DNS were undertaken. It is seen that the pair relative motion statistics obtained from the improved theory show excellent agreement with the DNS statistics. The radial distribution functions (RDFs), and relative velocity PDFs obtained from the improved-closure-based Langevin simulations are found to be in very good agreement with those from DNS. It was found that the RDFs and relative velocity RMS increased with $Re_\lambda$ for all $St_n$. The collision kernel also increased strongly with $Re_\lambda$, since it depended on the RDF and the radial relative velocities.

8:26AM M4.00003 Experimental studies of gas-particle mixtures under sudden expansion, HEATHER ZUNINO, RONALD ADRIAN, AMANDA CLARKE, Arizona State University, ARIZONA STATE UNIVERSITY COLLABORATION, UNIVERSITY OF FLORIDA COLLABORATION — High-speed video cameras and pressure sensors were used to capture the movement of a particle bed due to a passing expansion fan created by a diaphragm burst in a shock tube. The particle bed is placed on the high-pressure side ($p_d$) of the shock tube. Once the diaphragm bursts, it expands upward into the low-pressure region ($p_l$). Several interesting structures are captured and examined, including instabilities located at the top surface of the particle bed and particle vacant regions within the bed. These features are discussed along with their relevance to the spikes of material seen radially ejected outward during a cylindrical explosion. The characteristics of this flow are compared for several different pressure regimes. Two-dimensional and three-dimensional Fourier analyses are used to further explore and measure the frequency of the features emerged.

1Supported by the U.S. Department of Energy, National Nuclear Security Administration, Advanced Simulation and Computing Program, as a Cooperative Agreement under the Predictive Science and Academic Alliance Program, under Contract No. DE-NA0002378.

8:39AM M4.00004 Study of snow-atmosphere interactions over an Antarctic surface using large eddy simulations coupled with a Lagrangian stochastic model. FRANCESCO COMOLA, MARCO GIOMETTO, ERNESTO TRIJILLO, KATHERINE LEONARD, Ecole Polytechnique Federale de Lausanne, TED MAKSYM, Woods Hole Oceanographic Institution, MARC PARLANGE, University of British Columbia, MICHAEL LEHNING, Ecole Polytechnique Federale de Lausanne — The need for a better understanding of fluid and morphodynamic processes over Antarctic sea ice motivates the development of detailed models of small-scale snow-atmosphere interactions. At large scales, these interactions drive spatial patterns of snow distribution and snow transport from the marginal ice to the sea. However, challenges arise when representing the detailed sequence of processes involved, such as aerodynamic entrainment, particle dynamics, feedback on fluid momentum and particle impacts. We use a Lagrangian stochastic model coupled to large eddy simulations to represent particle trajectories in turbulent flows. An immersed boundary method is used to represent the underlying surface and a dynamic surface roughness model is used to account for the drag induced by the subgrid-scale roughness. The model is set up for an Antarctic sea ice floe over which pre- and post-storm snow distributions were measured using a terrestrial laser scanner. The dataset, collected as part of the Sea Ice Physics and Ecosystem Experiment 2, indicates marked changes in the snow distribution as a result of snow drift, providing valuable testing grounds for the model. Model results are in agreement with blowing snow concentrations at different heights and with the observed patterns of erosion and deposition.

8:52AM M4.00005 Preferential accumulation and enhanced relative velocity of inertial droplets due to interactions with homogeneous isotropic turbulence. COLIN BATES-SON, ALBERTO ALISED, University of Washington — We present results from wind tunnel experiments on the evolution of small inertial ($d \approx 10 - 200 \mu m$) water droplets in homogeneous, isotropic, slowly decaying grid turbulence. High-speed imaging and a Particle Tracking algorithm are used to calculate relative velocity distributions. We analyze the preferential concentration, via the 2D Radial Distribution Function, and enhanced relative velocity of droplets resulting from their inertial interactions with the underlying turbulence. The two-dimensional particle velocities, measured from multi-image tracks along a streamwise plane, are conditionally analyzed with respect to the distance from the nearest particle. We focus on the non-normality of the statistics for the particle-particle separation velocity component to examine the influence of the inertial interaction with the turbulence on the dynamics of the droplets. We observe a negative bias (in the mean and mode) in the separation velocity of particles for short separations, signaling a tendency of particles to collide more frequently than a random agitation by turbulence would predict. The tails of the distribution are interpreted in terms of the collision/coalescence process and the probability of collisions that do not lead to coalescence.

9:05AM M4.00006 Bringing Clouds into Our Lab! - The Influence of Turbulence on the Early Stage Rain Droplets, MEHMET ALTUG YAVUZ, RUDIE KUNNEN, GERTJAN HEIJST, HERMAN CLERCX, Fluid Dynamics Laboratory, Department of Physics, Eindhoven University of Technology, Netherlands — We are investigating a droplet-laden flow in an air-filled turbulence chamber, forced by speaker-driven air jets. The speakers are running in a random manner; yet they allow us to control and define the statistics of the turbulence. We study the motion of droplets with tunable size ($St_\text{d} = 0.13 - 9$) in a turbulent flow, mimicking the early stages of raindrop formation. 3D Particle Tracking Velocimetry (PTV) together with Laser Induced Fluorescence (LIF) methods are chosen as the experimental method to track the droplets and collect data for statistical analysis. Thereby it is possible to study the spatial distribution of the droplets in turbulence using the so-called Radial Distribution Function (RDF), a statistical measure to quantify the clustering of particles. Additionally, 3D-PTV technique allows us to measure velocity statistics of the droplets and the influence of the turbulence on droplet trajectories, both individually and collectively. In this contribution, we will present the clustering probability quantified by the RDF for different Stokes numbers. We will explain the physics underlying the influence of turbulence on droplet cluster behavior.

1This study supported by FOM/NWO Netherlands.
9:18AM M4.00007 Quantification of statistical phenomena in turbulent dispersions¹

Matthew Yates, David Hann, Buddhika Hewakandamby, University of Nottingham — Understanding of turbulent dispersions is of great importance for environmental and industrial applications. This includes developing a greater understanding of particle movement in atmospheric flows, and providing data that can be used to validate CFD models aimed at producing more accurate simulations of dispersed turbulent flows, aiding design of many industrial components. Statistical phenomena in turbulent dispersions were investigated using Particle Image Velocimetry. Experiments were carried out in a two dimensional channel over a Reynolds number range of 10000-30000, using water and 500 micron hydrogel particles. Particles were injected at the channel entrance, and dispersion properties were characterised at different distances downstream from the injection point. Probability density functions were compiled for the velocity components of the hydrogels for differing flow conditions. Higher order PDFs were constructed to investigate the mechanics of collisions between hydrogel particles, allowing for calculation of the coefficient of restitution. PIV algorithms were used to create velocity maps for the continuous phase for varying dispersed phase fractions.

¹Thanks to support of Chevron grant as part of TMF consortium

9:31AM M4.00008 Turbulent Soret Effect¹

Dhrubaditya Mitra, Nils Erlend L. Haugen, NTNU, SINTEF Energy Research, Igor Rogachevskii, Ben-Gurion University of the Negev — We study, turbophoresis—the clustering properties of heavy inertial passive particles in a inhomogeneous turbulent flow—by direct numerical simulation of inhomogeneously forced turbulence in a periodic box without walls. The forcing is a periodic function of one coordinate direction. The inertial particles cluster near the minima of the turbulent kinetic energy. Drawing analogy with Soret effect in near-equilibrium thermodynamics, we can describe the flux of particles as a sum of two fluxes, described by two turbulent transport coefficients, turbulent diffusion of particles and turbophoretic coefficient. The second (turbophoretic) flux is assumed to be proportional to the gradient of turbulent intensity. The ratio of these two coefficients would be analogous to Soret coefficient, hence we call this the turbulent Soret coefficient. Our numerical calculation show that such a description is a good description of our data. Furthermore, we find that the turbulent Soret coefficient is a non-monotonic function of the particle inertia (described by the Stokes number), i.e. beyond a critical Stokes number the clustering of the particles decreases, but in a smooth manner.

¹Swedish Research Council, Wallenberg Foundations

9:44AM M4.00009 Caustics and collisions of inertial particles

Rama Govindarajan, S. Ravichandran, Tata Institute of Fundamental Research, Centre for Interdisciplinary Sciences, Samriddhi Ray, International Centre for Theoretical Sciences, Tata Institute of Fundamental Research, P. Deepu, Tata Institute of Fundamental Research, Centre for Interdisciplinary Sciences — Caustics are formed when inertial particles of very different velocities collide in a flow, and are a consequence of the dissipative nature of particle motion in a suspension. Using a simple model for vortex-dominated flow with heavy particles, we suggested that sling caustics form only within a neighbourhood around a vortex, the square of which radius is proportional to the product of circulation and particle inertia. Particles starting close to this critical radius congregate close together, resulting in large spikes in (Lagrangian) number density. We test these predictions by counting the number of collisions of particles in a randomly forced flow and correlating the collision locations with vorticity. We also study the effect of caustics on droplet growth in a super-saturated environment. We hope that these studies will be of interest in long-standing problems of physical interest such as the mechanism of broadening of droplet spectra in a turbulent flow.

9:57AM M4.00010 Condensing aerosol Dynamics in homogeneous isotropic turbulence

Amjad Alsharawi, Antonio Attili, Fabrizio Bisetti, King Abdullah University of Science and Technology, Clean Combustion Research Center Team — The interaction of a condensing aerosol with homogeneous isotropic turbulence is simulated at $Re_k = 95$. The simulation consists of a three-dimensional direct numerical simulation of homogeneous isotropic turbulence with a statistically stationary forced velocity field. Patches of dry and cold gas mix with patches of hot carrier gas saturated with vapor of a condensable species, inducing the homogeneous nucleation of particles due to supersaturation. An approach based on the quadrature method of moments and a Lagrangian numerical scheme is adopted for the transport and dynamics of the liquid droplets [Attili & Bisetti, Comp. Fluids 84, 2013; Zhou et al., Phys. Fluids 26, 2014]. Two regimes related to the eddy turnover timescale are observed, i.e., a nucleation regime and a consumption regime [Alsharawi & Bisetti, J. Aerosol Sci., 81, 2015]. In the nucleation regime, at short eddy turnover timescales, mixing is fast enough to suppress nucleation by mixing the fluid to the mean state at which nucleation vanishes. In the consumption regime, at long eddy turnover timescales, mixing is slow and nucleation continues until it is suppressed by the consumption of the vapor phase due to the growth of the droplets.

Tuesday, November 24, 2015 8:00AM - 10:10AM — Session M5 Nonlinear Dynamics: General 104 - Kevin Mitchell, University of California, Merced

8:00AM M5.00001 Topological entropy and symbolic dynamics for three-dimensional fluid mixing

Kevin Mitchell, Bryan Maelfeyt, Joshua Arenson, University of California, Merced — Topological entropy provides an important metric of mixing in two-dimensional fluid flows; it has led to a quantification of mixing for various periodic stirring protocols and other chaotic flows. In this context, the topological entropy can be viewed as the exponential growth rate of a material line. In this talk, we explain how one can compute an analogous entropy for topological mixing in three-dimensional flows. This entropy amounts to an exponential growth rate in the size of material sheets. Our approach involves the extraction of symbolic dynamics from the intersections of two-dimensional stable and unstable manifolds of the flow field. We illustrate our theory with a mathematical model of a chaotic ring vortex.

8:13AM M5.00002 Passive scalars chaotic dynamics induced by two vortices in a two-layer geophysical flow with shear and rotation

Eugene Ryzhov, Pacific Oceanological Institute — Vortex motion in shear flows is of great interest from the point of view of nonlinear science, and also as an applied problem to predict the evolution of vortices in nature. Considering applications to the ocean and atmosphere, it is well-known that these media are significantly stratified. The simplest way to take stratification into account is to deal with a two-layer flow. In this case, vortices perturb the interface, and consequently, the perturbed interface transits the vortex influences from one layer to another. Our aim is to investigate the dynamics of two point vortices in an unbounded domain where a shear and rotation are imposed as the leading order influence from some generalized perturbation. The two vortices are arranged within the bottom layer, but an emphasis is on the upper-layer fluid particle motion. Point vortices induce singular velocity fields in the layer they belong to, however, in the other layers of a multi-layer flow, they induce regular velocity fields. The main feature is that singular velocity fields prohibit irregular dynamics in the vicinity of the singular points, but regular velocity fields, provided optimal conditions, permit irregular dynamics to extend almost in every point of the corresponding phase space.
8:26AM M5.00003 Maximal stochastic transport in the Lorenz equations , SAHIL AGARWAL, JOHN WETTLAUFER, Yale University, University of Oxford — We calculate the stochastic upper bounds for the Lorenz equations using an extension of the background method. In analogy with Rayleigh-Benard convection the upper bounds are for heat transport versus Rayleigh number. As might be expected the stochastic upper bounds are larger than the deterministic counterpart of Souza and Doering (2015), but their variation with noise amplitude exhibits surprising behavior. Below the transition to chaotic dynamics the upper bounds increase monotonically with noise amplitude. However, in the chaotic regime this monotonicity is lost; at a particular Rayleigh number the bound may increase or decrease with noise amplitude. The origin of this behavior is the coupling between the noise and unstable periodic orbits. This is confirmed by examining the close returns plots of the full solutions to the stochastic equations. Finally, we note that these solutions demonstrate that the effect of noise is equivalent to the effect of chaos.

8:39AM M5.00004 Flow primitives to manipulate the dynamics of inertial particles , SENBAGARAMAN SUDARANAN, PHANINDRA TALLAPRAGADA, Clemson University — The nonlinear dynamics of inertial particles in many microfluidic settings occurs in flows whose main feature is cell-like structures created due to specific distributions of vorticity. Examples include Dean vortices, Taylor-Couette vortices and streaming vortex cells. To obtain insights into the motion of inertial particles in such complex flows, in possibly confined domains, we develop certain flow primitives generated by point-vortex like structures. We model the motion of spherical inertial particles by the Maxey-Riley equation. With this governing equation the inertial particles demonstrate sensitive dependence on size and initial conditions in the fluid flow generated by the flow primitives. Size based particle segregation, trapping particles at the centers of vortex cores or on limit cycles is shown to be possible. We demonstrate some of these phenomena using Lagrangian coherent structures (LCS).

8:52AM M5.00005 Neimark-Sacker bifurcation and evidence of chaos in a discrete dynamical model of walkers , AMINUR RAHMAN, New Jersey Institute of Technology — Bouncing droplets on a vibrating fluid bath can exhibit wave-particle behavior, such as being propelled by the waves they generate. These droplets seem to walk across the bath, and the trajectories of these walkers can exhibit stochastic dynamical behavior. These strong indications of chaos, but many of the interesting dynamical properties have yet to be proven. In recent years discrete dynamical models have been derived and studied numerically. We prove the existence of a Neimark-Sacker bifurcation for a variety of eigenmode shapes of the waves from one such model. Then we reproduce numerical experiments and produce new numerical experiments and apply our theorem to the test functions used for that model in addition to new test functions. Further evidence of chaos is shown by numerically studying a global bifurcation.

9:05AM M5.00006 Topology of three-dimensional steady cellular flow in a two-sided lid-driven cavity , FRANCESCO ROMANO, TU Wien, STEFAN ALBENSOEDER, Carl von Ossietzky Universität Oldenburg, HENDRIK KUHLMANN, TU Wien — The topology of a laminar three-dimensional flow in a rectangular lid-driven cavity is investigated. A two-dimensional flow in the (x,y) plane is driven by two facing walls moving in opposite directions with equal velocities. The cross-sectional aspect ratio in the (x,y)-plane is 1.7. The cavity is assumed to be infinitely extended in the spanwise (z) direction. At a Reynolds number $Re = 212$ the flow becomes three-dimensional via an elliptic instability resulting in a steady cellular flow with spanwise half-period of $\lambda_z/2 = 1.365$. The nonlinear steady flows at $Re = 500$ and 700 are accurately computed using a Chebyshev spectral collocation method. The flow is analyzed with respect to regular (KAM tori) and chaotic regions. The shape of the KAM tori and associated closed streamlines as well as their dependence on the Reynolds number is discussed. Further considerations will be given to the symmetry, period and minimum distance between the KAM tori and the cavity walls.

9:18AM M5.00007 Using Persistent Homology to Describe Rayleigh-Bénard Convection1 , JEFFREY TITHOF, BALACHANDRA SURI, Georgia Institute of Technology, MU XU, Virginia Tech, MIROSŁAW KRAMAR, RACHEL LEVANGER, KONSTANTIN MISCHAIKOW, Rutgers University, MARK PAUL, Virginia Tech, MICHAEL SCHATZ, Georgia Institute of Technology — Complex spatial patterns that exhibit aperiodic dynamics commonly arise in a wide variety of systems in nature and technology. Describing, understanding, and predicting the behavior of such patterns is an open problem. We explore the use of persistent homology (a branch of algebraic topology) to characterize spatiotemporal dynamics in a canonical fluid mechanics problem, Rayleigh Bénard convection. Persistent homology provides a powerful mathematical formalism in which the topological characteristics of a pattern (e.g. the midplane temperature field) are encoded in a so-called persistence diagram. By applying a metric to measure the pairwise distances across multiple persistence diagrams, we can quantify the similarities between different states in a time series. Our results show that persistent homology yields new physical insights into the complex dynamics of large spatially extended systems that are driven far-from-equilibrium.

9:31AM M5.00008 Characterizing mixing in time periodic planar flows through the topology of almost cyclic sets , PRADEEPS RAO, MathWorks Inc., MARK STREMLER, SHANE ROSS, Virginia Tech — Almost Invariant Sets (AIS) can be used to identify coherent structures that move as Almost Cyclic Sets (ACS) for time-periodic planar flows. The relative motion of the ACS identified using the second most dominant eigenvector of the reversible matrix obtained from the discretized Perron-Frobenius operator provides a reduced order model for quantifying transport. This has been shown through the application of the Thurston-Nielsen classification theorem to the topology of the motions of the ACS for certain time periodic driven cavity Stokes flows. We extend this notion to more general flows with inertial effects. We provide a recipe for identifying the ACS whose dynamics provide a reduced order model for predicting mixing efficiency for such flows.

9:44AM M5.00009 Computing the Evans function via solving a linear boundary value ODE , COLIN WAHL, University of Wisconsin-Madison, ROSE NGUYEN, The University of Texas at Austin, NATHANIEL VENTURA, SUNY-Binghamton, BLAKE BARKER, BJORN SANDSTEDE, Brown University — Determining the stability of traveling wave solutions to partial differential equations can oftentimes be computationally intensive but of great importance to understanding the effects of perturbations on the physical systems (chemical reactions, hydrodynamics, etc.) they model. For waves in one spatial dimension, one may linearize around the wave and form an Evans function - an analytic Wronskian-like function which has zeros that correspond in multiplicity to the eigenvalues of the linearized system. If eigenvalues with a positive real part do not exist, the traveling wave will be stable. Two methods exist for calculating the Evans function numerically: the exterior-product method and the method of continuous orthogonalization. The first is numerically expensive, and the second reformulates the originally linear system as a nonlinear system. We develop a new algorithm for computing the Evans function through approximate linear boundary-value problems. This algorithm is cheaper than the previous methods, and we prove that it preserves analyticity of the Evans function. We also provide error estimates and implement it on some classical one- and two-dimensional systems, one being the Swift-Hohenberg equation in a channel, to show the advantages.

---

1This work is supported under NSF grant DMS-1125302.
Fluid flow regimes are often categorized based on the qualitative patterns observed by visual inspection of the flow field. For example, bluff body wakes are traditionally classified based on the number and groupings of vortices shed per cycle (e.g., 2S, 2P, P+S), as seen in snapshots of the vorticity field. Subsequently, the existence and nature of these identified flow regimes can be explained through dynamical analyses of the fluid mechanics. Unfortunately, due to the need for manual inspection, the approach described above can be impractical for studies that seek to learn flow regimes from large volumes of numerical and/or experimental snapshot data. Here, we appeal to established techniques from machine learning and data-driven dynamical systems analysis to automate the task of learning flow regimes from snapshot data. Moreover, by appealing to the dynamical structure of the fluid flow, this approach also offers the potential to reveal flow regimes that may be overlooked by visual inspection alone. Here, we will introduce the methodology and demonstrate its capabilities and limitations in the context of several model flows.

Tuesday, November 24, 2015 8:00AM - 10:10AM – Session M6 Nonlinear Dynamics: Transition and Turbulence

8:00AM M6.00001 Invariant solutions organizing turbulence in pipe flow experiments, SEBASTIAN ALTMEYER, JAKOB KHREN, MARKUS SCHANER, BJORN HOF, Institute of Science and Technology — A large number of unstable invariant solutions, e.g. traveling waves (TWs) or (relative-) periodic orbits, has been discovered and numerically studied in recent years for pipe flow. The proposed role of such states as building blocks of turbulence is however less clear and so far only limited experimental evidence has been provided. In experiments we used a modulated pipe segment to impose a certain symmetry on the experimental velocity field and in the non-modulated downstream pipe traveling waves could be observed persisting for many wavelengths. Measured velocity fields (PIV) were used as initial conditions for a numerical Newton search and converged to the exact invariant traveling wave solutions. All the experimentally observed TW’s correspond to lower branch states that are close to the laminar turbulent boundary (edge). Correspondingly in the experiments as the waves proceeded downstream flows would typically relaminarize but occasionally the TW’s would grow to turbulence. The latter observation confirms the relevance of these invariant states for the transition process.

8:13AM M6.00002 Experimental observations of direct laminar-turbulent transition in counter-rotating Taylor-Couette flow1, CHRISTOPHER CROWLEY, MICHAEL KRYGIER, Georgia Inst of Tech, DANIEL BORRERO-ECHEVERRY, Reed College, ROMAN GRIGORIEV, MICHAEL SCHATZ, Georgia Inst of Tech — The transition to turbulence in counter-rotating Taylor-Couette flow typically occurs through a sequence of supercritical bifurcations of stable flow states (e.g. spiral vortices, interpenetrating spirals (IPS), and wavy interpenetrating spirals). Coughlin and Marcus have proposed a mechanism by which these laminar spiral flows undergo a secondary instability that leads to turbulence. We report the discovery of a counter-rotating regime ($Re_{out} = −1000, Re_{in} ≈ 640$) of small aspect ratio/large radius ratio Taylor-Couette flow ($Γ = 5.26 / η = 0.91$), where the system bypasses the primary instability to stable laminar spirals and instead undergoes a direct transition to turbulence as the inner cylinder rotation rate is slowly increased. This transition is mediated by an unstable IPS state. We study the transition experimentally using flow visualization and tomographic PIV, and show that it is both highly repeatable and that it shows hysteresis as the inner cylinder rotation rate is decreased. As $Re_{in}$ is decreased, the turbulent flow relaminarizes into an intermediate, stable IPS state. Decreasing $Re_{in}$ further returns the system back to circular Couette flow.

This study was supported by NSF DMS-1125302 and NSF CMMI-1234436

8:26AM M6.00003 Numerical investigation of direct laminar-turbulent transition in counter-rotating Taylor-Couette flow, MICHAEL KRYGIER, ROMAN GRIGORIEV, Georgia Institute of Technology — A direct transition from laminar to turbulent flow has recently been discovered experimentally in the small-gap Taylor-Couette flow with counter-rotating cylinders. The subcritical nature of this transition is a result of relatively small aspect ratio, $Γ = 5.26$; for large $Γ$ the transition is supercritical and involves an intermediate stable state (Coughlin & Marcus, 1996) — interpenetrating spirals (IPS). We investigate this transition numerically to probe the dynamics in regimes inaccessible to experiments for a fixed $Re_{in} = −1000$ by varying $Re_i$. The numerics reproduce all the experimentally observed features and confirm the hysteretic nature of the transition. As $Re_i$ is increased, the laminar flow transitions to turbulence, with an unstable IPS state mediating the transition, similar to the Tollmien-Schlichting waves in plate Poiseuille flow. As $Re_i$ is decreased, turbulent flow transitions to a stable, temporally chaotic IPS state. This IPS state further transitions to either laminar or turbulent flow as $Re_i$ is decreased or increased. The stable IPS state is reminiscent of the pre-turbulent chaotic states found numerically in plane Poiseuille flow (Zammert & Eckhardt, 2015), but previously never observed experimentally.

8:39AM M6.00004 Rare event statistics and characteristic lifetimes in transient turbulence, TOBIAS KREILOS, LAURELINE HENTGEN, EPFL - Lausanne, BRUNO ECKHARDT, Fachbereich Physik, Universit"at Marburg, TOBIAS SCHNEIDER, EPFL - Lausanne — We numerically study transitional turbulence in plane Couette flow at Reynolds numbers where turbulence is transient. Monitoring the distance to the edge of chaos for a large number of turbulent trajectories, we compute the return period for reaching the vicinity of the edge. The measured return period of the turbulent state is linearly correlated with the characteristic lifetime of the decay. This suggests a way of predicting characteristic lifetime of transient shear turbulence.

8:52AM M6.00005 ABSTRACT WITHDRAWN —
9:05AM M6.00006 System Size Dependence of Finite-Amplitude Thresholds for Transition to Turbulence in Taylor-Couette Flow1, DANIEL BORRERO-ECHEVERRYY, BENJAMIN MORRISON, EVAN PEAIRS, Department of Physics, Reed College — Despite centuries of study, fluid dynamicists are still unable to explain why a large class of flows, including pipe flow and plane Couette flow, become turbulent. Hydrodynamic stability theory predicts these flows should be stable to infinitesimal perturbations, which means finite-amplitude perturbations need to be applied to destabilize them. We present the results of a series of experiments studying such subcritical transitions to turbulence in linearly-stable configurations of Taylor-Couette flow. In particular, we discuss how the stability of these flows depends on the size and duration of the applied perturbation as the aspect ratio of the experimental apparatus is varied. We show that for experimental configurations where the end caps rotate with the outer cylinder, the stability of the flow is enhanced at small aspect ratios. We find that at sufficiently high Reynolds numbers, perturbations must exceed a critical amplitude before the transition to turbulence can be triggered. The scaling of this threshold with Re appears to be different than that which has been reported for other linearly-stable shear flows.

1This work was supported by Reed College’s Summer Scholarship Fund, the James Borders Physics Student Fellowship, and the Reed College Science Research Fellowship. We also thank H.L. Swinney, who kindly donated the apparatus used in these experiments.

9:18AM M6.00007 Sudden relaminarisation and lifetimes in forced isotropic turbulence, MORITZ LINKMANN, ALEXANDER MOROZOV, Univ of Edinburgh — We demonstrate an unexpected connection between isotropic turbulence and wall-bounded shear flows. We perform direct numerical simulations of isotropic turbulence forced at large scales at moderate Reynolds numbers and observe sudden transitions from chaotic dynamics to a spatially simple flow, analogous to the laminar state in wall bounded shear flows. We find that the survival probabilities of turbulence are exponential and the typical lifetimes increase super-exponentially with the Reynolds number, similar to results on relaminarisation of localised turbulence in pipe and plane Couette flow. Results from simulations suggest that the observable large-scale flow to random perturbations of variable amplitude demonstrate that it is a linearly stable simple exact solution that can be destabilised by a finite-amplitude perturbation, like the Hagen-Poiseuille profile in pipe flow. Our results suggest that both isotropic turbulence and wall-bounded shear flows qualitatively share the same phase-space dynamics.

9:31AM M6.00008 Disruption of the vortex-wave interaction self-sustaining process in stratified plane Couette flow, T. S. EAVES, DAMTP, University of Cambridge, C. P. CAULFIELD, BPI & DAMTP, University of Cambridge — Minimal seeds for turbulence, initial conditions of smallest possible energy density $E_0 = E_c$, that eventually transition to turbulence, closely follow the edge manifold in state space before leaving the edge manifold for the turbulent attractor. The trajectories visit a number of coherent states, exact solutions to the Navier–Stokes equations, that are embedded within the edge manifold. In unstratified plane Couette flow these ‘edge states’ are manifestations of the ‘self-sustaining process’ (SSP) of Waleffe (1997) or the ‘vortex-wave interaction’ (VWI) of Hall and Smith (1991). We show that in density stratified plane Couette flow where both a constant, statically stable density difference $2\Delta \rho$ and a constant velocity difference $2\Delta U$ is maintained across a channel of depth $2h$, these states differ from the unstratified states at very small bulk Richardson numbers $Ri_B = g \Delta \rho h / \rho_0 \Delta U^2$ (where $g$ is the gravitational acceleration and $\rho_0 \gg \Delta \rho$ is a reference density) and that the new states are not of SSP/VWI type. We present a scaling argument to show this is to be expected for $Ri_B \geq O(1/Re)$, where $Re = \Delta U h / \nu$ and $\nu$ is the kinematic viscosity, and investigate the mechanisms through which the SSP/VWI states breakdown.

9:44AM M6.00009 Connecting exact coherent states to turbulent dynamics in channel flow1, JAE SUNG PARK, MICHAEL D. GRAHAM, University of Wisconsin-Madison — The discovery of nonlinear traveling wave solutions to the Navier-Stokes equations or exact coherent states has greatly advanced the understanding of the nature of turbulent shear flows. These solutions are unstable saddle points in state space, while the time evolution of a turbulent flow is a dynamical trajectory wandering around them. In this regard, it is of interest to investigate how closely the turbulent trajectories approach these invariant states. Here, we present connections between turbulent trajectories and one intriguing solution family in channel flow. A state space visualization of turbulent trajectories is presented in a three-dimensional space. The lifetime of the trajectories is well represented by closeness to two distinct solutions resembling in many ways the active and hibernating phases of minimal channel turbulence (Xi & Graham PRL 2010). The connections are then examined by comparing mean profiles and flow structures. More importantly, the connections are confirmed by calculating the $L_2$ distance between the trajectories and the traveling waves. Lastly, paths of an intermittent bursting phenomenon are identified in state space and the relationship between bursting paths and the traveling waves or hibernating turbulence is further discussed.

1This work was supported by the Air Force Office of Scientific Research through grant FA9550-15-1-0062 (Flow Interactions and Control Program).

9:57AM M6.00010 Temporal and spatial intermittencies within Newtonian turbulence1, ANUBHAV KUSHWAHA, MICHAEL GRAHAM, Univ of Wisconsin, Madison — Direct numerical simulations of a pressure driven turbulent flow are performed in a large rectangular channel. Intermittent high- and low-drag regimes within turbulence that have earlier been found to exist in minimal channels have been observed both spatially and temporally in full-size turbulent flows. These intermittent regimes, namely, “active” and “hibernating” turbulence, display very different structural and statistical features. We adopt a very simple sampling technique to identify these intermittent intervals, both temporally and spatially, and present differences between them in terms of simple quantities like mean-velocity, wall-shear stress and flow structures. By conditionally sampling of the low wall-shear stress events in particular, we show that the Maximum Drag Reduction (MDR) velocity profile, that occurs in viscoelastic flows, can also be approached in a Newtonian-fluid flow in the absence of any additives. This suggests that the properties of polymer drag reduction are inherent to all flows and their occurrence is just enhanced by the addition of polymers. We also show how the intermittencies within turbulence vary with Reynolds number.

1The work was supported by AFOSR grant FA9550-15-1-0062.
8:00AM M7.00001 Effect of asynchrony on numerical simulations of fluid flow phenomena, ADITYA KONDURI, BRYAN MAHONEY, DIEGO DONZIS, Texas A&M University — Designing scalable CFD codes on massively parallel computers is a challenge. This is mainly due to the large number of communications between processing elements (PEs) and their synchronization, leading to idling of PEs. Indeed, communication will likely be the bottleneck in the scalability of codes on Exascale machines. Our recent work on asynchronous computing for PDEs based on finite-differences has shown that it is possible to relax synchronization between PEs at a mathematical level. Communication then proceed regardless of the status of communication, reducing the idle time of PEs and improving the scalability. However, accuracy of the schemes is greatly affected. We have proposed asynchrony-tolerant (AT) schemes to address this issue. In this work, we study the effect of asynchrony on the solution of fluid flow problems using standard and AT schemes. We show that asynchrony creates additional scales with low energy content. The specific wavenumbers affected can be shown to be due to distinct effects: the randomness in the arrival of messages and the corresponding switching between schemes. Understanding these errors allow us to effectively control them, rendering the method’s feasibility in solving turbulent flows at realistic conditions on future computing systems.

8:13AM M7.00002 A Multiscale/Multifidelity CFD Framework for Robust Simulations, SEUNGOON LEE, Brown University, YANNIS KEVREKIDIS, Princeton University, GEORGE KARNIADAKIS, Brown University — We develop a general CFD framework based on multifidelity simulations to target multiscale problems but also resilience in exascale simulations, where faulty processors may lead to gappy simulated fields. We combine approximation theory and domain decomposition together with machine learning techniques, e.g. co-Kriging, to estimate boundary conditions and minimize communications by performing independent parallel runs. To demonstrate this new simulation approach, we consider two benchmark problems. First, we solve the heat equation with different patches of the domain simulated by finite differences at fine resolution or very low resolution but also with Monte Carlo, hence fusing multifidelity and heterogeneous models to obtain the final answer. Second, we simulate the flow in a driven cavity by fusing finite difference solutions with solutions obtained by dissipative particle dynamics – a coarse-grained molecular dynamics method. In addition to its robustness and resilience, the new framework generalizes previous multiscale approaches (e.g. continuum-atomic) in a unified parallel computational framework.

8:26AM M7.00003 Fast linear solvers for variable density turbulent flows, HADI POURANSARI, ALI MANI, ERIC DARVE, Stanford University — Variable density flows are ubiquitous in variety of natural and industrial systems. Two-phase and multi-phase flows in natural and industrial processes, astrophysical flows, and flows involved in combustion processes are such examples. For an ideal gas subject to low-Mach approximation, variations in temperature can lead to a non-uniform density field. In this work, we consider radiatively heated particle-laden turbulent flows as an example application in which density variability is resulted from inhomogeneities in the heat absorption by an inhomogeneous particle field. Under such conditions, the divergence constraint of the fluid is enforced through a variable coefficient Poisson equation. Inversion of the discretized variable coefficient Poisson operator is difficult using the conventional linear solvers as the size of the problem grows. We apply a novel hierarchical linear solve algorithm based on low-rank approximations. The proposed linear solver could be applied to variety of linear systems arising from discretized partial differential equations. It can be used as a standalone direct-solver with tunable accuracy and linear complexity, or as a high-accuracy pre-conditioner in conjunction with other iterative methods.

8:39AM M7.00004 Direct numerical simulation of fluid-particle mass, momentum, and heat transfers in reactive systems, ABDELKADER HAMMOUTI, ANTHONY WACHS, IFP Energies Nouvelles — Many industrial processes like coal combustion, catalytic cracking, gas phase polymerization reactors and more recently biomass gasification and chemical looping involve two-phase reactive flows in which the continuous phase is a fluid and the dispersed phase consists of rigid particles. Improving both the design and the operating conditions of these processes represents a major scientific and industrial challenge in a context of markedly rising energy cost and sustainable development. Thus, it is above all important to better understand the coupling of hydrodynamic, chemical and thermal phenomena in those flows in order to be able to predict them reliably. The aim of our work is to build up a multi-scale modelling approach of reactive particulate flows and at first to focus on the development of a microscopic-scale including heat and mass transfers and chemical reactions for the prediction of particle-laden flows in dense and dilute regimes. A first step is the upgrading and the validation of our numerical tools via analytical solutions or empirical correlations when it is feasible. These couplings are implemented in a massively parallel numerical code that already enable to take a step towards the enhanced design of semi-industrial processes.

8:52AM M7.00005 FFeasibility of Amazon Cloud Computing Platform for Parallel Multi-phase Flow Simulations, COLE FRENIERE, ASHISH PATHAK, MEHDI RAESSI, University of Massachusetts Dartmouth, UNIVERSITY OF MASSACHUSETTS DARTMOUTH TEAM — The feasibility of Amazon’s Elastic Compute Cloud (EC2) service is evaluated as a resource for multi-phase flow simulations. The results for two multi-phase flow solvers are presented: a 2D GPU-accelerated serial code and a 3D MPI-parallel GPU-accelerated solver. Improving both the design and the operating conditions of these processes represents a major scientific and industrial challenge in a context of markedly rising energy cost and sustainable development. Thus, it is above all important to better understand the coupling of hydrodynamic, chemical and thermal phenomena in those flows in order to be able to predict them reliably. The aim of our work is to build up a multi-scale modelling approach of reactive particulate flows and at first to focus on the development of a microscopic-scale including heat and mass transfers and chemical reactions for the prediction of particle-laden flows in dense and dilute regimes. A first step is the upgrading and the validation of our numerical tools via analytical solutions or empirical correlations when it is feasible. These couplings are implemented in a massively parallel numerical code that already enable to take a step towards the enhanced design of semi-industrial processes.

9:05AM M7.00006 Repartitioning Strategies for Massively Parallel Simulation of Reacting Flow, PATRICK PISCINUERI, ANGEN ZHENG, PEYMAN GIVI, ALEXANDROS LABRINIDIS, PANOS CHRYSANTHIS, Univ of Pittsburgh — The majority of parallel CFD simulators partition the domain into equal regions and assign the calculations for a particular region to a unique processor. This type of domain decomposition is vital to the efficiency of the solver. However, as the simulation develops, the workload among the partitions often become uneven (e.g. by adaptive mesh refinement, or chemically reacting regions) and a new partition should be considered. The process of repartitioning adjusts the current partition to evenly distribute the load again. We compare two repartitioning tools: Zoltan, an architecture-agnostic graph repartitioner developed at the Sandia National Laboratories; and Paragon, an architecture-aware graph repartitioner developed at the University of Pittsburgh. The comparative assessment is conducted via simulation of the Taylor-Green vortex flow with chemical reaction.

1Funding from the National Science Foundation CBET - 1236462 and UMass Dartmouth OUR is gratefully acknowledged.
9:18AM M7.00007 A GPU-accelerated semi-implicit ADI method for incompressible and compressible Navier-Stokes equations\(^1\), SANGHYUN HA, DONGHYUN YOO, Pohang Univ of Sci & Tech — Utility of the computational power of Graphics Processing Units (GPUs) is elaborated for solutions of both incompressible and compressible Navier-Stokes equations. A semi-implicit ADI finite-volume method (J. Comp. Phys. V. 230 (2011), pp. 7400-7417) for integration of the incompressible and compressible Navier-Stokes equations, which are discretized on a structured arbitrary grid, is parallelized for GPU computations using CUDA (Compute Unified Device Architecture). In the semi-implicit ADI finite-volume method, the nonlinear convection terms and the linear diffusion terms are integrated in time using a combination of an explicit scheme and an ADI scheme. Inversion of multiple tri-diagonal matrices is found to be the major challenge in GPU computations of the present method. Some of the algorithms for solving tri-diagonal matrices on GPUs are evaluated and optimized for GPU-acceleration of the present semi-implicit ADI computations of incompressible and compressible Navier-Stokes equations.

\(^1\)Supported by the Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Science, ICT and Future Planning Grant NRF-2014R1A2A1A11049599.

9:31AM M7.00008 Discrete Particle Model for Porous Media Flow using OpenFOAM at Intel Xeon Phi Coprocessors \(^1\), ZHI SHANG, Center for Computation and Technology, Louisiana State University, LA 70808, USA, KRISHNASWAMY NANDAKUMAR, Cain Department of Chemical Engineering, Louisiana State University, LA 70802, USA, HONGGAO LIU, Center for Computation and Technology, Louisiana State University, LA 70808, USA, MAYANK TYAGI, Department of Petroleum Engineering, Louisiana State University, LA 70808, USA, JAMES A. LUPO, Center for Computation and Technology, Louisiana State University, LA 70808, USA, KARTEN THOMPSON, Department of Petroleum Engineering, Louisiana State University, LA 70808, USA — The discrete particle model (DPM) in OpenFOAM was used to study the turbulent solid particle suspension flows through the porous media of a natural dual-permeability rock. The 2D and 3D pore geometries of the porous media were generated by sphere packing with the radius ratio of 3. The porosity is about 38% same as the natural dual-permeability rock. In the 2D case, the mesh cells reach 5 million with 1 million solid particles and in the 3D case, the mesh cells reach 180 million with 5 million solid particles. The solid particles are distributed by Gaussian distribution from 20 \(\mu\)m to 180 \(\mu\)m with expectation as 100 \(\mu\)m. Through the numerical simulations, not only was the HPC studied using Intel Xeon Phi Coprocessors but also the flow behaviors of large scale solid suspension flows in porous media were studied.

\(^1\)The authors would like to thank the support by IPCC@LSU-Intel Parallel Computing Center (LSU # Y1SY1-1) and the HPC resources at Louisiana State University (http://www.hpc.lsu.edu).

Tuesday, November 24, 2015 8:00AM - 10:10AM — Session M8 Microscale Flows: Interfaces and Wetting 108 - Hooman Tafreshi, Virginia Commonwealth University

8:00AM M8.00001 Transition States for Submerged Superhydrophobic Surfaces: Partially-Pinned Air-Water Interface\(^1\), HOOMAN TAFRESHI, AHMED HEMEDA, Virginia Commonwealth University, VCU TEAM — The pressure at which a superhydrophobic surface transitions from the Cassie state to the Wenzel state is often referred to as the critical pressure. Our mathematical simulations have shown that the Cassie-to-Wenzel transition is a gradual process that takes place over a range of pressures as oppose to an event that happens at a certain pressure. During the transition period, the air-water interface may go through a series pinned, partially-pinned, and de-pinned states that depend on the geometry of the surface asperities. This in turn indicates that the drag-reduction effect produced by a submerged superhydrophobic surface can vary with the hydrostatic pressure, and is highly dependent on sharpness of the surface asperities. The study reported here reviews our recent discoveries in simulating the wetted area and drag reduction effect of superhydrophobic surfaces with different microstructures.

\(^1\)National Science Foundation CMM 1029924 and CBET 1402655 programs

8:13AM M8.00002 Instantaneous Slip Length in Superhydrophobic Microchannels\(^1\), AHMED HEMEDA, HOOMAN TAFRESHI, Virginia Commonwealth University, VCU TEAM — Superhydrophobic (SHP) surfaces can be used to reduce the skin-friction drag in a microchannel. This favorable effect, however, can deteriorate over time if the surface geometry is not designed properly. This study presents a mathematical means for studying the time-dependent drag-reduction in a microchannel enhanced with SHP grooves of varying geometries. The performance of an SHP groove is found to be dependent on the interplay between the effects of the apparent contact angle of the air-water interface and the initial volume of the groove. The instantaneous slip length is calculated by solving the Navier–Stokes equations for flow in a microchannel with such SHP grooves, and the results are compared with the studies in the literature.

\(^1\)National Science Foundation CMMI 1029924 and CBET 1402655

8:26AM M8.00003 Predicting apparent slip at liquid-liquid interfaces without an interface slip condition\(^1\), PIETRO POESIO, ANGELO DAMONE, University of Brescia, OMAR MATAR, Imperial College London — We show that if we include a density-dependent viscosity into the Navier–Stokes equations then we can describe, naturally, the velocity profile in the interfacial region, as we transition from one fluid to another. This requires knowledge of the density distribution (for instance, via Molecular Dynamics [MD] simulations, a diffuse-interface approach, or Density Functional Theory) everywhere in the fluids, even at liquid-liquid interfaces where regions of rapid density variations are possible due to molecular interactions. We therefore do not need an artificial interface condition that describes the apparent velocity slip. If the results are compared with the computations obtained from MD simulations, we find an almost perfect agreement. The main contribution of this work is to provide a simple way to account for the apparent slip at liquid-liquid interfaces without relying upon an additional boundary condition, which needs to be calculated separately using MD simulations. Examples are provided involving two immiscible fluids of varying average density ratios, undergoing simple Couette and Poiseuille flows.

\(^1\)MIUR through PRIN2012-NANOBridge; Royal Society International Exchange Scheme (IEI41486)
Investigations of complex multiphase systems and regions of thermodynamic stability of various fluid configurations. As a side note, this demonstrates that DFT is an excellent tool for the DFT calculations and demonstrate how this statistical-mechanical framework is capable of yielding complex fluid structure, interfacial tensions, filling and condensation producing very rich phase diagrams even for relatively simple geometries.

We will also discuss practical aspects that are sensitive to both the range of the intermolecular forces and interfacial fluctuation effects. These transitions compete with wetting, fluids how novel, continuous, interfacial transitions associated with the first-order prewetting line, can occur on steps, in grooves and in wedges, also relevant to nanofluidics, chemical- and bioengineering. In this talk we will show using a microscopic Density Functional Theory (DFT) for their bulk counterparts. As well as being of fundamental interest to the modern statistical mechanical theory of inhomogeneous fluids, these are also relevant to nano-fluidics, chemical- and bioengineering.

Imperial College London, London, UK — Fluids adsorbed at walls, in capillary pores and slits, and in more exotic, sculpted geometries such as patterned surfaces. In this talk we will discuss the experimental characterization of slip-length measurement using BBM, and how novel, continuous, interfacial transitions associated with the first-order prewetting line, can occur on steps, in grooves and in wedges, also relevant to nanofluidics, chemical- and bioengineering.

8:52AM M8.00005 Transient coating of substrates with variable topography by viscous films \(^1\), NIKOS LAMPROPOULOS, YIANNIS DIMAKOPOULOS, JOHN TSAMOPOULOS, Univ of Patras — We study the transient coating of substrates exhibiting orthogonal trench. We use the VoF method via OpenFOAM to solve the transient NS eqs on an unstructured grid, which dynamically undergoes local refinement around the interfaces. An Euler implicit method is used with adjustable time-step. The computational cost is reduced by parallel execution via MPI. Completely different wetting patterns result depending on the 3 dimensions of the topography, the capillary and Reynolds numbers and the dynamic contact angle. On one hand, continuous coating can be achieved in which the thin film of fluid wets the entire trench, while a steady flow is established upstream and downstream the topography. This is the desirable pattern in coating microelectronic devices for their protection and planarization. The other extreme possibility is that the film completely bypasses the trench, entrapping air inside it. This pattern reduces the drag coefficient on the film and, therefore, it is desirable in super-hydrophobic surfaces for microfluidic applications. Between these two extremes, a large variety of patterns exists in which the film partially wets the trench forming an air inclusion all along its bottom surface or its upstream or downstream inner corners or the film may breakup periodically. We produce comprehensive maps of film configurations covering a wide range of parameter values.

9:05AM M8.00006 Hydrodynamic fundamentals of slippage over a superhydrophobic surface \(^1\), CLARISSA SCHÖNECKER, DAVID SCHÄFFEL, KALOJAN KOYNOV, DORIS VOLLMER, HANS-JÜRGEN BUTT, Max-Planck-Institute for Polymer Research — Water easily slips over superhydrophobic surfaces, making such surfaces attractive for the development of functional coatings. While the global behavior of flow past superhydrophobic surfaces has been widely investigated, the local physical fundamentals leading to slippage still remain unclear. Using fluorescence correlation spectroscopy, we performed detailed measurements of the local slip length for water in the Cassie state on a structured superhydrophobic surface. In combination with numerical calculations of the flow, we revealed that the local slip length of a superhydrophobic surface is finite, non-constant and anisotropic. Furthermore, it can be strongly influenced by the presence of surface active substances. All these properties can be explained by the local hydrodynamics within the air layer and at the air-water interface, such as the local flow field depending on the surface geometry or Marangoni forces. More general, these findings are also of relevance for the development of theoretical models of slippery surfaces that rely on a fluid being in the Cassie state.

9:18AM M8.00007 Robust liquid-infused surfaces through patterned wettability \(^1\), JASON WEXLER, ABBIGAIL GROSSKOPF, MELISSA CHOW, YUANG FAN, Princeton University, IAN JACOBI, Technion - Israel Institute of Technology, HOWARD STONE, Princeton University — Liquid-infused surfaces display advantageous properties that are normally associated with conventional gas-cushioned superhydrophobic surfaces. However, the surfaces can lose their novel properties if the infused liquid drains from the surface. We explore how drainage due to gravity or due to an external flow can be prevented through the use of chemical patterning. A small area of the overall surface is chemically treated to be preferentially wetted by the external fluid rather than the infused liquid. These sacrificial regions disrupt the continuity of the infused liquid, thereby preventing the liquid from draining from the texture. If the regions are patterned with the correct periodicity, drainage can be prevented entirely. The chemical patterns are created using spray-coating or deep-UV exposure, two economical techniques that are scalable to generate large-scale failure-resistant surfaces.

9:31AM M8.00008 The shear-driven failure of liquid-infused surfaces and superhydrophobic surfaces \(^1\), YING LIU, Princeton University, JASON WEXLER, Otherlab, HOWARD STONE, Princeton University — We study experimentally the failure of liquid-infused surfaces under shear. Most of the previous work on this topic focuses on situations where the infused fluid is much more viscous than the external fluid. Here, we study the opposite limit: the viscosity of the infused fluid is much lower than that of the external fluid. Also, we study how the air-filled cavities of superhydrophobic surfaces fill with water under shear, which is another topic that is little studied as compared with pressure-driven failure. In each case we systematically vary the flow rate and characterize both transient and steady-state responses.

9:44AM M8.00009 A computational DFT study of structural transitions in textured solid-fluid interfaces \(^1\), PETR YATSYSHIN, Department of Chemical Engineering, Imperial College London, London, UK, ANDREW O. PARRY, Department of Mathematics, Imperial College London, London, UK, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, London, UK — Fluids adsorbed at walls, in capillary pores and slits, and in more exotic, sculpted geometries such as grooves and wedges can exhibit many new phase transitions, including wetting, pre-wetting, capillary-condensation and filling, compared to their bulk counterparts. As well as being of fundamental interest to the modern statistical mechanical theory of inhomogeneous fluids, these are also relevant to nanofluidics, chemical- and bioengineering. In this talk we will show using a microscopic Density Functional Theory (DFT) for fluids how novel, continuous, interfacial transitions associated with the first-order prewetting line, can occur on steps, in grooves and in wedges, that are sensitive to both the range of the intermolecular forces and interfacial fluctuation effects. These transitions compete with wetting, filling and condensation producing very rich phase diagrams even for relatively simple geometries. We will also discuss practical aspects of DFT calculations, and demonstrate how this statistical-mechanical framework is capable of yielding complex fluid structure, interfacial tensions, and regions of thermodynamic stability of various fluid configurations. As a side note, this demonstrates that DFT is an excellent tool for the investigations of complex multiphase systems.

\(^1\) We acknowledge financial support from the European Research Council via Advanced Grant No. 247031

\(^1\) GSRT of Greece via the program “Excellence,” Grant 1918
experiments and molecular dynamics simulations silicon surface using molecular dynamics simulations. The incident energy is high. To clarify the reason of this anomalous directivity, we are analyzing the interaction between water molecules and the toward a certain angle close to the specular reflection one. The directivity is usually observed when a surface is flat in an atomic scale and the experimental results indicated that the scattering distribution was close to that of the cosine scattering due to the roughness of the silicon temperature. The scattering distribution and the mean translational energy of scattered molecules in each scattering angle were obtained. The non-polar gas molecules such as rare gases, nitrogen, and oxygen. In the experiment, we employed the molecular beam method and changed approach. Owing to the strong polarity of water molecules, water molecules and surface atoms would interact intricately compared to those of the University of Tokyo — The scattering behavior of water molecules on silicon(100) surface was investigated by experimental and numerical ratchet concept. The development of nanofluidic systems provides unprecedented possibilities for the control of biology and chemistry at the molecular level with potential applications in low energy cost devices, novel medical tools, and a new generation of sensors. CNTs offer a number of attractive features for the fabrication of fluidic nanodevices including fast flow, useful electronic and thermal properties, high mechanical strength and biocompatibility. Therefore, the transport of liquids in CNTs is now of great interest in nanofluidics. Thermophoresis is the phenomenon of objects respond to it differently, inducing a motion and segregation of the objects. Using molecular dynamics simulations, we explore the possibility to design thermophoretic pumping devices fabricated of CNTs for water transport in nanoconduits. The design of the nanopumps is based on the concept of the Feynman-Smoluchowski ratchet. The tube diameters are 1-2 nm and the tube lengths (i.e. the membrane thicknesses) are 2-6 longer fragments. The retention time in the channel for longer DNA chains is significantly shorter than for small chains. This therefore migrate more strongly in the temperature gradient than long strands. They are therefore closer to the channel walls and have a lower mean velocity than longer strands. The retention time in the channel for longer DNA chains is significantly shorter than for small chains. This technique has the advantage that long strands can be processed quickly, unlike traditional agarose gel techniques which require longer times for longer fragments.

We acknowledge partial support from Fondecyt project 11130559 and Redoc udec

The University of Tokyo — The scattering behavior of water molecules on silicon(100) surface was investigated by experimental and numerical approach. Owing to the strong polarity of water molecules, water molecules and surface atoms would interact intricately compared to those of non-polar gas molecules such as rare gases, nitrogen, and oxygen. In the experiment, we employed the molecular beam method and changed the incident energy of water molecules between 35 and 130 meV, which corresponds to the energy of thermal motion of gas molecules at room temperature. The scattering distribution and the mean translational energy of scattered molecules in each scattering angle were obtained. The experimental results indicated that the scattering distribution was close to that of the cosine scattering due to the roughness of the silicon surface when the incident energy was 130 meV. In contrast, when the incident energy was 35 meV, the scattering distribution had a directivity toward a certain angle close to the specular reflection one. The directivity is usually observed when a surface is flat in an atomic scale and the incident energy is high. To clarify the reason of this anomalous directivity, we are analyzing the interaction between water molecules and the silicon surface using molecular dynamics simulations.

We acknowledge partial support from Fondecyt project 11130559 and Redoc udec
Mass transfer properties of nanoconfined fluids at solid-liquid interfaces: from atomistic simulations to continuum models, Matteo Morciano, Matteo Fasano, Politecnico di Torino, Andrea Stoldrup Poulsen, Université de Montpellier, Wageningen University, Mattheus Sibley, Loughborough University, Benjamin Goddard, The University of Edinburgh, Eliodoro Chiazzolo, Pietro Asinari, Politecnico di Torino, Serafim Kalliadasis.

At the nanoscale, traditional continuum models are not sufficient to describe fluid flow. For example, the no-slip assumption may not hold for nanoconfined fluids due to surface roughness and boundary conditions. Hence, boundary conditions should be accurately taken into account. Possible interplays between fluid velocity, shear stress, surfactant chemistry, and roughness are observed. Unlike hydrodynamics, in molecular dynamics (MD), the boundary conditions are not specified a priori. Here, mass transfer properties for a Lennard-Jones fluid confined in a nanochannel are studied. Density, stress, and velocity profiles within the fluid are evaluated with different conditions. Our results show a strong anisotropic behavior of fluid properties along the channel section. Shear rates and velocity profiles allow calculating the distribution of viscosity along the channel. We also observe that hydrophilic surfaces lead to increased viscosity. Our findings may have potential impact on the design of nanofluidic devices for either engineering or biomedical applications.


We present an empirical potential-based quasi-continuum theory (EQT) framework for single component LJ fluids confined in slit-like graphene nanochannels. This framework allows to treat the fluid at multiple length-scales, ranging from few Angstroms to macroscopic scales. The EQT potentials can be used to describe the excess free energy, potential of mean force, and density distribution of confined fluids at multiple length-scales. The EqT-CFT framework was developed for single-component LJ fluids confined in slit-like graphene nanochannels as examples for low and high friction cases. This framework provides accurate predictions of potential of mean force and density distribution of confined fluids at multiple length-scales, ranging from few Angstroms to macroscopic scales. The results were found to be in agreement with MD simulations for both low and high friction cases. The proposed model yields good quantitative agreement with the velocity profiles obtained from non-equilibrium molecular dynamics simulations. Furthermore, we demonstrate that the slip length is constant for different channel widths for a fixed thermodynamic state under the linear response regime.

A novel multi-scale approach for the simulation of flow through nanochannels, Frederike Jaeger, Alex Wray, Erich Muller, Imperial College London.

A novel method for the simulation of flow through nanochannels is proposed. We use molecular dynamics (MD) simulations to determine relations between the pressure, shear and bulk viscosities and the density, as well as the slip length for different fluid-wall combinations. These relationships are then plugged into a steady, two-dimensional continuum-scale model that allows the simulation of a compressible (Lennard-Jones) fluid through channels. No restrictive assumptions are made on the nature of the fluid and its flow behaviour (e.g., fully-developed, parabolic velocity profiles for incompressible fluids). Direct comparisons between the MD and the continuum-scale predictions for the channel flow show good agreement. A major advantage of the proposed method is its computational efficiency, which allows for complex flow geometries to be studied whilst still retaining the accuracy of MD-based simulations. Furthermore, through the use of the statistical fluidic association theory (SAFT), more complex fluids can be modelled, providing a computational framework capable of representing realistic experimental set-ups.

Dissipative particle dynamics incorporating non-Markovian effect, Ikuya Kineuchi, Yuta Yoshimoto, Shu Takagi, Department of Mechanical Engineering, The University of Tokyo.

The coarse-graining methodology of molecular simulations is of great importance to analyze large-scale, complex hydrodynamic phenomena. In the present study, we derive the equation of motion for non-Markovian dissipative particle dynamics (NDPD) by introducing the history effects of the time evolution of the system [Y. Yoshimoto et al., Phys. Rev. E 88, 043305 (2013)]. Our formulation is based on the generalized Langevin equation, which describes the motions of the centers of mass belonging to microscopic particles. The mean, friction, and fluctuating forces in the NMDPD model are directly constructed from an underlying MD system without any scaling procedure. For the validation of our formulation, we construct NMDPD models for high-density Lennard-Jones systems, in which the typical time scales of the coarse-grained particle motions and the fluctuating forces are not fully separable. The NMDPD models reproduce the temperatures, diffusion coefficients, and viscosities of the corresponding MD systems more accurately than the conventional DPD models based on a Markovian approximation. Our results suggest that the NMDPD model is a promising alternative for simulating mesoscale flows where a Markovian approximation is not valid.

Effect of variable magnetic field on nanofluid flow and heat transfer, Mohammadakazem Sadoughi, Department of Mechanical Engineering, Iowa State University Science and Technology, Ames, IA 50011, United States, Mohsen Sheikholeslami, Department of Mechanical Engineering, Babol University of Technology, Babol, Islamic Republic of Iran, Hamed Shariatmadar, School of Mechanical Engineering, College of Engineering, University of Tehran, Tehran 1439955691, Iran.

In this paper, Control Volume based Finite Element Method is applied to simulate nanofluid flow and heat transfer in presence of variable magnetic field. Magnetohydrodynamic (MHD) equations are coupled with the energy equation due to the heat transfer by means of the Boussinesq approximation. Then, the 2D non-dimensional full MHD equations in terms of stream function, temperature, magnetic field and vorticity are solved by using CVFEM. The calculations were performed for different governing parameters namely; the Rayleigh number, nanoparticle volume fraction and Hartmann number arising from MHD. Results show that Nusselt number has direct relationship with Rayleigh number, nanoparticle volume fraction while it has reverse relationship with Hartmann number. Also it can be found that enhancement in heat transport is observed.
Effect of weak rotation on the large-scale circulation in turbulent convection with a Prandtl number $Pr = 12.3$, PING WEI, GUNTER AHLERS, University of California, Santa Barbara, CA — We report measurements of large-scale circulation properties for high-Rayleigh-number convection in a rotating cylindrical sample with aspect ratio $A = D/L = 1.00$ ($D$ is the diameter and $L$ the height). The Prandtl number was $Pr = 12.3$. The measurements covered the Rayleigh-number range $2 \times 10^{10} < Ra < 4 \times 10^{11}$ and the inverse Rossby-number range $0 \leq 1/Ro \leq 1/Ro_c = 0.28$ where the LSC was present. The azimuthal orientation $\theta_L$ of the LSC circulation plane remained fixed in the frame of the rotating sample for $Ra < Ra_0 \lesssim 5 \times 10^{10}$. The sloshing motion of the LSC showed oscillations with a short time period $\tau^L$ of several tens of seconds. The temperature amplitude $< \delta >$ of the LSC increased as $1/Ro$ approached $1/Ro_c$, and decreased rapidly beyond it. For $Ra > Ra_0$, the circulation plane underwent retrograde rotation and hence caused time-periodic temperature oscillations near the side wall with a large period $\tau_L$ of hundreds of seconds. Remarkably, $\tau_L$ persisted without a discontinuity even for $1/Ro > 1/Ro_c$, where the LSC ceased to exist, indicating that vortex structures in that regime undergo the same retrograde rotation as the LSC.

Supported by NSF Grant DMR11-58514

Rayleigh- and Prandtl-number dependence of the large-scale flow-structure in weakly-rotating turbulent thermal convection, STEPHAN WEISS, Max Planck Institute f. Dynamics and Self-Organization, PING WEI, GUNTER AHLERS, University of California Santa Barbara — Turbulent thermal convection under rotation shows a remarkable variety of different flow states. The Nusselt number ($Nu$) at slow rotation rates (expressed as the dimensionless inverse Rossby number $1/Ro$, for example, is not a monotonic function of $1/Ro$. Different $1/Ro$-ranges can be observed with different slopes $\partial Nu/\partial (1/Ro)$. Some of these ranges are connected by sharp transitions where $\partial Nu/\partial (1/Ro)$ changes discontinuously. We investigate different regimes in cylindrical samples of aspect ratio $A = 1$ by measuring temperatures at the sidewall of the sample for various Prandtl numbers in the range $3 < Pr < 35$ and Rayleigh numbers in the range $10^8 < Ra < 4 \times 10^{11}$. From these measurements we deduce changes of the flow structure. We learn about the stability and dynamics of the large-scale circulation (LSC), as well as about its breakdown and the onset of vortex formation close to the top and bottom plate. We shall examine correlations between these measurements and changes in the heat transport.

This work was supported by NSF grant DMR11-58514. SW acknowledges support by the Deutsche Forschungsgemeinschaft.

Tuning transitions in rotating Rayleigh-Bénard convection, PRANAV JOSHI, RUDIE KUNNEN, HERMAN CLERcx, Technische Universiteit Eindhoven — Turbulent rotating Rayleigh-Bénard convection, depending on the system parameters, exhibits multiple flow states and transitions between them. The present experimental study aims to control the transitions between the flow regimes, and hence the system heat transfer characteristics, by introducing particles in the flow. We inject near-neutrally buoyant silver coated hollow ceramic spheres (∼100 micron diameter) and measure the system response, i.e. the Nusselt number, at different particle concentrations and rotation rates. Both for rotating and non-rotating cases, most of the particles settle on the top and bottom plates in a few hours following injection. This rapid settling may be a result of "trapping" of particles in the laminar boundary layers at the horizontal walls. These particle layers on the heat-transfer surfaces reduce their effective conductivity, and consequently, lower the heat transfer rate. We calculate the effective system parameters by estimating, and accounting for, the temperature drop across the particle layers. Preliminary analysis suggests that the thermal resistance of the particle layers may affect the flow structure and delay the transition to the "geostrophic" regime.

Financial support from Foundation for Fundamental Research on Matter.

Combined effects of a magnetic field and a helical force on the onset of a rotating Rayleigh-Bénard convection with free-free boundaries, JEAN BIO CHABI OROU, GISÈLE POMALÈGNI, Université d’Abomey-Calavi — We investigate the combined effects of rotation, magnetic field and helical force on the onset of stationary and oscillatory convection in an electrically conducting fluid layer heated from below with free-free boundary conditions. For this investigation the linear stability analysis studied by Chandrasekhar (1961) is used. We obtain the condition for the formation of a large scale structure. In (Pomalègni et al, 2014) it was shown the existence of a critical value $S_c$, of the intensity of the helical force for which the apparition of two cells at marginal stability for the oscillatory convection is obtained. Then, we have shown here how the increasing of parameter $Ta$ influences this critical value of the helical force intensity.

Energetic dynamics of a rotating horizontal convection model with wind forcing, VARVARA ZEMSKOVA, BRIAN WHITE, ALBERTO SCOTTI, University of North Carolina at Chapel Hill — We present a new test case for rotating horizontal convection, where the flow is driven by differential buoyancy forcing along a horizontal surface. This simple model is used to understand and quantify the influence of surface heating and cooling and wind stress on the Meridional Overturning Circulation. The domain is a rectangular basin with surface cooling at both ends (the poles) and surface warming in the middle (equatorial) region. To model the effect of the Antarctic Circumpolar Current, reentrant channel is placed near the Southern pole. Free-slip boundary conditions are imposed in the closed box, while zonally periodic boundary conditions are enforced in the channel. The problem is solved using a 3D DNS model based on a finite-volume AMR solver for the Boussinesq Navier-Stokes equations with rotation. The relative contributions of surface buoyancy and wind forcing and the energetic balance are analyzed at a Rayleigh number of $10^{9}$ and a relatively high aspect ratio of [5, 10, 1] in zonal, meridional and vertical directions, respectively. The overall dynamics, including large-scale overturning, baroclinic eddying, and turbulent mixing are investigated using the local Available Potential Energy framework introduced in [Scotti and White, J. Fluid Mech., 2014].

This research is part of the Blue Waters sustained-petascale computing project, supported by the NSF (awards OCI-0725070, ACI-1238993 and ACI-14-41747) and the state of Illinois.
9:18AM M10.00007 Transitions in turbulent rotating convection , HADI RAJAEI, KIM ALARDS, RUDIE KUNNEN, FEDERICO TOSCHI, HERMAN CLERCKX, Eindhoven University of Technology, FLUID DYNAMICS LAB TEAM — This study aims to explore the flow transition from one state to the other in rotating Rayleigh-Bénard convection using Lagrangian acceleration statistics. 3D particle tracking velocimetry (3D-PTV) is employed in a water-filled cylindrical tank of equal height and diameter. The measurements are performed at the center and close to the top plate at a Rayleigh number $Ra = 1.28 \times 10^9$ and Prandtl number $Pr = 6.7$ for different rotation rates. In parallel, direct numerical simulation (DNS) has been performed to provide detailed information on the boundary layers. We report the acceleration pdfs for different rotation rates and show how the transition from weakly to strongly rotating Rayleigh-Bénard affects the acceleration pdfs in the bulk and boundary layers. We observe that the shapes of the acceleration PFDs as well as the isotropy in the cell center are largely unaffected while crossing the transition point. However, acceleration pdfs at the top show a clear change at the transition point. Using acceleration pdfs and DNS data, we show that the transition between turbulent states is actually a boundary layer transition between Prandtl-Blasius type (typical of non-rotating convection) and Ekman type.

9:31AM M10.00008 Lagrangian analysis of rotating Rayleigh-Bénard turbulence , KIM ALARDS, HADI RAJAEI, RUDIE KUNNEN, FEDERICO TOSCHI, HERMAN CLERCKX, Eindhoven University of Technology — Transitions between turbulent states can occur in Rayleigh-Bénard convection, for example, due to rotation, which is known to change the flow structure and the heat transport. In this study we want to characterize these different states of turbulence using Lagrangian statistics of tracer particles. Rayleigh-Bénard convection is modeled using DNS and tracer particles that perfectly follow the flow are included. The fluid velocity and the temperature at the particle position are calculated using a linear interpolation scheme. Lagrangian statistics of 16 particles are measured in the form of velocity, acceleration and temperature pdfs for different rotation rates. The influence of rotation on the flow structure and heat transport is analyzed. Statistics obtained in the cell center and near the top and bottom plate are compared in order to investigate the influence of the boundary layers on RB convection. On top of that the results are compared with experiments, in which neutrally buoyant particles are tracked in a rotating cylindrical RB setup. A good agreement between experiments and numerics is found.

9:44AM M10.00009 Retrograde rotation of the large-scale flow in turbulent rotating Rayleigh-Bénard convection with high Rossby number1, JIN-QIANG ZHONG, HUI-MIN LI, XUE-YING WANG, Tongji University — We present measurements of the azimuthal orientation $\theta(t)$ of the large-scale circulation (LSC) for turbulent Rayleigh-Bénard convection in the presence of weak rotations $\Omega$. Linear retrograde rotations of the LSC circulating plane are observed over the entire Rossby-number range ($1 \leq Ro \leq 300$) studied. When the $Ro$ increases, the ratio of the retrograde rotation rate, $\gamma = -\langle \theta / \Omega \rangle$, remains nearly a constant 0.12 in the range of $1 \leq Ro \leq 80$ and starts to increases when $Ro > 80$. When $Ro > 300$, $\gamma$ approaches a value of 0.36 close to the prediction from previous theoretical models. In a background of linear rotations, erratic changes in $\theta(t)$ accompanied by decreasing in the LSC amplitude $\delta$ are observed. These small-$\delta$ events give rise to the increasing $\gamma$ with very high $Ro$ numbers ($80 \leq Ro \leq 300$). In this range, the diffusivity of $\theta$ is proportional to $\delta^{-2}$. Moreover, the occurrence frequency of the small-$\delta$ events, and their average duration are independent on $Ro$. We propose a model to include additional viscous damping for the LSC azimuthal motion due to turbulent viscosity and provide theoretical interpretations of the experimental results.

1Work supported by NSFC Grant No. 11202151.

9:57AM M10.00010 The Göttingen rotating turbulent Rayleigh–Bénard convection facility1, EBERHARD BODENSCHATZ, MPI for Dynamics and Self-Organization, Göttingen, DENNIS VAN GILS, Max Planck Institute for Dynamics and Self-Organization, Göttingen; Physics of Fluids, University of Twente, Enschede, XIAOZHI OUHE, MPI for Dynamics and Self-Organization, Göttingen; Department of Physics, University of California, Santa Barbara, INTERNATIONAL COLLABORATION FOR TURBULENCE RESEARCH, EUHIT COLLABORATION — This presentation will focus on the newly commissioned rotating RBC facility at the Max Planck Institute for Dynamics and Self-Organization (MPIDS). The MPIDS has a pressure vessel, called the Uboot of Göttingen, which can house different RBC cells. By pressurizing the Uboot with sulfur hexafluoride, nitrogen, or helium up to 19 bars one can obtain Rayleigh numbers spanning $10^9 < Ra < 10^{11}$, at nearly constant Prandtl number $Pr$. Recently, a rotating table was constructed that can operate outside as well as in the Uboot, on top of which the current RBC cell of aspect ratio 0.50 can be installed. The accessible parameter space is $0.02 < Ro^{-1} < 20$ for the inverse Rossby number and $10^{-8} < Ek < 10^{-3}$ for the Ekman number. At strong rotation (small $Ek$) but still turbulently convective (large $Ra$) one enters the geostrophic turbulent regime. Recent experiments involve measuring in and near this regime of which preliminary results will be shown and discussed.

1We thank the Max Planck Society, the German Science Foundation SFB 963, the NSF grant DMR11-58514, and EuHIT for generous support.

Tuesday, November 24, 2015 8:00AM - 9:57AM – Session M11 Convection and Buoyancy-Driven Flows: Numerical Studies 111 - Rudie Kunnen, Eindhoven University of Technology

8:00AM M11.00001 Transition to geostrophic convection: the role of boundary conditions , RUDIE KUNNEN, Eindhoven University of Technology, RODOLFIO OSTILLO-MÓNICO, ERWIN VAN DER POEL, University of Twente, ROBERTO VERZICCO, University of Rome “Tor Vergata”, DETLEF LOHSE, University of Twente — The so-called geostrophic regime of rapidly rotating Rayleigh–Bénard convection is dominated by rotation with strong enough thermal forcing to attain a turbulent flow. It is the appropriate regime for the description of the large-scale geophysical and astrophysical convective flows. Only very recently, numerical simulations and experiments have become able to enter into this regime with distinctly different scalings than the traditional rotation-affected regime, with many open questions remaining. We explore the transition to the geostrophic regime using direct numerical simulations of the Navier–Stokes and heat equations by varying the rotation rate (Ekman number $Ek$) at two constant values of the thermal forcing (Rayleigh number $Ra = 1 \times 10^{10}$ and $5 \times 10^{10}$) and constant Prandtl number $Pr = 1$. We focus on the differences between the application of no-slip or stress-free boundary conditions on the horizontal plates. We find the transition as changes in heat transfer, boundary-layer thickness, bulk/boundary-layer distribution of dissipation and bulk mean temperature gradient. The transition is gradual: many statistics reveal a change in scaling, but not sharp and not at exactly matching $Ek$. 
8:13AM M11.00002 Buoyancy induced modification of the law-of-the-wall in an unstably stratified turbulent channel flow. FEDERICO TOSCHI, Eindhoven University of Technology, ANDREA SCAGLIARINI, University of Rome Tor Vergata, HALLDOR EINARSSON, ARMANN GYLFAASON, School of Science and Engineering, Reykjavik University — We present results on the influence of buoyancy on the boundary layer dynamics and on mean quantities, like velocity profiles, in an unstably stratified turbulent channel flow. The study is based on direct numerical simulations where we investigated a broad range of friction Reynolds numbers and Rayleigh numbers. We primarily focused on the modification of the logarithmic law of the wall, due to buoyancy, and we provide a simple phenomenological model that is able to capture the observed deviations, in the log-law region, from the usual neutral case.

8:26AM M11.00003 Compressible convection in geophysical fluids: comparison of anelastic, anelastic liquid and full numerical simulations. SCAGLIARINI, University of Rome Tor Vergata, HALLDOR EINARSSON, ARMANN GYLFAASON, School of Science and Engineering, Reykjavik University — We primarily focused on the modification of the logarithmic law of the wall, due to buoyancy, and the anelastic liquid approximation. Various equations of state are considered, from the ideal gas equation to equations related to liquid or solid condensed matter. We are particularly interested in the total value and spatial distribution of viscous dissipation. We analyze the solutions obtained with each approximation in a wide range of dimensionless parameters and compare the domain of validity of each of them.

8:39AM M11.00004 Eddy Sensitivity to Resolution and Viscosity in Density Driven Ocean Currents. SHANON RECKINGER, Montana State University, MARK PETERSEN, Los Alamos National Laboratory, SCOTT RECKINGER, Montana State University — Density driven currents (in the ocean, known as oceanic overflows) impact global ocean circulation and affect intermediate and deep-water properties in numerous regions in the ocean. General circulation models currently rely on parameterizations for representing dense overflows due to resolution restrictions. These parameterizations rely on a detailed understanding of the mixing properties, which is enhanced by studying idealized overflows.

8:52AM M11.00005 Quantitative saltwater modeling for validation of sub-grid scale LES turbulent mixing and transport models for fire. PIETRO MAISTO, ANDRE MARSHALL, MICHAEL GOLLNER, Department of Fire Protection Engineering, University of Maryland, College Park — A quantitative understanding of turbulent mixing and transport in buoyant flows is indispensable for accurate modeling of combustion, fire dynamics and smoke transport used in both fire safety design and investigation. This study describes the turbulent mixing behavior of scaled, unconfined plumes using a quantitative saltwater modeling technique. An analysis of density difference turbulent fluctuations, captured as the collected images scale down in resolution, allows for the determination of the largest dimension over which LES averaging should be performed. This is important as LES models must assume a distribution for sub-grid scale mixing, such as the %PDF distribution. We showed that there is a loss of fidelity in resolving the flow for a cell size above 0.54D∗, where D∗ is a characteristic length scale for the plume. Such a point represents the threshold above which the fluctuations start to monotonically grow. Turbulence statistics were also analyzed in terms of span-wise intermittency and time and space correlation coefficients. An unexpected condition for the core of the plume, where a substantial amount of ambient fluid (fresh water) is found, and the mixing process under buoyant conditions were found depending on the resolution of measurements used.

9:05AM M11.00006 Effect of inlet conditions on the turbulent statistics in a buoyant jet. RAJESH KUMAR, National Institute of Technology Kurukshetra, Kurukshetra - 136119, ANUPAM DEWAN, Indian Institute of Technology Delhi, New Delhi - 110016 — Buoyant jets have been the subject of research due to their technological and environmental importance in many physical processes, such as, spread of smoke and toxic gases from fires, release of gases from volcanic eruptions and industrial stacks. The nature of the flow near the source is initially laminar which quickly changes into turbulent flow. We present large eddy simulation of a buoyant jet. In the present study a careful investigation has been done to study the influence of inlet conditions at the source on the turbulent statistics far from the source. It has been observed that the influence of the initial conditions on the second-order buoyancy terms extends further in the axial direction from the source than their influence on the time-averaged flow and second-order velocity statistics. We have studied the evolution of vortical structures in the buoyant jet. It has been shown that the generation of helical vortex rings in the vicinity of the source around a laminar core could be the reason for the larger influence of the inlet conditions on the second-order buoyancy terms as compared to the second-order velocity statistics.

9:18AM M11.00007 The Role of Convective and Diffusive Mixing in Porous Media. SHYAM SUNDER GOPALAKRISHNAN, Fluid and Plasma Dynamics Unit, Université Libre de Bruxelles, JORGE CARBALLIDO-LANDEIRA, ANNE DE WIT, Nonlinear Physical Chemistry Unit, Université Libre de Bruxelles, BERNARD KNAEPEN, Fluid and Plasma Dynamics Unit, Université Libre de Bruxelles — The classical Rayleigh–Taylor (RT) instability that triggers convective and diffusive mixing when a denser fluid lies on top of a less dense one is characterised both numerically and experimentally in an ideal two-dimensional porous media. The universal nature of the flow dynamics starting with a stable diffusive regime, that is followed by a linearly unstable regime, and eventually to a nonlinear regime is presented. Though the fundamental behaviour has been studied extensively, the roles of convective and diffusive mixing on the flow features are not yet explored. It has been a long held view that diffusive mixing is significant only during the initial stages, and once the transition has occurred, the dynamics are governed by convection. We show that this is not the case, and both convection and diffusion play an important role even during the nonlinear regime, albeit at different regions of the flow with convection dominant locally at the tip of the fingers, and balanced by diffusion in the rest of the mixing zone. This also provides a quantitative measure for the evolution of the width of the fingers. The computational findings are well supported using our experimental observations, where an excellent agreement on the flow dynamics are obtained.
Moreover, in the superfrictional regime at very large speeds. Anisotropy is quantified from the alignment of the stress and strain rate tensors, with the strain rate computed using a least-squares fit. It is shown that in certain regions there is a strong anisotropic relationship, regardless of the speed of rotation. The effective friction coefficient is examined in order to determine the phase space in which the stress anomaly is caused by a novel secondary flow—a single toroidal vortex that spans the entire granular column. The vortex differs fundamentally in its origin and manifestation from the Taylor-Couette vortices in fluids. It is driven by dilatancy, and is sustained by gravity. Our results raise the possibility of similar secondary flows arising in other flow geometries, and call for caution in the interpretation of rheological measurements for granular materials.

It is shown that in certain regions there is a strong anisotropic relationship, regardless of the speed of rotation. The effective friction coefficient is examined in order to determine the phase space in which the stress anomaly is caused by a novel secondary flow—a single toroidal vortex that spans the entire granular column. The vortex differs fundamentally in its origin and manifestation from the Taylor-Couette vortices in fluids. It is driven by dilatancy, and is sustained by gravity. Our results raise the possibility of similar secondary flows arising in other flow geometries, and call for caution in the interpretation of rheological measurements for granular materials.
8:39AM M12.00004 Dynamics of an intruder pulled slowly from a granular material

YUE ZHANG, Duke University. ABE CLARK, Yale University. ROBERT BEHRINGER, Duke University — What is the response of a granular material and an object buried in the material as the object is pulled out? To address this question, we use an experiment where the grains are 2D photoelastic disks to visualize the pull out dynamics for different circular intruders. We apply forces that are close to the minimum to initiate intruder motion. We observe the intruder motion, $z(t)$, and the disk photoelastic response. We numerically differentiate $z(t)$, to yield the intruder velocity, $v(t)$, and acceleration, $a(t)$. After transients, we find $w(t) = c \exp(bt)$, where coefficients $c$ and $b$ depend on the intruder, particularly $b$ decreases when increasing intruder size. Why does velocity depend exponentially on time, or equivalently why does acceleration linearly change with displacement? To answer this question, we compute the drag force caused by the granular disks from the acceleration of the intruder. The result shows that the drag force depends linearly on the thickness of disks above the intruder, which also changes linearly with the displacement of the intruder. However, the drag force is much bigger than the weight of particles above the intruder. Ongoing work focuses on illuminating the cause for the observed drag force.

1 We would like to acknowledge grant agencies, NSF DMR1206531 and NASA NNX15AD38G.

8:52AM M12.00005 Continuum equations for dense shallow granular flows

VISWANATHAN KUMARAN, Indian Inst of Science — Simplified equations are derived for a granular flow in the 'dense' limit where the volume fraction is close to that for dynamical arrest, and the 'shallow' limit where the stream-wise length for flow development ($L$) is large compared to the cross-stream height ($h$). In the dense limit, the equations are simplified by taking advantage of the power-law divergence of the pair distribution function $\chi$ proportional to $(\phi_{ad} - \phi)^{-\alpha}$, where $\phi$ is the volume fraction, and $\phi_{ad}$ is the volume fraction for arrested dynamics. When the height $h$ is much larger than the conduction length, the energy equation reduces to an algebraic balance between the rates of production and dissipation of energy, and the stress is proportional to the square of the strain rate (Bagnold law). The analysis reveals important differences between granular flows and the flows of Newtonian fluids. One important difference is that the Reynolds number (ratio of inertial and viscous terms) turns out to depend only on the layer height and Bagnold coefficients, and is independent of the flow velocity, because both the inertial terms in the conservation equations and the divergence of the stress depend on the square of the velocity/velocity gradients.

9:05AM M12.00006 Drag on intruder in dense granular flows

HU ZHENG, Hohai University. DUKE UNIVERSITY, JONATHAN BARES, DONG WANG, ROBERT BEHRINGER, Duke University — We perform an experimental study on an intruder dragged at a constant force in a quasi-statically cyclic-sheared granular medium. A Teflon disk is embedded in a layer of bidisperse photoelastic disks. The granular medium is contained in a horizontal square cell, which can be deformed into a parallelogram with the same area to produce simple shear. We find that the forward motion of the intruder happens at the fragile state during shear reversals, while only reversible affine motion could be found at the jammed state. There is a burst of non-affine motion for the granular particles at each shear reversal. For a range of packing fractions, the cumulative intruder displacement shows a linear increase proportional to the number of cycles of shear. To explain the behavior of intruder motion, we analyze the coordination number, density, affine and non-affine motion of disk-granular system variations as the shear strain.

1 We acknowledge support from NSF Grant No. DMR1206351, NASA Grant No. NNX15AD38G and the W.M. Keck Foundation.

9:18AM M12.00007 Steady State Erosion of Granular Particles by Shear Flow

BENJAMIN ALLEN, ARSHAD KUDROLLI, Clark University — Despite decades of scientific observation of rivers, streams and laboratory experiments the process of erosion still is not understood. Empirical fits are used to determine when erosion starts with more than an order of magnitude scatter or a shifting power law determining how much material erodes away. In order to study the many body problem of multiple particles we first need to understand the basics of a single particle eroding from a potential well in laminar flow. Using different particle densities and different distances between particles and the distances at different times and the balance of forces and torques create a prediction that a single particle will erode over a barrier of a given height as a function of shear rate and viscosity. We then create a steady state system in which to image erosion as it happens and simultaneously measure flow velocity and particle movement. Measuring particle movement allows us to determine when steady state erosion occurs and calculate the fluxes and slip velocities at the beginning of the erosion process as we transition from rolling particles to particles suspended in the fluid flow.

1 NSF Grant Number CBET 1335928

9:31AM M12.00008 DEM simulation of flow of dumbbells on a rough inclined plane

SANDIP MANDAL, DEVYAN KHAKHAR, Department of Chemical Engineering, Indian Institute of Technology Bombay, Powai, Mumbai 400076, India — The rheology of non-spherical granular materials such as food grains, sugar cubes, sand, pharmaceutical pills, among others, is not understood well. We study the flow of non-spherical dumbbells of different aspect ratios on a rough inclined plane by using soft sphere DEM simulations. The dumbbells are generated by fusing two spheres together and a linear spring dashpot model along with Coulombic friction is employed to calculate inter-particle forces. At steady state, a uni-directional shear flow is obtained which allows for a detailed study of the rheology. The effect of aspect ratio and inclination angle on mean velocity, volume fraction, shear rate, shear stress, pressure and viscosity profiles is examined. The effect of aspect ratio on probability distribution of angles, made by the major axes of the dumbbells with the flow direction, average angle and order parameter is analyzed. The dense flow rheology is well explained by Bagnold’s law and the constitutive laws of JFP model [Jop et. al., Nature 441, 727 (2006)]. The dependencies of first and second normal stress differences on aspect ratio are studied. The probability distributions of translational and rotational velocity are analyzed.

9:44AM M12.00009 The propagation and deposition process of a finite dry granular mass down a rough incline

GENG LIN LEE, FU-LING YANG, Department of mechanical engineering, National Taiwan University, Taipei, Taiwan — This work presents a theoretical analysis on the propagation and arresting process of a 2D finite granular mass in shallow configuration down a rough incline. The coherence-length constitutive model proposed by Ertas and Halsey (2002) is used to solve the bulk motion and local coherence length scale, $l(x,t)$, which characterizes internal granular clusters. Flow depth profile, $h(x,t)$, governed by an advection-diffusion equation is solved by the matched asymptotic method under shallowness and used to determine a flow front trajectory, $x_f(t)$. The solutions reveal $l(x,t) < h(x,t)$ in the front indicating the clusters can move freely and transport momentum flux in a flowing bulk. The trend of $l(x,t)$ shows monotonic growing and becomes comparable to $h(x,t)$ upstream, indicating clusters transmit basal decelerating impulse to decelerate the flow, giving rise to rear deposit. The critical location where $l(x,t) = h(x,t)$ is solved to the leading order to determine a deposition front trajectory, $x_d(t)$. Under the constraint of conserved total mass, finite run-out distance, $L_d$, and arrested time, $T_d$, are estimated and used to construct a modified front propagation model, $x_{fm}(t)$, which compares well to the experimental data reported in Pouliquen and Forterre (2002)
This material is based upon work supported by the National Science Foundation Graduate Research Fellowship under Grant No. DGE7114469.

1The support for this study comes from AFSOR, NSF, EPRI and Johns Hopkins E2SHI Seed Grant.

1This material is based upon work supported by the National Science Foundation Graduate Research Fellowship under Grant No. DGE7114469.

8:00AM M13.00001 Energy Harvesting for Micropower Applications by Flow-Induced Flutter of an Inverted Piezoelectric Flag1. Kourosh Shoele, Rajat Mittal, Johns Hopkins University — Piezoelectric flexible flags can be used to continuously generate energy for small-scale sensor used in a wide variety of applications ranging from measurement/monitoring of environmental conditions (outdoors or indoors) to in-situ tracking of wild animals. Here, we study the energy harvesting performance as well as the flow-structure interaction of an inverted piezoelectric flag. We use a coupled fluid-structure-electric solver to examine the dynamic response of the inverted flag as well as the associated vortical characteristics with different inertia and bending stiffness. Simulations indicate that large amplitude vibrations can be achieved over a large range of parameters over which lock-on between the flag flutter and the intrinsic wake shedding occurs. The effects of initial inclination of the flag to the prevailing flow as well as Reynolds number of the flow are explored, and the effect of piezoelectric material parameters on the energy harvesting performance of this flutter state is examined in detail. The maximum energy efficiency occurs when there is a match between the intrinsic timescales of flutter and the piezoelectric circuit. The simulations are used to formulate a scaling law that could be used to predict the energy harvesting performance of such devices.

1The support for this study comes from AFSOR, NSF, EPRI and Johns Hopkins E2SHI Seed Grant.

8:13AM M13.00002 A novel immersed boundary method applied to the inverted flag problem1. Andres Goza, Timothy Colonius, Caltech — This work uses a 2-D immersed boundary method to study the inverted flag problem, in which a deformable flag is pinned at the trailing edge with its leading edge free to flap. Compared with the canonical flag problem, the inverted configuration more readily undergoes large flapping behavior for a wide range of mass ratios, making it promising in the field of flow-energy harvesting. A previous study identified several flapping modes as a function of dimensionless mass and stiffness ratios (Kim, D. et. al. J. Fluid Mech, 2013). The present work investigates the role of vortex formation and wake-flag interaction on the different flapping regimes. Simulations are performed using a 2-D immersed boundary method that accurately computes surface stresses imposed on the body by the fluid boundary conditions. Unlike many immersed boundary methods that reconstruct surface stresses from the velocity field, the current method only uses information from the immersed surface, leading to a smaller algebraic system for the fluid-solid coupling. The large flapping behavior of the inverted flag problem highlights the method's ability to simulate flows around bodies undergoing large deformations.

1This material is based upon work supported by the National Science Foundation Graduate Research Fellowship under Grant No. DGE7114469.

8:26AM M13.00003 Theoretical and Experimental Comparison of Aerodynamic Characteristics for Flexible Membrane Wings with Cambered Frames 1, Andrew Wrist1, James Hubnier2. The University of Alabama — Flexible membrane wings of the MAV (micro air vehicle) can experience improved lift/drag ratios, delays in stall, and decreased time-averaged flow separation when compared to rigid wings. Previous research examined the effect of frame camber on the time-averaged shapes of membrane wings and observed that increasing frame camber results in increased aero-induced membrane camber. This study involves a more in-depth DIC (Digital Image Correlation) analysis of the previous research to increase the understanding of the time-averaged shapes for membrane wings with cambered frames and offers a theoretical comparison to the experimental results. The author performed a theoretical lifting-line analysis based on the time-averaged shape for the membrane wings to calculate lift, induced drag, and circulation. The calculations include the effects of geometric twist, aspect ratio, and effective angle-of-attack. The wings, with an aspect ratio of 2, were fabricated with silicone rubber membranes and 3D printed cambered frames differing in percent camber, maximum camber location, and thickness. The DIC images were acquired in The University of Alabama's MAV wind tunnel as tests were performed at 10 m/s (Re = 50,000). The analysis will be discussed in the presentation.

1Graduate Research Assistant
2Associate Professor

8:39AM M13.00004 Kite propulsion: single and multi-kite stability. Emmanuel Du Pontavice, Ladhyx, Ecole Polytechnique/PMMH ESPCI, Yves Parliger, Beyond the sea, David Quere, Christophe Clanet, Ladhyx, Ecole Polytechnique/PMMH ESPCI — Kite propulsion is one way to harvest wind energy. The typical force is 1 kilo Newton per square meter, which means that with kites in the range 100 to 1000 square meters, one is able to propel ships from the trailer to the tanker. The stability of the kite with no active control is however an issue that needs to be addressed in order to develop viable systems. Under certain conditions, kites tend to engage into large oscillations and eventually crash. Through wind tunnel experiments and basic mechanic modeling, we try to understand (and avoid) this instability. In order to increase the traction of kite propulsion devices, one needs to increase their surface. One way is to superpose a large number of kites. It appears that these chains of kites are much more stable than single kites. A simple physical model is developed to understand this behavior.
tube arrays reduces uncertainties in the experimental data, which usually do not account for the effect of Reynolds number. and then analyze the tube responses. Increasing Reynolds number has a strong de-stabilizing effect for rotated arrays. For in-line arrays, effect we perform simulations that vary Reynolds number between about 100 and 13,600 independent of flow velocity at fixed Scruton numbers, Navier–Stokes equations in order to accurately model turbulent flow through tightly packed tube arrays. To investigate the Reynolds number are not sufficient to understand or quantify this effect. We use a high resolution pseudo-spectral scheme to solve two-dimensional penalized vortex-sheet model. We find that the critical flutter speed is increased due to the extra damping effect of the PZT, and can also be altered by tuning the output inductance-resistance circuit. Optimal resistance and inductance are found to either maximize or minimize the flutter speed. Thus, it is of interest to investigate if piezoelectric films can be used as a structural member of an MAV wing and generate both lift and energy through passive vibrations. Both a shaker test and a wind tunnel test have been conducted to characterize and assess energy production and aerodynamic characteristics including lift, drag and efficiency. The piezoelectric film has been successful as a lifting surface and produces a measurable charge.

This work has been supported by the EDX-DGA program.

Coupled-flutter of two slender flags, MOUGEL JÉRÔME, MICHELIN SÉBASTIEN, LadHyX, Ecole Polytechnique, DOARÉ OLIVIER, UME, ENSTA — A flag in axial flow is subject to flutter instability that leads to large-amplitude flapping of the structure. When two flags are placed parallel to each other, they interact hydrodynamically leading to coupled dynamics of the system. The understanding of the possible dynamical regimes is crucial in the recent context of energy harvesting using piezoelectric flags. In this study, we consider coupled-flutter of two slender flags. Based on an extension of the famous model by Lighthill commonly called Large Amplitude Elongated Body Theory to the two-flags case, both linear and large-amplitude dynamics of such a coupled system will be presented.

This work was supported by NSF REU Site Award 1358991.

Non-Linear Aerodynamic Coupling of Piezoelectric Harvesters in Grid Turbulence, AMIR DANESH-YAZDI, Penn State Univ, Erie, OLEG GOUSHCHA, NIELL ELVIN, YIannis ANDREOPULOS, City College of New York — Experimental and analytical results relating to the extraction of fluidic energy from decaying homogeneous and isotropic turbulence using two side-by-side cantilever beams with attached piezoelectric patches are reported. Turbulence carries mechanical energy distributed over a range of temporal and spatial scales and the resulting interaction of these scales with the immersed piezoelectric beams creates a strain field in the beam which generates electric charge. Experiments are carried out in a large scale wind tunnel in which a turbulence-generating grid is used to excite the piezoelectric cantilever beams for different gap widths between the beams at various distances from the grids and for different flow velocities. Optimal resistance and inductance are found to either maximize or minimize the flutter speed. The former application is useful for the vibration control while the latter is important for energy harvesting. We also discuss the scalability of above system to the actual application in air and water.

This work has been supported by the EDX-DGA program.

Stability and scalability of piezoelectric flag, XIAOLIN WANG, SILAS ALBEN, CHENYANG LI, YIN LU YOUNG, Univ of Michigan - Ann Arbor — Piezoelectric material (PZT) has drawn enormous attention in the past decades due to its ability to convert mechanical deformation energy into electrical potential energy, and vice versa, and has been applied to energy harvesting and vibration control. However, it is important to investigate if piezoelectric films can be used as a structural member of an MAV wing and generate both lift and energy through passive vibrations. Both a shaker test and a wind tunnel test have been conducted to characterize and assess energy production and aerodynamic characteristics including lift, drag and efficiency. The piezoelectric film has been successful as a lifting surface and produces a measurable charge.

This work was supported by NSF REU Site Award 1358991.

The role of Reynolds number in the fluid-elastic instability of tube arrays, NICHOLAS KEVLAHAN, ALI GHASEMI, McMaster University — The onset of fluid-elastic instability in tube arrays is thought to depend primarily on the mean flow velocity, the Scruton number and the natural frequencies of the tubes. However, there is evidence from experiments and numerical simulations that the Reynolds number is also an important parameter, although the available data are not sufficient to understand or quantify this effect. We use a high resolution pseudo-spectral scheme to solve two-dimensional penalized Navier–Stokes equations in order to accurately model turbulent flow through tightly packed tube arrays. To investigate the Reynolds number effect we perform simulations that vary Reynolds number between about 100 and 13,600 independent of flow velocity at fixed Scruton numbers, and then analyze the tube responses. Increasing Reynolds number has a strong de-stabilizing effect for rotated arrays. For in-line arrays, although Reynolds number still affects the instability threshold, the effect is not monotonic with increasing Reynolds number. The main de-stabilizing effect of increasing Reynolds number appears to be broadening of the vortex shedding frequency spectrum. This study increases reduces uncertainties in the experimental data, which usually do not account for the effect of Reynolds number.

Tuesday, November 24, 2015 8:00AM - 10:10AM –
Session M14 Vortex Dynamics: General 202 - Nicholas Kevlahan, McMaster University

The role of Reynolds number in the fluid-elastic instability of tube arrays, NICHOLAS KEVLAHAN, ALI GHASEMI, McMaster University — The onset of fluid-elastic instability in tube arrays is thought to depend primarily on the mean flow velocity, the Scruton number and the natural frequencies of the tubes. However, there is evidence from experiments and numerical simulations that the Reynolds number is also an important parameter, although the available data are not sufficient to understand or quantify this effect. We use a high resolution pseudo-spectral scheme to solve two-dimensional penalized Navier–Stokes equations in order to accurately model turbulent flow through tightly packed tube arrays. To investigate the Reynolds number effect we perform simulations that vary Reynolds number between about 100 and 13,600 independent of flow velocity at fixed Scruton numbers, and then analyze the tube responses. Increasing Reynolds number has a strong de-stabilizing effect for rotated arrays. For in-line arrays, although Reynolds number still affects the instability threshold, the effect is not monotonic with increasing Reynolds number. The main de-stabilizing effect of increasing Reynolds number appears to be broadening of the vortex shedding frequency spectrum. This study increases reduces uncertainties in the experimental data, which usually do not account for the effect of Reynolds number.
We also employ the derivative of the LAVD, the Instantaneous Vorticity Deviation (IVD), to uncover instantaneous Eulerian vortex boundaries objectively (i.e., in a frame invariant fashion) in massively-separated flow. We employ a recently developed objective definition and extraction— We study the formation and shedding of vortices in two vortex-dominated flows around a pitching panel in order to detect coherent structures objectively measured by the Instantaneous Vorticity Deviation (IVD). We show how the LAVD and the IVD detect rotationally coherent Lagrangian and Eulerian vortices objectively in analytic flow models and numerical flow data.

Energy problems, we study a family of axisymmetric vector fields, having the structure of a pair of vortex rings with swirl, that maximize the instantaneous vorticity moment theory. It is revealed that the LEV provides positive lift whereas the starting vortex and secondary vortex provide negative lift during the whole motion. The negative lift produced by the starting vortex or secondary vortex is not trivial and cannot be ignored. Regarding the drag, the LEV reduces the total drag whereas the starting vortex, the secondary vortex increases the total drag. As the AoA increases, the drag decreases. The starting vortex increases quickly and occurs almost all the total drag for the AoA of 72. The relations between the motion of the vortical structures and the forces are also investigated. 

Three crossing speeds are examined at a Reynolds number of $Re = 2.0 \times 10^5$, and a mostly stationary response is revealed with vortex position as each encounter traverses a similar range of flow regimes, including instances of vortex pairing of the tip and incident vortices with mutual induction/attenuation, tip vortex suppression as the impingement passes inboard of the wingtip, and induced separation that precipitates an abrupt transition at the leading edge. However, an advanced upstream development of a spiraling mode instability is observed in the incident vortex with slower encounters, which is attributed to prolonged exposure of the vortex to the adverse pressure gradient on the underside of the wing that decelerates the core axial flow below known stability bounds of the vortex, and precipitates a more pronounced spiraling mode. A dynamic loading effect is also identified prior to the vortex crossing inboard of the wingtip, whereby the incident structure’s relative position at peak loading shifts outboard with higher speeds due to a strengthened tip vortex. 

We study the formation and shedding of vortices in two vortex-dominated flows around a pitching panel in order to detect coherent structures objectively (i.e., in a frame invariant fashion) in massively-separated flow. We employ a recently developed objective definition and extraction technique for rotationally coherent Lagrangian vortices. This method renders material vortex boundaries as outermost convex level surfaces of the Lagrangian-Averaged Vorticity Deviation (LAVD), i.e., the trajectory integral of the normed difference of the vorticity from its spatial mean. LAVD-based vortices turn out to be objective, i.e., invariant under time-dependent rotations and translations of the reference frame. In the limit of vanishing Rossby numbers in geostrophic flows, cyclonic LAVD vortex centers can be proven to coincide with the observed attractors for light particles. A similar result holds for heavy particles in anticyclonic LAVD vortices. We also discuss a relationship between rotationally coherent Lagrangian vortices and their instantaneous Eulerian counterparts. The latter are formed by tubular surfaces of equal material rotation rate, objectively measured by the Instantaneous Vorticity Deviation (IVD). We show how the LAVD and the IVD detect rotationally coherent Lagrangian and Eulerian vortices objectively in analytic flow models and numerical flow data.

**8:26AM M14.00003** Vortex force generation of an impulsively started wing at high angle of attack\(^1\). XUAN FU, FUXIN HONG, LIU SUYANG, QIN YANG XIAO, Shanghai Jiaotong Univ — A wing at high angle of attack (AoA) impulsively started from rest is a fundamental motion employed by insects during flight. Previous studies have almost solely focused on the lift enhancement by the leading-edge vortex (LEV). However, the influences of the starting vortex and secondary vortex on both the lift and drag generation have been less studied. In this paper, the vorticity fields for three AoAs of 45, 58.5 and 72 are obtained numerically. The roles of the LEV, starting vortex and secondary vortex in generating the lift and drag are quantitatively studied using the vorticity moment theory. It is revealed that the LEV provides positive lift whereas the starting vortex and secondary vortex provide negative lift during the whole motion. The negative lift produced by the starting vortex or secondary vortex is not trivial and cannot be ignored. Regarding the drag, the LEV reduces the total drag whereas the starting vortex, the secondary vortex increases the total drag. As the AoA increases, the drag decreases. The starting vortex increases quickly and occurs almost all the total drag for the AoA of 72. The relations between the motion of the vortical structures and the forces are also investigated. 

\(^1\)Financial support from the State Key Development Program of Basic Research of China (2014CB744802) is gratefully acknowledged.

**8:39AM M14.00004** Transient Crossings of a NACA0012 Wing into a Streamwise-Oriented Vortex. DANIEL GARMANN, MIGUEL VISBAL, Air Force Resch Lab - WPABF — High-fidelity numerical simulations are conducted to examine the unsteady interactions of a finite NACA0012 wing maneuvering into a streamwise-oriented vortex as a representative problem of wake encounters. Three crossing speeds are examined at a Reynolds number of $Re = 2.0 \times 10^5$, and a mostly stationary response is revealed with vortex position as each encounter traverses a similar range of flow regimes, including instances of vortex pairing of the tip and incident vortices with mutual induction/attenuation, tip vortex suppression as the impingement passes inboard of the wingtip, and induced separation that precipitates an abrupt transition at the leading edge. However, an advanced upstream development of a spiraling mode instability is observed in the incident vortex with slower encounters, which is attributed to prolonged exposure of the vortex to the adverse pressure gradient on the underside of the wing that decelerates the core axial flow below known stability bounds of the vortex, and precipitates a more pronounced spiraling mode. A dynamic loading effect is also identified prior to the vortex crossing inboard of the wingtip, whereby the incident structure’s relative position at peak loading shifts outboard with higher speeds due to a strengthened tip vortex. 

**8:52AM M14.00005** Pressure Evolution in the Shear Layer of Vortex Rings as an Indicator of Pinch-Off. KRISTY SCHLUETER, Caltech, JOHN DABIRI, Stanford University — Vortex development and shedding occurs in a variety of biological settings, but the physical mechanisms driving this process are poorly understood. In a seminal 1998 paper, Gharib et al. linked the shedding of vortex rings to the integrated velocity of a piston pushed through a hollow cylinder, a metric referred to as formation number. However, the absence of a piston/cylinder configuration in most biological settings makes formation number a confusing metric by which to develop generalized principles of vortex ring pinch-off. In this study, a recently developed algorithm was used to generate pressure fields from PIV data of vortex ring development and pinch-off. The pressure evolution in the shear layer feeding the vortex ring was examined in detail. The formation time at the occurrence of a local maximum in the pressure in the shear layer was found to be a strong indicator of vortex ring pinch-off. It is hypothesized that a pressure maximum separates fluid that becomes a part of the leading vortex ring from fluid that ends up in the trailing jet. By focusing on the pressure evolution in the shear layer, instead of the formation number, which can be difficult to measure or ambiguous to define for biological flows, generalizations to other vortex shedding flows are possible. 

**9:05AM M14.00006** Objective detection of vortices in massively-separated flow\(^2\). YANGZI HUANG, Syracuse Univ, ALIREZA HADJIGHASEM, ETH Zurich, MELISSA GREEN, Syracuse Univ, GEORGE HALLER, ETH Zurich — We study the formation and shedding of vortices in two vortex-dominated flows around a pitching panel in order to detect coherent structures objectively (i.e., in a frame invariant fashion) in massively-separated flow. We employ a recently developed objective definition and extraction technique for rotationally coherent Lagrangian vortices. This method renders material vortex boundaries as outermost convex level surfaces of the Lagrangian-Averaged Vorticity Deviation (LAVD), i.e., the trajectory integral of the normed deviation of the vorticity from its spatial mean. We also employ the derivative of the LAVD, the Instantaneous Vorticity Deviation (IVD), to uncover instantaneous Eulerian vortex boundaries in an objective fashion. These Eulerian vortex boundaries, therefore, remain the same in all possible rotating and translating unsteady frames. The multiple methods we use identify and track both leading edge and trailing edge vortices as they develop and shrink. This helps in describing the relationship between the vortex dynamics and the loss of lift during dynamic stall on a 2D flat plate undergoing a 45 degree pitch-up maneuver.

\(^2\)Dr. Jeff Eldredge and his research group at UCLA are gratefully acknowledged for sharing the database of simulation results for the current research. This work was supported by the Air Force Office of Scientific Research under AFOSR Award No. FA9550-14-1-0007.

**9:18AM M14.00007** On the growth of enstrophy in axisymmetric 3D Euler flows with swirl. DIEGO AYALA, CHARLES DOERING, University of Michigan — By numerically solving suitable PDE-constrained optimization problems, we study a family of axisymmetric vector fields, having the structure of a pair of vortex rings with swirl, that maximize the instantaneous production of enstrophy in the context of 3-dimensional (3D) incompressible Euler flows. The axisymmetric fields are parametrized by their energy $K$, enstrophy $E$, and helicity $H$. The imposed symmetry is justified by the results from the seminal work of Doering & Lu (2008), recently confirmed independently by Ayala & Protas (2015), where highly localized pairs of colliding vortex rings were found to be instantaneously optimal for enstrophy production in 3D Navier-Stokes flows. The axisymmetry allows for an exhaustive exploration of the parameter space ($K, E, H$), as the 3D problem is effectively reduced to a 2-dimensional system of partial differential equations for the modified azimuthal vorticity and the azimuthal circulation density, with the corresponding reduction in computational complexity. Possible connections between these optimal axisymmetric fields with swirl and the "blow-up" problem are discussed. 

Financial support from the State Key Development Program of Basic Research of China (2014CB744802) is gratefully acknowledged.
9:31AM M14.00008 Darwinian drift: Effects of Wake Vortices and Multiple Obstacles, SERGEI MELKOUNIAN, BARTOSZ PROTAS, McMaster University — When a body passes through an unbounded fluid, it induces a net displacement of fluid particles. The difference between the initial and final positions of a fluid particle is defined as the Darwinian drift and plays an important role in the characterization of the stirring occurring in multiphase flows and in the context of biogenic mixing. Traditional studies of drift have mainly focused on single obstacles moving in a potential flow. In the present investigation we consider the effect of wake vorticity, represented by a pair of Föppl point vortices, and the combined effect of multiple obstacles. The drift in various configurations is determined using methods of complex analysis and careful numerical computations. It is demonstrated that, while the total drift increases with the size of the wake for large vortex strengths, it is actually decreased for small circulation values. We also discuss how the interaction of two obstacles affects the drift in comparison to the case of two isolated obstacles. In particular, we identify the lower and upper bound on the drift due to two identical cylinders. In certain cases our results are supported by asymptotic analysis. A physical explanation of the observed affects is offered in terms of the trajectories of individual particles.

9:44AM M14.00009 Momentum transport in the wake of a finite-length thin flat plate1. ARMAN HEMMATI, DAVID H. WOOD, ROBERT J. MARTINUZZI, University of Calgary — A comparison of the wakes of thin flat plates with aspect ratios (AR) 1.0, 1.6, 2.0 and 3.2, normal to a uniform stream, are conducted based on Direct Numerical Simulations (DNS) at Re=1200. Typical anti-symmetric Karman shedding of high AR plates, AR>2.0, is initiated by detachments at the plate corners. Shear layer detachment on the longer edges triggers shedding from the shorter edges. Thus, there is only a single shedding frequency detected in the wake. At lower AR, however, an interaction between adjacent shear layers occurs prior to detachment, which elongates the base vortex, i.e. from 1.56H for AR=3.2 to 2.69H for AR=1.6. This change of shedding mechanism has significant impact on wake structures and instantaneous pressure loads. The dominant shear layers on the longer sides appear to maintain the Karman shedding at higher AR. Karman shedding is intermittently interrupted for lower AR plates due to shear layer interactions, which increases the turbulence kinetic energy, production and dissipation rates and Reynolds stresses. To better understand dependence of the wake topology on AR, mean and fluctuating flow variables are evaluated at various locations along the chord. Moreover, comparisons to wakes of finite-height cylinders and circular plates are considered.

This work is supported by AITF and NSERC fellowship grants.

9:57AM M14.00010 Rolling up of Large-scale Laminar Vortex Ring from Synthetic Jet Impinging onto a Wall1. YANG XU, CHONG PAN2, JINJUN WANG, Fluid Mechanics Key Laboratory of Education Ministry, BeiHang University, FLOW CONTROL LAB TEAM — Vortex ring impinging onto a wall exhibits a wide range of interesting behaviors. The present work devotes to an experimental investigation of a series of small-scale vortex rings impinging onto a wall. These laminar vortex rings were generated by a piston-cylinder driven synthetic jet in a water tank. Laser Induced Fluorescence (LIF) and Particle Image Velocimetry (PIV) were used for flow visualization/quantification. A special scenario of vortical dynamic was found for the first time: a large-scale laminar vortex ring is formed above the wall, on the outboard side of the jet. This large-scale structure is stable in topology pattern, and continuously grows in strength and size along time, thus dominating dynamics of near wall flow. To quantify its spatial/temporal characteristics, Finite-Time Lyapunov Exponent (FTLE) fields were calculated from PIV velocity fields. It is shown that the flow pattern revealed by FTLE fields is similar to the visualization. The size of this large-scale vortex ring can be up to one-order larger than the jet vortices, and its rolling-up speed and entrainment strength was correlated to constant vorticity flux issued from the jet.

1This work was supported by the National Natural Science Foundation of China (Grants No.11202015 and 11327202)
2Corresponding author

Tuesday, November 24, 2015 8:00AM - 9:57AM –
Session M15 Vortex Dynamics: Applications

8:00AM M15.00001 Helicity Annihilation in Trefoil Reconnection: Simulations, ROBERT M. KERR, University of Warwick — The simulated evolution and self-reconnection of a perturbed trefoil vortex knot is compared to the Scheeler et al, PNAS 111 (2014) experiment. To have a single initial reconnection, as in the experiments, the trefoil is perturbed by 4 weak vortex rings. Visualizations show that the simulations and experiments undergo similar topological changes. Quantitative comparisons using the helicity and global topological number show that both are preserved for a long period before reconnection begins, as in the experiments. Unlike the experiments, once reconnection begins, a significant fraction of the helicity is dissipated and the global topological number changes by a discrete amount in a fixed time. Helicity spectra and physical space correlations show that the change in helicity is associated with the appearance of negative helicity at lower wavenumbers and in the outer regions of the trefoil. Furthermore, using a range of Reynolds numbers, with the highest comparable to the experiments, it is demonstrated that a Reynolds number independent fraction of the initial helicity is dissipated in a finite time. This observation does not violate any current mathematics restricting the strong growth of Navier-Stokes norms as the viscosity goes to zero due to the structure of the trefoil.

8:13AM M15.00002 3D characterization of leading-edge vortex formation and growth1. KYOHEI ONOUE, KENNETH BREUER, Brown University — We examine the vorticity transport mechanisms responsible for regulating the stability and strength of the leading-edge vortex (LEV) on rapidly pitching plates with different planforms (swept vs. rectangular) in a uniform airflow. All experiments are carried out using a cyber-physical experimental setup (Onoue et al. 2015, JFS vol. 55) and synchronized 3D PIV measurements. In the case of a swept wing, two distinct regions of intense spanwise flow are observed around the LEV centroid—a feature conspicuously absent on a rectangular pitching plate. The interaction between these spanwise flows and the LEV core seems to play a role in prolonging the LEV residence time at the cost of the vortex circulation growth rate and magnitude. Detailed control volume analysis is performed to elucidate the flow physics at work.

1This research is funded by the Air Force Office of Scientific Research (AFOSR)
8:26AM M15.00003 The Flow Dynamics of the Garden-Hose Instability  
FANGFANG XIE, XIAONING ZHENG, MICHAEL TRIANTAFYLLOU, Massachusetts Institute of Technology, YIANNIS CONSTANTINIDES, Chevron Energy Technology Company, GEORGE KARNAIYAKIS, Brown University — We present for first time full simulations of flow-structure interactions in a flexible pipe conveying incompressible fluid. We show that the Reynolds number plays a significant role in the onset of flutter in a fluid-conveying pipe under similar boundary conditions as for the classic garden-hose problem. We investigate the complex interaction between structural and fluid dynamics and obtain a phase diagram of dynamic transition between states as a function of two non-dimensional parameters, the fluid-tension parameter, and the Reynolds number. We observe that the precise flow patterns inside the pipe determine the type of induced motion. For unsteady flow, symmetry along one direction leads to in-plane motion whereas breaking of the flow symmetry results in out-of-plane motion. Above a critical Reynolds number, as the pipe vibrates, complex flow patterns result as there is continuous generation of new vorticity due to pipe wall acceleration, which is subsequently shed in the confined space of the pipe interior.

8:39AM M15.00004 Flow development over low aspect ratio cantilevered circular cylinders in the laminar shedding regime  
CHRIS MORTON, MOHAMMAD SAEEDI, ROBERT MARTINUZZI, University of Calgary — The flow development over a cantilevered circular cylinder of aspect ratio 4 at Re = 300 has been investigated numerically by employing a laminar flow solution to the Navier-Stokes equations. The results show that two distinct wake modulation frequencies are detectable downstream of the cylinder, differing from higher Reynolds number turbulent flow cases where only one dominant frequency is present. In particular, there is a low frequency modulation with a well-defined narrow-band peak (f_m), and a high frequency contribution from the shedding of vortices (f_s). The fluctuating loading on the cylinder in the streamwise direction is tightly coupled with the low frequency modulation, while the transverse direction forces show only a weak correlation with the vortex shedding frequency. Coherent flow structures have been analyzed using proper orthogonal decomposition (POD) to provide insight into the nature of vortex formation and associated coupling with the detected low frequency modulation. The temporal coefficients obtained from the POD analysis have been used to construct a low-order model for the investigation of the overall flow development. While the high frequency component is known to be related to the formation and shedding of vortices, the low frequency component is shown to be associated with a modulation in upwash and downwash intensity.

YINGZHEN LIU, SHAOFEI WANG, School of Mechanical Engineering, Shanghai Jiao Tong University, China — Influence of the unique surface variation of the harbor seal vibrisa on its vortex-induced vibration was extensively investigated in a low-speed wind tunnel. Toward this end, a scaled-up model of the harbor-vibrisa-shaped cylinder was employed for measurement, while a circular cylinder sharing the same hydraulic diameter, mass ratio and frequency ratio was used as baseline case. Two configurations with and without the approaching Karman-vortex street were respectively tested for both cylinders at various free stream flow speeds. Here, the Karman-vortex street was generated by placing a circular cylinder far upstream. A laser displacement sensor having a high time-spatial resolution was used to capture the cross-stream displacement of the cylinders. The fluctuating pressure distribution on the surface and the wake flow pattern were captured by microphone array and the planar Particle Image Velocimetry, respectively.

1 National Natural Science Foundation of China (grant nos. 51176108, and 11372189).

9:05AM M15.00006 Drag and Strouhal number measurements for porous circular cylinders  
ANUP KANALE, PRABU SELLAPPAN, MITUL LUHAR, University of Southern California — Flow past solid bluff bodies has been studied extensively, and both experimental and computational results are well documented. However, there is limited data available for flows past porous bluff bodies, in spite of their abundance in nature. As an effort in this direction, we study the wake behind porous circular cylinders via water channel experiments employing particle image velocimetry (PIV). The experiments systematically test the effect of three dimensionless parameters: Re_p, the Reynolds number based on pore size, Re_D, the Reynolds number based on cylinder diameter, and φ the porosity of the sleeve. Specifically the PIV data are used to estimate the drag coefficient and Strouhal number for 600 ≤ Re_p ≤ 5000, 0 ≤ φ ≤ 80, and 0.33 ≤ φ ≤ 0.75. The results obtained are compared with solid cylinders to identify the effect of cylinder permeability on flow characteristics.

9:18AM M15.00007 A thin-walled Taylor column surrounding a bathtub vortex in rotating tank.  
CHIN-CHOU CHU, KUAN-RUEI LAI, YIN-CHUNG CHEN, CHIEN-CHENG CHANG, Institute of Applied Mechanics, National Taiwan University, VORTEX DYNAMICS TEAM — Numerical simulations and laboratory experiments were jointly conducted to investigate a bathtub vortex under the influence of a protruding cylinder in a rotating tank. The flow pattern depends on Rossby number (Ro=ω/H/R), Ekman number (Ek = ν/R²), and height ratio, h/H, where R is the radius of the cylinder, f the Coriolis parameter, ν the kinematic viscosity of the fluid, h the vertical length of the cylinder and H the height of the tank. Steady-state solutions obtained by numerically solving the Navier-Stokes equations in the rotating frame are shown to have good agreements with flow visualizations measurements. The bathtub vortex exhibits an interesting two-cellled pattern employing with an inner Ekman pumping and an outer up-drafting motion. The two regions of up-drafting motion are separated by a notable finite-thickness structure, identified as thin-walled Taylor column. The Taylor column sets a barrier to the fluid flow that flows into the inner region only through the narrow gaps, one above the Taylor column and one beneath it. Moreover, the height and thickness of the thin-walled Taylor column on angular velocity ratio of cylinder to background rotation (φ/Ω), ranging from -8/3 to 8/3 are also discussed.

1 Supported by Ministry of Science and Technology, TAIWAN ROC, under contract no’s 102-2221-E-002-068-MY3 & 103C-4514-1

9:31AM M15.00008 Effects of Canard on the Flowfield over a Wing  
ARASH NAYEBZADEH, HANIEH TABKHI, University of Central Florida — Surface and flowfield pressure measurements have been done over delta wing/canard configuration in a variety of canard vertical and horizontal locations and angles of attack. The experimental model consisted of wing, canard and a body to accommodate pressure tubing and canard rotation mechanism. All the tests have been performed at subsonic velocities and the effect of canard were analyzed through comparison between surface and flowfield pressure distributions. It was found that vortex flow pattern over the wing is dominated mainly by canard vertical position and in some cases, by merging of canard and wing vortices. In addition, the pressure loss induced by canard vortex on the wing surface moves the wing vortex toward the leading edge. In the mid canard configuration, canard and wing vortices merge at x/c greater than 0.5 and as a result of this phenomenon, abrupt pressure loss induces more stable vortex flow over the wing. It is also shown that canard plays a vital role in vortex break down over the wing.
been harnessed for the self-assembly of ordered arrays of metallic nanoparticles (ACS App. Mat. and Int., 2014, 6, 5835).

...up, and the spatial organization of the resulting drops. We compare our results to recent experiments, where this instability mechanism has...

Institute of Technology — The nanoscale interaction between liquid and solid molecules underlies fundamental phenomena for systems involving...

In our previous coherent-structure theories [1,2], we showed that bound states play a crucial role in the dynamics of film flows. In this study, we present a rigorous analysis of other dynamic states emerging when pulses are sufficiently close to each other, namely oscillatory states. We show that the oscillatory dynamics is associated with a peculiar object, the so-called resonance pole, which may give rise to either self-sustained or damped oscillations, something that largely depends on the particular values of the system parameters and the initial pulse separation length. We find excellent agreement between analytical and numerical work.


Interfacial instability of thin liquid films at the walls of a parallel-plate channel, sheared by pressure-driven gas flow. MIKLÓS VÉCESI, MATHIAS DIETZEL, STEFFEN HARDT. Technische Universität Darmstadt — Gas flow between liquid films is a commonly used model system for flows in the respiratory system and is also present during flow boiling in microchannel. The emergence of long-wavelength interfacial instabilities due to viscous stresses is a well-known property of these systems. We show that its description is often reducible to two coupled partial differential equations. Thus the characteristic quantities, such as the most unstable wavelength and the marginally stable wavenumber, can be obtained in a straightforward manner from the linear stability analysis. The analysis of the weakly nonlinear equations shows that if the material properties of the liquid films and their undisturbed thicknesses are identical, their interfaces should only be destabilized by the inertial forces. Moreover, for this configuration the emerging patterns on the two interfaces are found to be identical in the long-time limit. A different setup, where the liquid films have identical material properties, but their undisturbed thicknesses differ, is studied numerically. The results show that even for this configuration the interfacial deformations of the two films remain closely correlated for a broad range of parameters.

Direct Numerical Simulation of Nanofilm Instability Driven by Liquid/Solid Interactions. KYLE MAHADY, University of Tennessee, SHAHRIAR AFKHAMI, LOU KONDIC, New Jersey Institute of Technology — The nanoscale interaction between liquid and solid molecules underlies fundamental phenomena for systems involving liquids on surfaces. In addition to giving rise to the contact angle of drops, this interaction drives the spontaneous rupture of nanofilms. We study this process by means of direct simulation of the Navier-Stokes equations using the Volume of Fluid interface tracking method. Our numerical method simulates the liquid/solid interaction, and permits the study of the film rupture process with inertial effects and arbitrarily large contact angles, in both two and three dimensions. We focus in particular on the evolution of length scales in a perturbed film as it breaks up, and the spatial organization of the resulting drops. We compare our results to recent experiments, where this instability mechanism has been harnessed for the self-assembly of ordered arrays of metallic nanoparticles (ACS App. Mat. and Int., 2014, 6, 5835).
use of empirical correlations. This work are compared to available experimental data and numerical solutions based on approximate, one-dimensional models relying on the velocity, and slug formation frequency, are determined for various gas and liquid superficial velocities for a given pipe geometry. The results of the present work, the evolution of oil-water-air three-phase slug flow in a horizontal cylindrical pipe is investigated using two-dimensional and attention in the literature both experimentally and computationally but there has been very little work carried out on three-phase slugging. In velocity; this can potentially cause structural damage, particularly at pipe bends and junctions. Two-phase slug flows have received considerable

common flow regimes in pipelines is that of slug flow: slug bodies corresponding to alternating blocks of aerated liquid which bridge the 1

flows in horizontal pipes 1

Calcite and dolomite, whose composition evolution is part of the full pore problem. During oscillatory flow conditions, concentration levels of gaseous CO2 concentration, temperature, fluid flow and hydrochemistry modify the pore radius in space and time. Substrates are composed of affect the local chemical equilibrium of the aquifer, and we consider a long-wave model of this process for a single axisymmetric pore where to the injected water to compensate potential precipitates of carbonates and to prevent structural changes to the aquifer. Both of these effects

LFMI EPFL, Lausanne, Switzerland, BENOIT SCHEID, TIPS ULB, Bruxelles, Belgium — A liquid film flowing above an inclined plane can be unstable due to inertial effects: this instability, first studied by Kapitza in 1948, is always of convective nature. When the plane is reversed, there is also a destabilizing effect of gravity and the system exhibits a transition from convective to absolute instability as shown recently by Brun and coworkers using a lubrication equation. In addition, the latter authors observed experimentally that this transition corresponds roughly to the limit of dripping. We investigate further this idea numerically by using a more sophisticated set of equations including inerita and second-order viscous terms (Ruyer-Quil-Manneville model). When getting closer to horizontality, 2D stationary wave profiles computed by continuation exhibit a transition from Kapitza-like waves to more symmetric drop profiles riding on a thinner substrate. Their 2D/3D secondary modes of instability are analyzed using a Floquet linear stability analysis and compared with time simulations of the equations model.

Role of slip on the Yih-Marangoni instability in an interface dominated channel flow, GEETANJALI CHATTOPADHYAY, R USHA, IIT Madras, India — A linear stability analysis of Poiseuille flow of two immiscible fluids of different viscosities and densities in a slippery channel, in the presence of an insoluble surfactant at the interface is examined, within the framework of Orr-Sommerfeld system. The equations governing the flow system are solved numerically by a Chebyshev collocation method for a wide range of dimensionless parameters describing the flow system. The effects of slip on the neutral stability boundaries for the interface modes in the presence/absence of an insoluble surface-active agent are examined for different thickness ratios of the two layers. Slip conditions at the wall show a promise for control of the Yih-Marangoni instability of the corresponding flow system in a rigid channel. The influence of the parameters on the critical Reynolds number for the shear mode is assessed. The interaction between the two modes under the influence of different parameters displays interesting scenarios such as coalescence of modes. The study reveals that it is possible to control instabilities in interface dominated rigid channel flows by designing the walls of the channel as hydrophobic/rough/porous or undulated surfaces as these can be modeled as one with slip at the substrates.

Thin film instabilities on heated substrates: conjugate heat transfer, MICHAEL DALLASTON, Department of Chemical Engineering, Imperial College London, DMITRI TSELUIKO, Department of Mathematical Sciences, Loughborough University, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London — Heat transported from a surface by a thin coating film of liquid is greatly affected by instabilities on the free surface of the film. If the solid substrate is heated above the ambient temperature, the hydrodynamic instability of the flow at sufficiently large Reynolds number is exacerbated by Marangoni stresses that result due to the temperature gradient in the fluid. Most studies of this phenomenon assume constant temperature or heat flux at the wall. Here we discuss the less-studied but more realistic situation in which the heat flow within the liquid film is coupled to conduction within the solid substrate, which has a complicated effect on the stability of the free surface. Analytical progress is made possible by linear stability analysis and low-dimensional nonlinear evolution equations derived using a weighted residual method.

On Pore Dynamics and Calcite Solubility in Carbonaceous Aquifers used in Energy Storage Applications, BS TILLEY, DS BRADY, Worcester Polytechnic Institute, M UECKERT, T BAUMANN, Technische Universitaet Muenchen — Geothermal energy harvesting applications use deep groundwater aquifers to store harvested energy. The impact of this additional energy to the aquifer chemistry is crucial for long-term operation. Gaseous CO2 is added to the injected water to compensate potential precipitates of carbonates and to prevent structural changes to the aquifer. Both of these effects affect the local chemical equilibrium of the aquifer, and we consider a long-wave model of this process for a single axisymmetric pore where gaseous CO2 concentration, temperature, fluid flow and hydrochemistry modify the pore radius in space and time. Substrates are composed of calcite and dolomite, whose composition evolution is part of the full pore problem. During oscillatory flow conditions, concentration levels of the dissolved species can be sufficient to overcome elevated temperature levels and promote pore closure for sufficiently thin pores. We identify the conditions under which this pore closure takes place.

Support from the Bavarian State Ministry for the Economy is acknowledged.

Tuesday, November 24, 2015 8:00AM - 10:10AM –
Session M17 Flow Instability: Multiphase Flow 205 - Omar Matar, Imperial College London

Three-dimensional numerical simulations of three-phase slug flows in horizontal pipes, YAN WANG, JUNFENG YANG, OMAR MATAR, Imperial College London — One of the most common flow regimes in pipelines is that of slug flow: slug bodies corresponding to alternating blocks of aerated liquid which bridge the pipe, and which travel at velocities that exceed the mixture superficial velocity; this can potentially cause structural damage, particularly at pipe bends and junctions. Two-phase slug flows have received considerable attention in the literature both experimentally and computationally but there has been very little work carried out on three-phase slugging. In the present work, the evolution of oil-water-air three-phase slug flow in a horizontal cylindrical pipe is investigated using two-dimensional and three-dimensional computational fluid dynamics simulations. The parameters characterizing three-phase slug flow, e.g. slug length, propagation velocity and frequency are determined for a gas flow rate which incurs a critical velocity and liquids at a given ratio of water-to-air. The results of this work are compared to available experimental data and numerical solutions based on approximate, one-dimensional models relying on the use of empirical correlations.

1EPSRC Programme Grant, MEMPHIS, EP/K0039761/1
8:13AM M17.00002 The stability of Taylor bubbles in large-diameter tubes: Linear theory1, HABIB ABUBAKAR, OMAR MATAR, Imperial College London — Taylor bubbles are a characteristic feature of the slug flow regime in gas-liquid pipe flows. With increasing pipe diameter, previous experimental observations have shown that at sufficiently large diameter (> 0.1 m), the slug flow regime, and hence Taylor bubbles, are not observed in gas-liquid flows in vertical pipes. Numerical simulations of a Taylor bubble rising in a quiescent liquid (see companion talk at this APS/DFD conference) have also shown that the wake of Taylor bubbles rising in a riser of such sizes is turbulent and has great impact on the stability of the subsequent, trailing bubbles. In view of these observations, a linear stability analysis is carried out to establish the stability conditions for a Taylor bubble rising in a turbulent flowing liquid. The stability of an axisymmetric Taylor bubble to a small-amplitude, three-dimensional, perturbation is studied and the dimensionless flow parameters of the liquid investigated include the Froude number, the inverse viscosity number, and the Eotvos numbers.

1EPSRC Programme Grant, MEMPHIS, EP/K0039761/1

8:26AM M17.00003 The stability of Taylor bubbles in large-diameter tubes: direct numerical simulations1, AMANJALOT DHANJAL, MAYA SARAVAN-BUTLER, SYDNEY SMITH, JUNFENG YANG, OMAR MATAR, Imperial College London — Slug flow corresponds to intermittent Taylor bubbles and liquid slugs, and is widely observed in the oil-and-gas industry. The fluctuating flow rate caused by Taylor bubbles is problematical; thus, the destabilisation of this regime would be beneficial. To gain better understanding of this regime in vertical tubes, three-dimensional CFD simulations of Taylor air bubble rise in initially stagnant water and progressively larger diameter tubes, are carried out. Tubes with diameters in the range of 0.032m-0.290m and a height of 1000 and kinematic viscosity ratio ≈ 60. We use code BLUE, a three-dimensional, two-phase, high performance, parallel numerical code based on a hybrid Front-Tracking/Level Set algorithm for Lagrangian tracking of arbitrarily deformable phase interfaces and a precise treatment of surface tension forces. The parallelization of the code is based on the technique of domain decomposition where the velocity field is solved by a parallel GMres method for the viscous terms and the pressure by a parallel multi-grid/GMRes method. Communication is handled by MPI message passing procedures. The interface method is also parallelized and defines the interface both by a discontinuous density field as well as by a triangular Lagrangian mesh and allows the interface to undergo large deformations including the rupture and/or coalescence of interfaces.

1EPSRC Programme Grant, MEMPHIS, EP/K0039761/1

8:39AM M17.00004 Parallel direct numerical simulation of three-dimensional spray formation1, JALEL CHERGUI, DAMIR JURIC, LIMSI-CNRS, SEUNGWON SHIN, Hongik University, Republic of Korea, LYES KAHOUADJI, OMAR MATAR, Imperial College London — We present numerical results for the breakup mechanism of a liquid jet surrounded by a fast coaxial flow of air with density ratio (water/air) ∼ 1000 and kinematic viscosity ratio ∼ 60. We code BLUE, a three-dimensional, two-phase, high performance, parallel numerical code based on a hybrid Front-Tracking/Level Set algorithm for Lagrangian tracking of arbitrarily deformable phase interfaces. The parallelization of the code is based on the technique of domain decomposition where the velocity field is solved by a parallel GMres method for the viscous terms and the pressure by a parallel multi-grid/GMRes method. Communication is handled by MPI message passing procedures. The interface method is also parallelized and defines the interface both by a discontinuous density field as well as by a triangular Lagrangian mesh and allows the interface to undergo large deformations including the rupture and/or coalescence of interfaces.

1EPSRC Programme Grant, MEMPHIS, EP/K0039761/1

8:52AM M17.00005 Stability of a liquid jet in a weak crossflow, GHOBAD AMINI, MEHDI JADIDI, ALI DOLATABADI, Concordia University — The atomization of liquid jets in crossflow is a critical process in numerous engineering systems including fuel injection and thermal spray. In an effort to elucidate the primary breakup step, a theoretical model for three-dimensional linear stability of a viscous liquid jet injected in a weak gaseous cross flow is developed. Focusing on the early stages of the jet evolution, the problem is formulated for an oblique incidence of gas flow to the liquid jet. In the context of Kelvin-Helmholtz and Rayleigh-Taylor instabilities, a characteristic equation accounting for the growth of columnar and azimuthal waves is obtained and the most dominant wavelength and the corresponding growth rates are calculated. Symmetric and asymmetric modes of liquid jet disturbance are investigated for a wide range of viscous, surface tension, and aerodynamic force ratios. The predicted results for asymptotic cases of coflow and crossflow are examined against the experimental observations available in the literature.

9:05AM M17.00006 Drop size selection in axially heated co-axial fiber capillary instability1, SAVIZ MOWLAVI, Department of Mechanical Engineering, Massachusetts Institute of Technology, 77 Massachusetts Ave., Cambridge, MA 02139, PIERRE-THOMAS BRUN, Department of Mathematics, Massachusetts Institute of Technology, Cambridge, Massachusetts 02139, USA, FRANCOIS GALLAIRE, Laboratory of Fluid Mechanics and Instabilities, STI, EPFL, Lausanne, Switzerland — We analyze the space size selection mechanism in silicon-in-silica sphere formation through the application of an external axial thermal gradient to a co-axial silicon-in-silica fiber (Gumennik et al., Nature Com., 2013). We first apply a convective/absolute stability analysis to the in-fibre capillary instability governing the sphere formation and demonstrate that the resulting wavelength selection predicts a finite but still too large wavelength. A global stability analysis is then pursued, which accounts for the spatial inhomogeneity of the base flow.

1F.G. acknowledges funding from ERC SimCoMiCs 280117

9:18AM M17.00007 Flapping jets and monodisperse droplets formed by the Kelvin-Helmholtz instability, OLIVER MCRAE, Boston University, ANTOINE GAillard, Boston University/Ecole Normale Superieure, JAMES BIRD, Boston University — When a straw is used to blow air into a glass of liquid, typically one of two behaviors is observed: a dimple in the liquid’s surface, or a frenzy of waves and bubbles. However, under certain conditions intermediate regimes can develop. In these regimes periodic waves progress into a flapping jet that can develop into monodisperse airborne droplets. The precise mechanism for the formation of these regimes is not well understood. Here we show that the Kelvin-Helmholtz instability is responsible for the formation of the flapping jet. We inject a continuous stream of gas into the liquid surface and observe both optically and acoustically the deformation of the liquid-air interface as we systematically adjust various parameters. Previous research has shown that the frequency of a liquid-gas oscillator can be regulated by the compressibility of the gas phase. Here we present the Kelvin-Helmholtz instability, with the treatment of the fluids as incompressible, as the regulator of the frequency. The formation of the jet droplets can thus be characterized by the Kelvin-Helmholtz and Rayleigh-Plateau instabilities. We anticipate the flapping jet phenomenon could be exploited to create monodisperse aerosols and emulsions, and may be relevant in analogous systems such as pulmonary flow.
Solutal Marangoni instability in layered two-phase flow

Jason Picardo, T.G. Radhakrishna, S Pushpavanam, Indian Inst of Tech-Madras — In this work, the instability of layered two-phase flow caused by the presence of a surface-active solute is studied. The fluids are density matched to focus on surfactant effects. The fluids flow between two flat plates, which are maintained at different solute concentrations. This establishes a constant flux of soluble surfactant from one fluid to the other, in the base state. A linear stability analysis is carried out, supported by energy budget calculations. The flow is first analyzed in the creeping flow regime. Long wave as well as short wave Marangoni instabilities are identified, each with a distinct energy signature. The short wave instability manifests as two distinct modes, characterized by the importance of interfacial deformations or lack thereof. The primary instability switches between these different modes as parameters are varied. The effect of small but finite inertia on these solutal Marangoni modes is then examined. The effect of soluble surfactant on a finite inertia flow is also studied, with focus on the transition from the viscosity-induced instability to solutal Marangoni instability. This analysis is relevant to microfluidic applications, such as solvent extraction, in which mass transfer is carried out between stratified immiscible fluids.

The effect of surfactant on counter-current gas-liquid flows in vertical tubes

Ivan Zadrazil, Omar Matar, Christos Markides, Imperial College London — Counter-current gas-liquid flows in vertical tubes are often accompanied by flow reversals. This so-called flooding phenomenon could occur for at least a part of the liquid phase from a counter-current to a co-current state, against the action of gravity. This phenomenon is of central importance to the oil-and-gas and nuclear industries, and has received considerable attention experimentally. The large majority of the previous work in this area, however, has considered the case of pure fluids, in the absence of additives; the latter are used frequently in industry in an attempt to control the onset of various flow regimes with little understanding of the mechanisms underlying their influence on the interfacial dynamics. In this study, we address this issue by investigating the dynamics of flooding in the presence of surfactants in a 4 m long, 32.4 mm nominal bore polymethyl methacrylate test section using high-speed shadowgraphy, and axial-view imaging. The system parameters include the superficial gas and liquid velocities, and surfactant concentration. We show that the presence of surfactant can have a dramatic effect on the flow structures and the onset of flooding. The mechanisms responsible for these phenomena are analysed.

Stability analysis of two phase stratified flow in a rectangular channel

Dinesh Bhagavatula, Pushpavanam S, Indian Inst of Tech-Madras — Two phase stratified flows arise in extraction operations in microfluidic systems. It is well established that stratified flows in between two infinite plates is always unstable. However such flows are experimentally observed in microchannels. To understand this paradox we perform a linear stability analysis of stratified two phase Poiseuille flow in a rectangular duct. A two-dimensional fully developed flow through the rectangular channel is considered. The linearized equations along with the boundary conditions in primitive variable formulation are numerically solved using Chebyshev collocation method. All the primitive variables, which are the velocity and pressure fields, are retained in the linearised governing equations. Since boundary conditions for disturbance pressure do not exist, the corresponding compatibility conditions derived from the Navier-Stokes equations are collocated both at the walls and the interface. The resulting eigen-value problem is solved using a shift and invert Arnoldi algorithm. The role of different parameters such as Aspect ratio, density ratio, viscosity ratio on the stability characteristics is analyzed. The stability results are validated in the limit of large Aspect Ratios. The flow fields are sought as a combination of Chebyshev polynomials in both y and z directions.

Rayleigh-Taylor mixing with time-dependent acceleration

Snezhana Abarzhi, Carnegie Mellon University — We extend the momentum model to describe Rayleigh-Taylor (RT) mixing driven by a time-dependent acceleration. The acceleration is a power-law function of time, similarly to astrophysical and plasma fusion applications. In RT flow the dynamics of a fluid parcel is driven by a balance per unit mass of the rates of momentum gain and loss. We find analytical solutions in the cases of balanced and imbalanced gains and losses, and identify their dependence on the acceleration exponent. The existence is shown of two typical regimes of self-similar RT mixing: acceleration-driven Rayleigh-Taylor-type and dissipation-driven Richtmyer-Meshkov-type with the latter being in general non-universal. Possible scenarios are proposed for transitions from the balanced dynamics to the imbalanced self-similar dynamics. Scaling and correlations properties of RT mixing are studied on the basis of dimensional analysis. Departures are outlined of RT dynamics with time-dependent acceleration from canonical cases of homogeneous turbulence as well as blast waves with first and second kind self-similarity.

Stochastic model of Rayleigh-Taylor mixing with time-dependent acceleration

Nora Swisher, Snezhana Abarzhi, Carnegie Mellon University — We report the stochastic model of Rayleigh-Taylor (RT) mixing with time-dependent acceleration. RT mixing is a statistically unsteady process, where the means values of the flow quantities as well as the fluctuations around these means are time-dependent. A set of nonlinear stochastic differential equations with multiplicative noise is derived on the basis of rigorous momentum model and group theory analyses to account for the randomness of RT mixing. A broad range of parameter regime is investigated: self-similar asymptotic solutions are found; new regimes of RT mixing dynamics are identified. We show that for power-law asymptotic solutions describing RT mixing the exponent is relatively insensitive and pre-factor is sensitive to the fluctuations, and find the statistic invariants of the dynamics in each of the new regimes.

Support of the National Science Foundation is warmly appreciated

Stochastic model of Rayleigh-Taylor mixing with time-dependent acceleration

Nora Swisher, Snezhana Abarzhi, Carnegie Mellon University — We report the stochastic model of Rayleigh-Taylor (RT) mixing with time-dependent acceleration. RT mixing is a statistically unsteady process, where the means values of the flow quantities as well as the fluctuations around these means are time-dependent. A set of nonlinear stochastic differential equations with multiplicative noise is derived on the basis of rigorous momentum model and group theory analyses to account for the randomness of RT mixing. A broad range of parameter regime is investigated: self-similar asymptotic solutions are found; new regimes of RT mixing dynamics are identified. We show that for power-law asymptotic solutions describing RT mixing the exponent is relatively insensitive and pre-factor is sensitive to the fluctuations, and find the statistic invariants of the dynamics in each of the new regimes.

Support of the National Science Foundation is warmly appreciated

Flow Instability: Rayleigh-Taylor II

Rayleigh-Taylor mixing with time-dependent acceleration

Snezhana Abarzhi, Carnegie Mellon University — We extend the momentum model to describe Rayleigh-Taylor (RT) mixing driven by a time-dependent acceleration. The acceleration is a power-law function of time, similarly to astrophysical and plasma fusion applications. In RT flow the dynamics of a fluid parcel is driven by a balance per unit mass of the rates of momentum gain and loss. We find analytical solutions in the cases of balanced and imbalanced gains and losses, and identify their dependence on the acceleration exponent. The existence is shown of two typical regimes of self-similar RT mixing: acceleration-driven Rayleigh-Taylor-type and dissipation-driven Richtmyer-Meshkov-type with the latter being in general non-universal. Possible scenarios are proposed for transitions from the balanced dynamics to the imbalanced self-similar dynamics. Scaling and correlations properties of RT mixing are studied on the basis of dimensional analysis. Departures are outlined of RT dynamics with time-dependent acceleration from canonical cases of homogeneous turbulence as well as blast waves with first and second kind self-similarity.

Support of the National Science Foundation is warmly appreciated

Stochastic model of Rayleigh-Taylor mixing with time-dependent acceleration

Nora Swisher, Snezhana Abarzhi, Carnegie Mellon University — We report the stochastic model of Rayleigh-Taylor (RT) mixing with time-dependent acceleration. RT mixing is a statistically unsteady process, where the means values of the flow quantities as well as the fluctuations around these means are time-dependent. A set of nonlinear stochastic differential equations with multiplicative noise is derived on the basis of rigorous momentum model and group theory analyses to account for the randomness of RT mixing. A broad range of parameter regime is investigated: self-similar asymptotic solutions are found; new regimes of RT mixing dynamics are identified. We show that for power-law asymptotic solutions describing RT mixing the exponent is relatively insensitive and pre-factor is sensitive to the fluctuations, and find the statistic invariants of the dynamics in each of the new regimes.

Support of the National Science Foundation is warmly appreciated
8:26AM M18.00003 The Evolution of the single-mode Rayleigh-Taylor instability under the influence of time-dependent accelerations1, PRAVEEN RAMAPRABHU, VARAD KARKHANIS, University of North Carolina at Charlotte, RAHUL BANERJEE, St. Paul’s Cathedral Mission College, Kolkata, HILDA VARSHOCHI, University of North Carolina at Charlotte, MANORANJAN KHAN, Jadavpur University, ANDREW LAWRIE, University of Bristol, VARIABLE G RT COLLABORATION — From detailed numerical simulations of the single-mode Rayleigh-Taylor (RT) instability driven by time-varying acceleration histories, we report on several findings of relevance to the performance of Inertial Confinement Fusion capsules. The incompressible, Direct Numerical Simulations (DNS) were performed in two- and three-dimensions, and over a range of density ratios of the fluid combinations (characterized by the Abwood number). We have investigated several acceleration histories, including acceleration profiles $g(t)$ of the general form $t^n$, with $n > -2$. For the 2D flow, results from numerical simulations are compared with a potential flow model developed and reported as part of this work. When the simulations are extended to three dimensions, bubble and spike growth rates are in agreement with an extension to the drag buoyancy model with modifications for time-dependent acceleration histories. We have come up with simple analytic solutions to the Drag Buoyancy model for variable $g$ flows, and compared the solution with the 2D and 3D DNS results.

1This work was supported in part by the (U.S.) Department of Energy (DOE) under Contract No. DE-AC52-06NA2-5396.

8:39AM M18.00004 Simulations of mixing in Inertial Confinement Fusion with front tracking and sub-grid scale models, VERINDER RANA, HYUNKYUNG LIM, JEREMY MELVIN, Stony Brook University, BAOLIAN CHENG, Los Alamos National Laboratory, JAMES GLIMM, Stony Brook University, DAVID SHARP, Los Alamos National Laboratory — We present two related results. The first discusses the Richtmyer-Meshkov (RMI) and Rayleigh-Taylor instabilities (RTI) and their evolution in Inertial Confinement Fusion simulations. We show the evolution of the RMI to the late time RTI under transport effects and tracking. The role of the sub-grid scales helps capture the interaction of turbulence with diffusive processes. The second assesses the effects of concentration on the physics model and examines the mixing properties in the low Reynolds number hot spot. We discuss the effect of concentration on the Schmidt number. The simulation results are produced using the University of Chicago code FLASH and Stony Brook University’s front tracking algorithm.

8:52AM M18.00005 Rayleigh-Taylor instability (RTI) for a yield-stress fluid, ILHAM MAIMOUNI, Paris-Est University (Navier Laboratory) / Schlumberger (SRPC), JULIE GOYON, Paris-Est University (Navier Laboratory), ETIENNE LECQ, NICOLAS FLAMANT, THIBAULT PRINGUEY, Schlumberger (SRPC), PHILIPPE COUSSOT, Paris-Est University (Navier Laboratory) — RTI is of great interest in several domains such as oil industry, geology and high-energy density physics. We experimentally study this instability for a yield-stress fluid, i.e. a fluid that is solid under a certain critical yield stress and liquid above. For that, we superimpose, at rest, two rheologically - controlled immiscible fluids of different densities, a yield stress fluid under a heavier Newtonian one, and we observe the interface. For a given density difference, the instability occurs below a critical yield stress in the form of fingers of one fluid abruptly spreading through the other one. Above this critical yield stress, the interface remains undeformed. This set of data provides an empirical criterion for interface. For a given density difference, the instability occurs below a critical yield stress in the form of fingers of one fluid abruptly spreading through the other one. Above this critical yield stress, the interface remains undeformed. This set of data provides an empirical criterion for this instability. We find that this criterion is neither predicted by simple elastic material theory, nor by the assumption of yielding phenomena at sufficient initial perturbation amplitude. Instead, RTI occurs for a sufficiently larger density difference to yield stress ratio and the finger wavelength is independent of the sample size. Finally we show that the instability characteristics can be explained by the ability of a local perturbation beyond a critical size to penetrate the material.

9:05AM M18.00006 A 3D Bubble Merger Model for RTI Mixing, BAOLIAN CHENG, Los Alamos National Laboratory — In this work we present a model for the merger processes of bubbles at the edge of an unstable acceleration driven mixing layer. Steady acceleration defines a self-similar mixing process, with a time-dependent inverse cascade of structures of increasing size. The time evolution is itself a renormalization group evolution. The model predicts the growth rate of a Rayleigh-Taylor chaotic fluid-mixing layer. The 3-D model differs from the 2-D merger model in several important ways [1]. Beyond the extension of the model to three dimensions, the model contains one phenomenological parameter, the variance of the bubble radii at fixed time. The model also predicts several experimental numbers: the mixing rate, the bubble height separation at the time of merger, and the bubble height ratio to the radius aspect ratio, which is in good agreement with experiments. Applications to recent NIF and Omega experiments will be discussed. This work was performed under the auspices of the U.S. Department of Energy by the Los Alamos National Laboratory under Contract No. W-7405-ENG-36.


9:18AM M18.00007 The effect of an obstruction on the Rayleigh-Taylor instability, CHRISTOPHER BROWN, STUART DALZIEL, University of Cambridge — This talk discusses the effect of an obstruction on the evolution of the Rayleigh-Taylor instability in a confined geometry at low Atwood numbers. The introduction of an obstacle at the height of the initial interface results in dramatic changes to the dynamics of mixing, even when this obstacle is only a few percent of the domain width. Two situations are investigated using laboratory experiments and implicit large eddy simulations. In the first case, a single horizontal opening connects the upper and lower layers. A bidirectional flow exchanges fluid through the opening, this establishes a circulation cell in each layer. These cells exist quasi-steadily for long periods, increasing the time required for mixing compared with the classical case and resulting in a more uniformly mixed final stratification. The second case has two horizontal openings, one either side of the obstruction. This results in markedly different dynamics. The flow through each of the openings switches back and forth between being bidirectional (as with the single opening case) and unidirectional, with the direction of the unidirectional exchange reversing with a constant period. ©British Crown Owned Copyright 2015/AWE

9:31AM M18.00008 Dripping from a curved ceiling: a linear optimal transient growth analysis, GIOELE BALESTRA, Laboratory of Fluid Mechanics and Instabilities, STI, EPFL, Lausanne, Switzerland, ANNA LEE, JOEL MARTHELOT, Department of Mechanical Engineering Massachusetts Institute of Technology Cambridge, MA 02139, USA, PIERRE-THOMAS BRUN, Department of Mathematics Massachusetts Institute of Technology Cambridge, MA 02139, USA, PEDRO M. REIS, Department of Mechanical Engineering Massachusetts Institute of Technology Cambridge, MA 02139, USA, FRANCOIS GALLAIRE, Laboratory of Fluid Mechanics and Instabilities, STI, EPFL, Lausanne, Switzerland — We investigate theoretically the stability of a thin viscous film on the underside of a curved cylindrical surface. Gravity acts both as a stabilizing force originating in the progressive drainage of the film and as a destabilizing force prone to form dripping droplets. This result first reported by Trinh et al. when studying the region near the top of a coated cylinder is here generalized to the entire structure. The governing parameters, namely the Bond number, which prescribes the relative importance of gravity and surface tension forces, and the initial film thickness to cylinder radius ratio are found not to play a role in the long time stability of the film. However, the system displays a linear transient growth potential which increases exponentially with the Bond number. Depending on its value, there is a critical initial disturbance amplitude above which non-linear effects yield the formation of droplets, suggesting that the transition to dripping is noise and roughness dependent.
9:44AM M18.00009 Validation of Nek5000 against low-Atwood, single-mode Rayleigh Taylor experiments¹. MAXWELL HUTCHINSON, Univ of Chicago — Experiments by Wilkinson and Jacobs [1] demonstrate the stagnation and reacceleration phases of the low-Atwood, single-mode Rayleigh-Taylor instability between two water mixtures. We reproduce the experimental conditions of three runs in direct numerical simulations using the spectral element code Nek5000. The simulations required 17 billion grid points on 512 thousand cores of the Mira supercomputer to reach Rayleigh numbers up to 90 million. We extend the vertical dimension to reach higher bubble aspect ratios and demonstrate the limits of wall-bounded single-mode studies. Finally, exploration of the full-field results reveals spanwise secondary flows that enhance mixing at low to moderate Reynolds number.

¹This research used resources of the Argonne Leadership Computing Facility, which is a DOE Office of Science User Facility supported under Contract DE-AC02-06CH11357.

9:57AM M18.00010 Scale-coupling and Nonlinear Dynamics in Compressible Rayleigh-Taylor Instability, DONGXIAO ZHAO, Department of Mechanical Engineering, University of Rochester, HUSSEIN ALUIE, 1. Laboratory for Laser Energetics, University of Rochester 2. Department of Mechanical Engineering, University of Rochester, RICCARDO BETTI, 1. Laboratory for Laser Energetics, University of Rochester 2. Department of Mechanical Engineering & Physics and Astronomy, University of Rochester — The Rayleigh-Taylor instability (RTI) is a ubiquitous instability occurring in laser-accelerated targets and in many geophysical and astrophysical environments. Mass ablation or evaporation can significantly alter the RTI evolution in laser-driven plasmas as well as in molecular clouds and supernovae ejecta. We perform single and multimode simulations of 3D compressible RTI using a hybrid pseudospectral-compact finite difference scheme. We will present preliminary results on how different length scales are dynamically coupled at various stages of the instability, especially in the highly nonlinear regime. Our goal is to understand how ablation alters this scale-coupling and its effect on the overall growth of the mixed layer, which may have significant ramifications to modeling efforts in implosion physics.

Tuesday, November 24, 2015 8:00AM - 10:10AM
Session M19 Turbulence: Measurements

8:00AM M19.00001 The Decay of Turbulence After it Stops Rotating.¹. J. BLAIR PEROT, CHRIS ZUSI, Univ of Mass - Amherst — It is well known that the value of the power-law decay rate is reduced when turbulence is rotated. Less well known is how rotating turbulence behaves when system rotation stops. 512³ DNS simulations of properly initialized isotropic turbulence at a variety of Reynolds numbers and rotation rates are used to show that immediately after rotation stops decaying turbulence has an exponential and not a classical power-law decay. Exponential decay is equivalent to an infinite power-law decay exponent and is a result of a constant physical turbulent timescale. In contrast, classical power-law decaying turbulence has a turbulent timescale that is proportional to the time itself. The implications for the modeling of the dissipation rate, and the physics of the turbulent decay process, are discussed.

¹This work is support by the National Science Foundation

8:13AM M19.00002 On the Relation between Spatio-Temporal Forcing and Structure of Turbulence, DOUGLAS CARTER, FILIPPO COLETTI, University of Minnesota — The different methods to force turbulence in physical or numerical experiments can have significant effects on the fluid dynamics. Understanding how a given forcing scheme maps on the flow statistics is important both to reproduce desired flow features, and to gain insight in the transfer of energy across the scales. Here we consider the case of the turbulent flow generated by the interaction of pulsating jets. We present a novel installation where pressurized air is issued through 256 independently actuated valves, arranged in symmetric rectangular arrays over two facing planes. The small net mass fluctuations used in the conditioning, we can test both direction and locality. We use these tools on experimental data taken from a flowing soap film, an approximately 2D turbulent flow. The Reynolds number is varied over a wide range to determine the entropy’s scaling with Reynolds number

8:26AM M19.00003 Correlational signatures of time-reversal symmetry breaking in two-dimensional flow, CHARLIE HOGG, NICHOLAS OUELLETTE, Stanford University — Classical turbulence theories posit that broken spatial symmetries should be (statistically) restored at small scales. But since turbulent flows are inherently dissipative, time reversal symmetry is expected to remain broken throughout the cascade. However, the precise dynamical signature of this broken symmetry is not well understood. Recent work has shed new light on this fundamental question by considering the Lagrangian structure functions of power. Here, we take a somewhat different approach by studying the Lagrangian correlation functions of velocity and acceleration. We measured these correlations using particle tracking velocimetry in a quasi-two-dimensional electromagnetically driven flow that displayed net inverse energy transfer. We show that the correlation functions of the velocity and acceleration magnitudes are not symmetric in time, and that the degree of asymmetry can be related to the flux of energy between scales, suggesting that the asymmetry has a dynamical origin.

8:39AM M19.00004 Determining the direction of a turbulent cascade¹. WALTER GOLDBURG, University of Pittsburgh, RORY CERBUS, Okinawa Institute of Science and Technology — In two-dimensional (2D) turbulence, one expects a cascade of energy to larger spatial scales, while the enstrophy cascade is to smaller ones. Here we present a new tool to study cascades using simple ideas borrowed from information theory. It is entirely unrelated to the Navier-Stoke’s equations or any scaling arguments. We use the conditional entropy (conditioned uncertainty) of velocity fluctuations on one scale conditioned on another larger or smaller scale. If the entropy is larger after conditioning on larger scales rather than smaller ones, then the cascade is to smaller scales. By varying the scale of the velocity fluctuations used in the conditioning, we can test both direction and locality. We use these tools on experimental data taken from a flowing soap film, an approximately 2D turbulent flow. The Reynolds number is varied over a wide range to determine the entropy’s scaling with Reynolds number

¹OIST
8:52AM M19.00005 On velocity gradient dynamics and fine-scale structure: experiments support DNS and models , JOHN LAWSON, University of Cambridge, JAMES DAWSON, Norwegian University of Science and Technology — The fine scales of turbulence are embodied by statistics of velocity gradients. In solving exact equations for their evolution, the challenge is to specify how the pressure Hessian acts. This is determined by the footprints that “structures” of enstrophy and strain leave in conditional average pressure fields. We use direct and approximate conditional averaging methods to extract this structure from different turbulence datasets: a direct numerical simulation and a unique scanning tomography experiment in a “French washing machine”. Direct comparisons between simulation and experiment show the structure and resulting dynamics are in excellent, quantitative agreement. This evidence supports existing modelling approaches and provides insights towards their refinement. Moreover, it demonstrates the dynamical significance and the reproducibility of fine-scale structure.

9:05AM M19.00006 Multi-level segment analysis: definition and applications in turbulence , LIPO WANG, Shanghai JiaoTong Univ. — The interaction of different scales is among the most interesting and challenging features in turbulence research. Existing approaches used for scaling analysis such as structure-function and Fourier spectrum method have their respective limitations, for instance scale mixing, i.e. the so-called infrared and ultraviolet effects. For a given function, by specifying different window sizes, the local extremal point set will be different. Such window size dependent feature indicates multi-scale statistics. A new method, multi-level segment analysis (MSA) based on the local extrema statistics, has been developed. The part of the function between two adjacent extremal points is defined as a segment, which is characterized by the functional difference and scale difference. The structure function can be differentially derived from these characteristic parameters. Data test results show that MSA can successfully reveal different scaling regimes in turbulence systems such as Lagrangian and two-dimensional turbulence, which have been remaining controversial in turbulence research. In principle MSA can generally be extended for various analyses.

9:18AM M19.00007 Estimation of Turbulent Wall Jet Velocity Fields for Noise Prediction , ADAM NICKELS, LAWRENCE UKIELEY, University of Florida, ROBERT REGER, LOUIS CATTAFAESTA, Florida State University — Estimation of the time-dependent turbulent velocity field of a planar wall jet based on discrete surface pressure measurements is performed using stochastic estimation in both the time and frequency domains. Surface pressure measurements were acquired simultaneously with planar Particle Image Velocimetry (PIV) snapshots, obtained at a relatively reduced rate. Proper Orthogonal Decomposition (POD) is then applied to both the surface pressure probes and the PIV snapshots, allowing for the isolation of portions of the wall pressure and velocity field signals that are well correlated. Using the time-varying pressure expansion coefficients as unconditional variables, velocity expansion coefficients are estimated and used to produce reconstructed estimates of the velocity field. Optimization in terms of number of unconditional probes employed, location of probes, and effects of PIV discretization are investigated with regards to the resulting estimates. Coupled with this analysis, Poisson’s equation for fluctuating pressure is solved such that the necessary source terms of an acoustic analogy can be calculated for estimates of the far-field acoustic. Specifically in this work, the effects of using estimated velocity fields to solve for the hydrodynamic pressure and acoustic pressure will be studied.

9:31AM M19.00008 An experimental Lagrangian study of inhomogeneous turbulence , NICKOLAS STELZENMULLER, NICOLAS MORDANT, Laboratoire des Ecoulements Gophysiques et Industriels (LEGI) — We investigate experimentally the Lagrangian properties of inhomogeneous turbulence in the general scope of dispersion studies in natural and industrial flows. Inhomogeneous turbulence is generated using a random Fourier modes method with a varying power-law spectral slope, whereas the turbulence is produced using the random Fourier modes method with a varying power-law spectral slope, whereas the roughness topography, whether it is idealized 2D and 3D or irregular with multi-scale features, impacts the frictional drag. A previous study from Flack and Schultz (2010) presented a new model to estimate frictional drag based on surfaces extremal points is defined as a segment, which is characterized by the functional difference and scale difference. The structure function can be differentially derived from these characteristic parameters. Data test results show that MSA can successfully reveal different scaling regimes in turbulence systems such as Lagrangian and two-dimensional turbulence, which have been remaining controversial in turbulence research. In principle MSA can generally be extended for various analyses.

9:44AM M19.00009 Skin-Friction Measurements on Mathematically Generated Roughness in a Turbulent Channel Flow , JULIO BARROS1, MICHAEL SCHULTZ2, KAREN FLACK3, United States Naval Academy — Engineering systems are affected by surface roughness, however, predicting frictional drag has proven to be challenging. One open question is how roughness topography, whether it is idealized 2D and 3D or irregular with multi-scale features, impacts the frictional drag. A previous study from Flack and Schultz (2010) presented a new model to estimate frictional drag based on surfaces extremal points is defined as a segment, which is characterized by the functional difference and scale difference. The structure function can be differentially derived from these characteristic parameters. Data test results show that MSA can successfully reveal different scaling regimes in turbulence systems such as Lagrangian and two-dimensional turbulence, which have been remaining controversial in turbulence research. In principle MSA can generally be extended for various analyses.

9:57AM M19.00010 Inner–outer interactions in a turbulent boundary layer overlying complex roughness , GOKUL PATHIKONDA, Univ of Illinois, Urbana-Champaign, KENNETH T. CHRISTENSEN, Univ of Notre Dame, Notre Dame — Stereo PIV and hot-wire measurements were performed in a rough-wall turbulent boundary layer to investigate the inner–outer interactions across the roughness sublayer. The PIV was performed in a spanwise–wall-normal plane and hot-wire measurements were conducted with single- and two-probe methods. The complex roughness with a wide distribution of roughness scales has been shown previously to induce alternating high- and low- momentum pathways (HMPs and LMPs)—imprints of roughness-induced secondary flows. Differences in the streamwise velocity and turbulent kinetic energy between a HMP-LMP pair are est turbed in the current study. The respective inner–outer interactions are quantified by the amplitude modulation correlation coefficient, the time-delayed velocity correlation coefficient maps and by a 2-point correlation method proposed by Mathis et al. (J. Fluid Mech. 681 (2011): 537-566). It is observed that the strength of such interactions, as measured in this calibration-framework, is generally stronger in the rough-wall than smooth-wall flow, and relatively stronger at an LMP than at a corresponding HMP.

1Department of Mechanical Engineering
2Naval Architecture and Ocean Engineering Department
3Department of Mechanical Engineering
Tuesday, November 24, 2015 8:00AM - 10:10AM – Session M20 Turbulence: Wakes and Flows Behind Grids 208 - Gregory Bewley, Max Planck Institute for Dynamics and Self-Organization, Germany

8:00AM M20.00001 Turbulence decay downstream of an active grid1. GREGORY BEWLEY, EBERHARD BODENSCHATZ, Max Planck Institute for Dynamics and Self-Organization — A grid in a wind tunnel set up turbulence that has a certain large-scale structure. The moving parts in a so-called “active grid” can be programmed to produce different structures. We use a special active grid in which each of 129 paddles on the grid has its own position-controlled servomotor that can move independently of the others. We observe among other things that the anisotropy in the amplitude of the velocity fluctuations and in the correlation lengths can be set and varied with an algorithm that oscillates the paddles in a specified way. The variation in the anisotropies that we observe can be explained by our earlier analysis of anisotropic “soccer ball” turbulence (Bewley, Chang and Bodenschatz 2012, Phys. Fluids). We define the influence of this variation in structure on the downstream evolution of the turbulence.

1 with Eberhard Bodenschatz and others

8:13AM M20.00002 Some effects of vortex shedding in grid-generated turbulence1. GIANFRANCESCO MELINA, PAUL J.K. BRUCE, JOHN CHRISTOS VASSILICOS, Imperial College London — We perform hot-wire measurements in a wind tunnel downstream of different types of turbulence-generating grids: a regular grid (RG60), a fractal square grid (FSG17) and a single square grid (SSG). We characterize the flow highlighting similarities and differences between the grids and between the production and the decay regions of turbulence. We focus on the effects of vortex shedding from the bars of the grids. For this purpose we design a novel 3D configuration formed by the SSG and a set of four splitter plates detached from the grid. We show that, by placing the splitter plates, the peak of turbulence intensity on the centerline is reduced and its location is moved downstream. We compare data from the different turbulence generators and find that a reduction of vortex shedding energy correlates with an increase in the magnitudes of the skewness and flatness of the turbulent velocity fluctuations in the production region.

1The authors acknowledge support form the EU through the FP7 Marie Curie MULTISOLVE project (grant agreement No. 317269).

8:26AM M20.00003 Alignments and small scale statistics in the production region of grid turbulence. IMMANUVEL PAUL, GEORGE PAPADAKIS, JOHN CHRISTOS VASSILICOS, Department of Aeronautics, Imperial College London, TURBULENCE, MIXING AND FLOW CONTROL GROUP TEAM — Direct Numerical Simulation (DNS) of turbulent flow generated by a single square grid is investigated using an unstructured finite volume method. The maximum value of the Taylor length-based Reynolds number throughout the computed flow field is about 40. The main focus of this study is on the production region which lies in the lee of the grid where turbulence builds up. Statistics of vorticity and of eigenvalues ($\lambda_i$, where $i=1,2,3$) and eigenvectors ($e_i$, where $i=1,2,3$) of the fluctuating strain rate tensor ($S_{ij}$) are analyzed. It is observed that the PDFs of all the eigenvalues in the production region are highly non-gaussian. The PDFs of the compressive ($\lambda_1$) and intermediate ($\lambda_2$) eigenvalues are strongly skewed to negative and positive values respectively. The energy spectrum of the streamwise fluctuating velocity has a well-defined power law with an exponent around -2 or -5/3 over more than one decade depending on the position in the production region. It is also observed that the most extensive eigenvector ($e_1$) and the intermediate eigenvector ($e_2$) align significantly with vorticity vector in the production region, which in turn increases average enstrophy production.

8:39AM M20.00004 Decay of grid turbulence in a closed box. STPHANE PERRARD, WILLIAM IRVINE, James Franck Institute, University of Chicago, IRVINE’S LAB TEAM — We investigate the decay of a turbulent flow in the absence of mean flow. By accelerating a square grid in a water tank, we generate an array of wakes that induces a 3 dimensional turbulent flow with a Reynolds number of about $Re \approx 5 \times 10^4$. After the impulse excitation (about 100ms), a decay in time of this turbulent flow is observed. The entire decay process lasts for hours while the dissipative length rises up through scales over time. We follow and characterize both in space and time this turbulent decay process through several decades.

8:52AM M20.00005 The turbulent flow generated by inhomogeneous multiscale grids. SHAOKAI ZHENG, PAUL J K BRUCE, J MICHAEL R GRAHAM, JOHN CHRISTOS VASSILICOS, Imperial College London — A group of inhomogeneous multiscale grids have been designed and tested in a low speed wind tunnel in an attempt to generate bespoke turbulent shear flows. Cross-wire anemometry measurements were performed in different planes parallel to the grid and at various streamwise locations to study turbulence development behind each of the different geometry grids. Two spatially separated single hot wires were also used to measure transverse integral length scale at selected locations. Results are compared to previous studies of shearless mixing layer grids and fractal grids, including mean flow profiles and turbulence statistics.

9:05AM M20.00006 Power Law Decay in High Intensity Turbulence. TIMOTHY KOSTER, ALEJANDRO PUGA, BAO LONG NGUYEN, JOHN LARUE, Univ of California - Irvine — In the study reported herein, the region where the power decay law is applicable for active grid generated turbulence is found by an iterative approach which determines the largest range where the ratio of the dissipation from the power law and the dissipation from the temporal velocity derivative are unity. The square of the Taylor microscale, as noted by Batchelor (1953), is linearly related to downstream distance relative to the virtual origin and can be used in a straightforward manner to find the virtual origin. The fact that the decay of downstream velocity variance is described by a power law is shown to imply a power law behavior for various other parameters such as the dissipation, the integral length scale, the Taylor microscale, the Kolmogorov microscale and the Taylor Reynolds number and that there is an algebraic relationship between the various power law exponents. Results are presented for various mean velocities to show the decay exponent as a function of the Taylor Reynolds number.

9:18AM M20.00007 Dissipative Effects on Inertial-Range Statistics at High Reynolds Numbers. MICHAEL SINHUBER, GREGORY BEWLEY, EBERHARD BODENSCHATZ, Max Planck Institute for Dynamics and Self-Organization — Using the unique capabilities of the Variable Density Turbulence Tunnel at the Max Planck Institute for Dynamics and Self-Organization, we were able to measure extremely long time series of up to $10^{10}$ samples of the turbulent fluctuating velocity in a well-controlled environment at a wide range of high Reynolds numbers up to $Re = 1600$. These classical grid measurements were conducted using both classical hot-wire probes as well as NSTAP probes developed at Princeton University. With these long datasets, we were able to uncover fine details of the structure functions and their scaling behavior. We find that deviations from ideal scaling is anchored to the small scales and that dissipation influences the inertial-range statistics even up to $r/\eta = 1000$. 

Interaction of two high Reynolds number axisymmetric turbulent wakes, M. OBLIGADO, Imperial College London, S. KLEIN, TU Braunschweig, J.C. VASSILICOS, Imperial College London — With the recent discovery of non-equilibrium high Reynolds number scalings in the wake of axisymmetric plates (Nedic et al., PRL, 2013), it has become of importance to develop an experimental technique that permits to easily discriminate between different wake scalings. We propose an experimental setup that tests the presence of non-equilibrium turbulence using the streamwise variation of velocity fluctuations between two bluff bodies facing a flow. We have studied two different sets of plates (one with regular and another with irregular peripheries) with Hot-Wire Anemometry in a wind tunnel. By acquiring streamwise profiles for different plate separations and identifying the wake interaction length for each separation it is possible to estimate the streamwise evolution of the single wake width. From this evolution it is also possible to deduce the turbulence dissipation scalings. This work generalizes previous studies on the interaction of plane wakes (see Gomes-Fernandes et al., JFM, 2012) to include axisymmetric wakes. We find that the wake interaction length proposed in this cited work and a constant anisotropy assumption can be used to collapse the streamwise developments of the first three moments.

Scale-by-scale energy fluxes in anisotropic non-homogeneous turbulence behind a square cylinder, FELIPE ALVES PORTELA, GEORGE PAPADAKIS, JOHN CRISTOS VASSILICOS, Imperial College London — The turbulent wake behind a square section cylinder is studied by means of high resolution direct numerical simulations using an in-house finite volume code. The Reynolds number based on the cylinder side is 3900. Single- and two-point statistics are collected in the lee of the cylinder for over 30 shedding periods, allowing for an extensive description of the development of the turbulence. The power spectrum in the frequency domain of velocity fluctuations displays a near -5/3 power law in the near wake, where the turbulence is neither isotropic nor homogeneous. In the same region of the flow, two-point statistics reveal a direct cascade of fluctuating kinetic energy down the scales as a result of the combined effect of linear and non-linear interactions. For scales aligned with the mean flow the non-linear interactions dominate the cascade. Conversely, for scales normal to the mean flow the cascade is dominated by the linear interactions while the non-linear term is mostly responsible for redistributing energy to different orientations.

The authors acknowledge support from the EU through the FP7 Marie Curie MULTISOLVE project (grant agreement No. 317269)

On the Large Scale Dynamics in the Wake of a Fractal Obstacle, JONATHAN HIGHLAND, WERNER BREVIS, Univ of Sheffield — In a water flume three-dimensional Particle Tracking Velocimetry is used to capture the turbulent wake of two full-width and wall-mounted obstacles: The first obstacle is a uniformly spaced array of square cylinders of same length-scale; the second is a three-iteration pre-fractal based on a the deterministic Sierpinski Carpet. Both obstacles emerge from the water surface and had the same porosity. For the description of the instantaneous vortical structures the velocity gradient tensor is analysed. It is found that whilst the largest length scales of the vorticity field in the wake, the smaller length-scale within the obstacle caused intense vortical structures within the near field of the wake. To further investigate the spatio-temporal behaviour of the wake a simple and integrated use of the Proper Orthogonal Decomposition (POD) and Dynamic Mode Decomposition (DMD) is introduced. POD is used to rank the spatial structures relatable to the total variance (i.e. vorticity) while DMD is used to identify their dominant oscillation frequencies and spatial characteristics. From the POD it is clear that the largest length-scale creates spatially dominant structures, whilst the DMD extracts a set of oscillatory frequencies relatable to each fractal length-scale.

Tuesday, November 24, 2015 8:00AM - 9:57AM – Session M21 Turbulence: Modeling II

A low-dimensional model for large-scale coherent structures, KUNLUN BAI, DANDAN JI, ERIC BROWN, Yale University — We demonstrate a methodology to predict the dynamics of the large-scale coherent structures in turbulence using a simple low dimensional stochastic model proposed by Brown and Ahlers (Phys. Fluids, 2008). The model terms are derived from the Navier-Stokes equations, including a potential term depending on the geometry of the system. The model has previously described several dynamical modes of the large-scale circulation (LSC) in turbulent Rayleigh-Bénard convection. Here we test a model prediction for the existence of a new mode where the LSC stochastically changes direction to align with different diagonals of a cubic container. The model successfully predicts the switching rate of the LSC at different tilting conditions. The success of the prediction of the switching mode demonstrates that a low-dimensional turbulent model can quantitatively predict the existence and properties of different dynamical states that result from boundary geometry.

Revisit on Proper Orthogonal Decomposition Method, MAHDI HOSSEINALI, JOSEPH HALL, University of New Brunswick — Understanding the underlying mechanisms of seemingly random movements in turbulent flows is the most challenging ongoing area of fluid dynamics. Structures with characteristic length scale comparable to the geometry of the flow, so called coherent structures, are assumed to be responsible for the major characteristic behaviors of the flow. These structures then break down to smaller structures and so on until they get damping on viscous level. Identification of coherent structures thus is of paramount importance in fluid dynamics. Among numerous methods POD seems to be the most successful approach to breaks the sophisticated turbulent field into a series of unbiased modes. Since its introduction to fluid dynamic community by Lumley the only major improvement was method of snapshots by Sirovich which is used today on PIV measurements. This talk is aimed to look at different forms of POD kernels which are mostly based on a physical point of view rather than pure mathematics.

Interaction of two-dimensional turbulence with a sheared channel flow: a numerical study, LEON KAMP, VITOR MARQUES ROSAS FERNANDES, GERTJAN VAN HEIJST, HERMAN CLERCKX, Eindhoven University of Technology — Interaction of large-scale flows with turbulence is of fundamental and widespread importance in geophysical fluid dynamics and also, more recently for the dynamics of fusion plasma. More specifically the interplay between two-dimensional turbulence and so-called zonal flows has gained considerable interest because of its relevance for transport and associated barriers. We present numerical results on the interaction of driven two-dimensional turbulence with typical sheared channel flows (Couette and Poiseuille). It turns out that a linear shear rate that is being sustained by moving channel walls (Couette flow) is far more effective in suppressing turbulence and associated transport than a Poiseuille flow. We explore the mechanisms behind this in relation to the width of the channel and the strength of the shear of the background flow. Also the prominent role played by the no-slip boundaries and the Reynolds stress is discussed.
8:39AM M21.00004 The effect of a solid boundary on homogeneous isotropic turbulence: an experimental investigation, BLAIR JOHNSON, EDWIN COWEN, Cornell University — An experimental study is performed to investigate the turbulent boundary layer at a smooth solid boundary in the absence of mean shear. Driven by a spatio-temporally varying randomly actuated synthetic jet array suspended above an enclosed water tank, high Reynolds number horizontally homogeneous isotropic turbulence is generated with negligible mean flow. Acoustic Doppler velocimetry and particle image velocimetry measurement techniques are used to characterize the near-boundary flow with statistical metrics such as turbulence intensities, turbulent kinetic energy, temporal and spatial spectra, and integral length scales. We compare various methods of computing dissipation rates and evaluate the assumptions of isotropy that are typically invoked. Furthermore, we consider Eulerian frequency spectra to improve dissipation estimates from single-point velocity measurements. Our investigations examine the effect of altering jet firing parameters on the integral length scale and resolving turbulent structures. We conclude with thoughts on the use of the dissipation rate to parameterize the bed stress in the absence of mean shear where traditional friction velocity methods struggle to fully capture the local stresses and energy present in turbulence.

8:45AM M21.00005 Evolution of the velocity gradient tensor in the near field of a square cylinder, MASSIMILIANO BREDA, OLIVER BUXTON, Imperial College London — The condition of the velocity gradient tensor (VGT) is analysed in the near field of the flow past a square cylinder. The data was acquired by tomographic particle image velocimetry at a moderate Reynolds number (Re = 16,000). The analysis focused on the evolution of the joint pdf (jpdf) between the second and third invariants of the characteristic equation for the VGT. These invariants are known to fully characterise the state of the VGT and have been previously used to observe the transition to a fully developed turbulent state in the interface region between turbulent and non-turbulent flows. We analyse the flow very close to the cylinder, where developed turbulence is not necessarily expected. The findings show that in the mean recirculation region, where the intermittency of the shear layer is low no tear drop shaped jpdf, indicative of fully developed turbulence, is found. Once the intermittency increases, tear drop shaped jpdfs are found where the velocity fluctuations are not Gaussian distributed, suggesting the small scales reach a fully developed turbulent state ahead of the large ones. This is further investigated by analysing the geometry of the local strain, the vorticity-strain rate alignment and enstrophy production.

8:52AM M21.00006 Hierarchical Structure of Fast Stretching Vortices in Turbulent Flows, MASATO HIROT,A YU NISHIO, SEIICHIRO IZAWA, YU FUKUNISHI, Tohoku University — Geometric relations between fast stretching vortices of a certain scale and vortices twice larger are investigated to understand the energy cascade process in a turbulent flow from a view point of vortex interactions. Multi-scaled vortices are extracted from a homogeneous isotropic turbulence using a band-pass filter based on the Fourier decomposition. An extracted vortex is reconstructed as a set of short cylindrical vortex segments. The stretching speed of a vortex segment, caused by the velocity field, which vortices twice the scale generate, is measured. Then, the vortex segments are classified into the three categories by their stretching speeds: the first is the "super fast" stretching vortices, whose stretching speed is in top 1% of total segments. The second is "moderately fast" stretching vortices, whose stretching speed is in top 10% of total segments. The third is "slowly or not" stretching vortices, whose stretching speed is lower than the top 10%. The geometric relations between the vortex segments and its surrounding vortices which are larger are analyzed in terms of the distances and the relative angles. The result shows that vortex segments tend to be aligned parallel or anti-parallel to the larger vortices for the "slowly or not" stretching vortices. For the "super fast" stretching vortices, it is found that they tend to be orthogonal to the vortices of double size. Meanwhile, no particular tendency is found for the "moderately fast" stretching vortices.

8:58AM M21.00007 On the Distribution of Velocity Gradients, Viscosity and Reynolds Stresses in Varied Bed Elevation Turbulent Flow, HANIEH TABKHI, ARASH NAYEBZADEH, University of Central Florida — There is wide variation of depth across a cross section along the axis of the estuaries and coastal regions. In addition to bed turbulence and secondary circulations, structure of turbulence flow at these regions is affected by variation in the depth of the bed. Lateral variations of depth cause strong transverse free shear layers due to steep velocity gradient. Many experimental and laboratory studies have mentioned these shear layers in previous studies. At present study, three dimensional modeling in a Cartesian coordinate system is performed for varied bed elevation flow. Normal and shear Reynolds stresses have been calculated applying k-epsilon and k-omega models. Model is validated by previous related studies and mutual effects of velocity gradients on turbulence viscosity and Reynolds stresses have been investigated. Results show that velocity gradients monotonically increase by increasing magnitude of turbulence viscosity and Reynolds stresses.

9:04AM M21.00008 On the viscosity stratification in temporal mixing layer1, LUMINITA DANAILA, NOUREDDINE TAJUELMIMT, ABDELAH HADJADI, CORIA UMR6614, University of Rouen & INSA Rouen, TURBULENCE TEAM — We assess the effects of viscosity variations in low-speed temporally-evolving turbulent mixing layer. The two streams are density-matched, but the slow fluid is Rv times more viscous than the rapid stream. Direct Numerical Simulations (DNS) are performed for several viscosity ratios, Rv varying between 1 and 9. The space-time evolution of Variable-Viscosity Flow (VVF) is compared with that of the Constant-Viscosity Flow (CVF). The velocity fluctuations occur earlier and are more enhanced for VVF. In particular, the kinetic energy peaks earlier and is up to three times larger for VVF than for CVF at the earliest stages of the flow. Over the first stages of the flow, the temporal growth rate of the fluctuations kinetic energy is exponential, in full agreement with linear stability theory. The transport equation for the fluctuations kinetic energy is favourably compared with simulations data. The enhanced kinetic energy for VVF is mainly due to an increased production at the interface between the two fluids, in tight correlation with enlarged values of mean velocity gradient at the inflection point of the mean velocity profile. The transport equations of the one-and two-point kinetic energy show that self-preservation cannot be complete in variable-viscosity flows.

1ANR is acknowledged for financial support.
8:00AM M22.00001 Realistic simulations of coaxial atomisation. STEPHANE ZALESKI, DANIEL FUSTER, TOMAS ARRUFAT JACKSON, YUE LING, UPMC Univ Paris 06, d’Alember Institute, MATTEO CENNI, UPMC Univ Paris 06, d’Alembert Institute and University of Bologna, RUBEN SCARDOVELLI, University of Bologna, GRETar Tryggvason, University of Notre Dame — We discuss advances in the methodology for Direct Numerical Simulations of coaxial atomization in typical experimental conditions. Such conditions are extremely demanding for the numerical methods. The key difficulty seems to be the combination of high density ratios, surface tension, and large Reynolds numbers. We explore how using a momentum-conserving Volume-Of-Fluid scheme allows to improve the stability and accuracy of the simulations. We show computational evidence that the use of momentum conserving methods allows to reduce the required number of grid points by an order of magnitude in the simple case of a falling rain drop. We then apply these ideas to coaxial atomization. We show that in moderate-size simulations in air-water conditions close to real experiments, instabilities are still present and then discuss ways to fix them. Among those, removing small VOF debris and improving the time-stepping scheme are two important directions. The accuracy of the simulations is then discussed in comparison with experimental results and in particular the angle of ejection of the structures. The code used for this research is free and distributed at http://parissimulator.sf.net.

8:13AM M22.00002 DNS of coflowing planar jet atomization: can one reach convergence?1, YUE LING, STEPHANE ZALESKI, Institut d’Alembert, UPMC-Paris 6, GRETar TRYGGVASON, University of Notre Dame, DANIEL FUSTER, Institut d’Alembert, UPMC-Paris 6, RUBEN SCARDOVELLI, MATTEO CENNI, Universita di Bologna, TOMAS ARRUFAT, Institut d’Alembert, UPMC-Paris 6 — Atomization of a liquid jet assisted by a coflowing fast gas jet is commonly seen in fuel injection systems. Three-dimensional direct numerical simulations are performed to investigate the turbulent multiphase flow characteristics in coflowing planar jet atomization, with the interface tracked by the Volume-of-fluid method. Although many numerical simulations of atomization were reported in the recent years, whether the atomization characteristics such as droplet formation and size distribution are fully resolved is often unclear. In this work, a series of very large-scale simulations of different grid resolution (up to four billion grid points) are conducted and particular attention is focused on examining whether we can achieve converged results on the statistical atomization characteristics. The statistical characteristics of the turbulence (such as turbulence kinetic energy) and of the spray (such as droplet size distribution, liquid volume fraction, and gas-liquid interfacial area) are calculated by averaging the DNS data spatially and temporally. The complex multiscale droplet formation mechanisms due to the interaction between the interface and the turbulence are also revealed by the simulation results.

8:26AM M22.00003 Droplet dynamics in homogeneous isotropic turbulence, DANIEL ALBERNAZ, MINH DO-QUANG, GUSTAV AMBERG, Kungliga Tek Hogscolan KTH — This study investigates the droplet dynamics in homogeneous isotropic turbulence using a lattice Boltzmann model for multiphase flows. The thermodynamics is taken into account with a non-ideal equation of state allowing phase change and by solving a scalar transport energy equation. The system is considered close to the critical point, where a saturated hydrocarbon droplet is surrounded by vapor. The droplet deformation and frequency spectra are analyzed in detail, where the surface tension and local temperature gradients play a major role. The effects of the turbulence intensity and droplet size are also discussed. The droplet behavior under turbulent flows is essential to gain in-depth insight into the different physical phenomena taking place inside sprays and liquid jets.

8:39AM M22.00004 Rain formation via turbulent mixing of droplet distributions, MIHKEL KREE, JAAN KALDA, Institute of Cybernetics, Tallinn University of Technology — It is well known that the growth of water droplets in a cloud due to vapor diffusion alone is insufficiently slow to explain the rapid onset of rain formation. In recent years, there have been several proposals of turbulent mechanisms leading to enhanced collision rates. It has been understood that a broadening of droplet size spectra can provide a sufficient boost to the collision rate. However, the broadening of the droplet size spectra also needs to be explained. Here, we propose a novel approach based on the idea that turbulent mixing brings together droplets of very different histories and hence, of very different sizes, similarly to how passive scalar fronts are formed. We provide relevant analytical estimates, and simulations based on 1D model of turbulence (stochastic triplet map similar to the Baker’s map). This mapping model captures the essential stretching and folding nature of turbulent flows. The triple mapping is accompanied by averaging of neighboring distributions corresponding to local diffusive mixing of droplets. In particular, we study the widths (variances) of local droplet size distributions, which appear to follow a power law. Accordingly, we witness occasional instances of extremely broad droplet size distributions, which can trigger the rain formation.

8:52AM M22.00005 Scalewise investigation of two-phase flow turbulence in upward turbulent bubbly pipe flows1, JUN HO LEE, HYUNSEOK KIM, HYUNGMIN PARK, Seoul National University — In the present study, the two-phase flow turbulence in upward turbulent bubbly pipe flows (at the Reynolds number of 5300) is investigated, especially focusing on the changes in flow structures with bubbles depending on the length scales. For the scalewise investigation, we perform the wavelet multi-resolution analysis on the velocity fields at three streamwise locations, measured with high-speed two-phase particle image velocimetry technology. While we intentionally introduce asymmetrically distributed bubbles at the pipe inlet, the mean volume void fraction is varied from 0.3% to 1.86% and the considered mean bubble diameter is roughly maintained at 3.8 mm. With the present condition, turbulence enhancement is achieved for most cases but the turbulent suppression is also captured near the wall for the smallest void fraction case. Comparing the scalewise energy contribution, it is understood that the flow structures with length scales between bubble radius and bubble wake size are enhanced due to bubbles, resulting in the turbulence enhancement. On the other hand, flow structure with smaller length scales (mostly existing near the wall) may decrease depending on the bubble condition, which may be one of the explanations in turbulence suppression with bubbles.

1Supported by the NRF grant funded by the Korea government (NRF-2012M2A8A4055647) via SNU-1AMD.
9:05AM M22.00006 Bubble-induced turbulence study in homogeneous turbulent flow using DNS approach1. JINYONG FENG, IGOR BOLOTOV, North Carolina State Univ — The effect of a single bubble on the energy transfer to a homogeneous turbulent flow using DNS approach is investigated for various conditions. The single-phase turbulence is numerically generated by pressure-gradient driven uniform flow through a fully resolved turbulence generating grid. The turbulent intensity measured is uniform normal to the flow direction. The decay rate of the turbulent kinetic energy is validated against analytical power law. The collected instantaneous velocity is used as inflow condition for single-bubble simulations to study the bubble-induced turbulence (BIT). In interface-resolved two-phase simulation the bubble is kept at fixed positions by using a proportional-integral-derivative controller. This simulation set allows estimating the turbulent kinetic energy before and after the bubble, quantifying the BIT. Effects of bubble deformability, velocity and turbulent intensity are separately studied. We observe that for a nearly spherical bubble, the bubble-induced turbulence is positive, increasing the level of turbulent kinetic energy in the liquid phase. BIT is influenced by the other studied parameters and the presented work will contribute to the closure BIT model development in multiphase computational fluid dynamics modeling.

1The work is supported by NSF-CBET-Fluid Dynamics, Award #1333993

9:18AM M22.00007 Influence of bubble clusters over the turbulent structure in upward bubbly channel flows; YOSHITO SEIKIGUCHI, WENHAO ZHANG, HIROAKI NAKANISHI, The University of Tokyo, JUN SAKAKIBARA, Meiji University, SHU TAKAGI, The University of Tokyo — We conducted the PIV measurement of upward, turbulent bubbly channel flows. In our experiment, bubbles do not coalesce and become mono-dispersed 1 mm spherical shape due to surfactants in the liquid phase. Adding the surfactant in some specific conditions, these bubbles are attracted toward the wall by the shear induced lift force and form bubble clusters. While they flow near wall, the Reynolds stress of the liquid phase near wall comes close to zero [Takagi, S. and Matsumoto, Y., Annu. Rev. Fluid Mech. (2011)]. This suggests that the turbulent structure change dramatically due to bubble clusters. For the further investigation of the turbulent structure, we constructed the measurement system of Scanning Stereoscopic PIV (SSPIV) which can visualize the three-dimensional velocity field. Using this system, we acquire the velocity field and extracted the large scale vortices which dominate the turbulent structure. Also, we constructed another measurement system for tracking the bubble cluster’s flow. Through the simultaneous measurement of vortices and bubble cluster, we analyze the influence of bubble cluster over the turbulent structure. The results will be discussed in the presentation.

9:31AM M22.00008 Numerical simulations of bubbly Taylor-Couette turbulence in co- and counter rotating regime, VAMS SPANDAN, ROBERTO VERZICCO, DETLEF LOHSE, Physics of Fluids, University of Twente — Two-phase Taylor-Couette (flow between two co-axial independently rotating cylinders) is simulated using a two-way coupled Euler-Lagrange approach in which the bubbles are treated as point particles with effective forces such as drag, lift, added mass and buoyancy acting on them. The momentum equations for the fluid and the bubbles are solved in the frame of reference of the outer cylinder. While it is already known that when the outer cylinder is stationary, within a certain Taylor number range \( (T a \sim 10^6 \sim 10^9) \) the bubbles disrupt the plume ejection regions and the coherent vortical structures leading to drag reduction, their effect and arrangement in the gap-width when both cylinders are rotating is still unknown. In this study we focus on studying the effect of bubbles on the angular velocity transport for various rotation rates of the cylinders. We find that the net percentage drag reduction persists even with a rotating outer cylinder, but is there a optimum for various rotation rates? How does the spatial distribution of bubbles vary with in the co- and counter rotating regime? These are some questions we attempt to answer in this work.

9:44AM M22.00009 Two-dimensional Turbulence in Symmetric Binary-Fluid Mixtures: Coarsening Arrest by the Inverse Cascade, PRASAD PERLEKAR, TIFR Centre for Interdisciplinary Sciences, 21 Brundavan Colony, Narsing, Hyderabad, India, NAIRITA PAL, RAHUL PANDIT, Centre for Condensed Matter Theory, Indian Institute of Science, Bangalore 560012, India — We study two-dimensional (2D) binary-fluid turbulence by carrying out an extensive direct numerical simulation (DNS) of the forced, statistically steady turbulence in the coupled Cahn-Hilliard and Navier-Stokes equations. In the absence of any coupling, we choose parameters that lead (a) to spinodal decomposition and domain growth, which is characterized by the spatiotemporal evolution of the Cahn-Hilliard order parameter \( \phi \), and (b) the formation of an inverse-energy-cascade regime in the energy spectrum \( E(k) \), in which energy cascades towards wave numbers \( k \) that are smaller than the energy-injection scale \( k_{inj} \). In the turbulent fluid. We show that the Cahn-Hilliard-Navier-Stokes coupling leads to an arrest of phase separation at a length scale \( L_c \), which we evaluate from \( S(k) \), the spectrum of the fluctuations of \( \phi \). We demonstrate that (a) \( L_c \sim L_H \), the Hinze scale that follows from balancing inertial and interfacial-tension forces, and (b) \( L_c \) is independent, within error bars, of the diffusivity \( D \). We elucidate how this coupling modifies \( E(k) \) by blocking the inverse energy cascade at a wavenumber \( k_c \), which we show is \( \approx 2\pi/L_c \). We compare our work with earlier studies of this problem.

9:57AM M22.00010 Suppression of turbulent energy cascade due to phase separation in homogenous binary mixture fluid1. YOUEI TAKAGI, SACHIYA OKAMOTO, Osaka University — When a multi-component fluid mixture becomes thermophysically unstable state by quenching from well-melting condition, phase separation due to spinodal decomposition occurs, and a self-organized structure is formed. During phase separation, free energy is consumed for the structure formation. In our previous report, the phase separation in homogenous mixture fluid was numerically simulated and the coarsening process of phase separation was discussed. In this study, we extended our numerical model to a high Schmidt number fluid corresponding to actual polymer solution. The governing equations were continuity, Navier-Stokes, and Cahn-Hilliard equations as same as our previous report. The flow field was an isotropic homogenous turbulence, and the dimensionless parameters in the Cahn-Hilliard equation were estimated based on the thermophysical condition of binary mixture. From the numerical results, it was found that turbulent energy cascade was drastically suppressed in the inertial subrange by phase separation for the high Schmidt number fluid. By using the identification of turbulent and phase separation structure, we discussed the relation between total energy balance and the structures formation processes.

1This study is financially supported by the Grand-in-Aid for Young Scientists (B) (No. T26820045) from the Ministry of Education, Culture, Sports, Science and Technology of Japan.

Tuesday, November 24, 2015 8:00AM - 10:10AM –
Session M23 Biofluids: Undulatory Swimming in Newtonian and Non-Newtonian Fluids 300 - Arezoo Ardekani, Purdue University
Undulatory swimming in non-Newtonian fluids. Arezoo Ardekan, Gaojin Li, Purdue University — Microorganisms often swim in complex fluids exhibiting both elasticity and shear-thinning viscosity. The motion of low Reynolds number swimmers in complex fluids is important for better understanding the migration of sperms and formation of bacterial biofilms. In this work, we numerically investigate the effects of non-Newtonian fluid properties, including shear-thinning and elasticity, on the undulatory locomotion. Our results show that elasticity hinders the swimming speed, but a shear-thinning viscosity in the absence of elasticity enhances the speed. The combination of the two effects hinders the swimming speed. The swimming boost in a shear-thinning fluid occurs even for an infinitely long flagellum. The swimming speed has a maximum, whose value depends on the flagellum oscillation amplitude and fluid rheological properties. The power consumption, on the other hand, follows a universal scaling law.

1This work is supported by NSF CBET-1445955 and Indiana CTSI TR001108.

Undulatory swimming in shear-thinning fluids: Experiments with Caenorhabditis elegans. David Gagnon, Paulo Arratia, University of Pennsylvania — The swimming behavior of microorganisms can be strongly affected by the rheology of their fluidic environment. In this talk, we experimentally investigate the swimming behavior of the nematode Caenorhabditis elegans (≈1 mm length, 80 µm diameter) in shear-thinning fluids using tracking and velocimetry methods. We find substantial differences in the resulting flow fields between the shear-thinning and Newtonian cases, even though the swimming kinematics (e.g., speed and frequency) remain similar. For example, velocimetry data show that shear-thinning viscosity enhances vorticity and increases circulation near the strongest body vortex, located near the head of the nematode. These findings are in good agreement with recent theoretical and numerical results. We then estimate the local viscosity around the swimmer, measure the spatial decay of the flow field, and estimate the mechanical power (i.e., viscous dissipation) due to the worm’s motion in shear-thinning fluids. We find that the flow decays more slowly in shear-thinning fluids than in Newtonian fluids, but the resulting mechanical power is approximately the same for swimming in shear-thinning fluids when compared to the Newtonian case.

Locomotion in a liquid crystal near a wall. Thomas Powers, Madison Krieger, Brown Univ, Saverio Spagnolie, Univ of Wisconsin, Madison — Recent observations of bacteria swimming in nematic liquid crystal solution motivate the theoretical study of how swimming speed depends on liquid crystal properties. We consider the Taylor sheet near a wall, in which propulsion is achieved by the propagation of traveling waves along the length of the swimmer. Using the lubrication approximation, we determine how swimming speed depends on the Ericksen number, which is the ratio of elastic to viscous stresses. We also study the effect of anchoring strength, at the surface of the swimmer and the surface of the wall.

A Simple Method to Measure Nematodes’ Propulsive Thrust and the Nematode Ratchet. Haim Baú, Jinhzhou Yuan, David Raizen, University of Pennsylvania — Since the propulsive thrust of microorganisms provides a more sensitive indicator of the animal’s health and response to drugs than motility, a simple, high-throughput, direct measurement of the thrust is desired. Taking advantage of the nematode C. elegans being heavier than water, we devised a simple method to determine the propulsive thrust of the animals by monitoring their velocity when swimming along an inclined plane. We find that the swimming velocity is a linear function of the sine of the inclination angle. This method allows us to determine, among other things, the animals’ propulsive thrust as a function of genotype, drugs, and age. Furthermore, taking advantage of the animals’ inability to swim over a still incline, we constructed a sawteeth ratchet-like track that restricts the animals to swim in a predetermined direction.

Flow analysis of C. elegans swimming. Thomas Montenegro-Johnson, University of Cambridge, David Gagnon, Paulo Arratia, University of Pennsylvania, Eric Lauga, University of Cambridge — Improved understanding of microscopic swimming has the potential to impact numerous biomedical and industrial processes. A crucial means of analyzing these systems is through experimental observation of flow fields, from which it is important to be able to accurately deduce swimmer physics such as power consumption, drag forces, and efficiency. We examine the swimming of the nematode worm C. elegans, a model system for understanding micro-propulsion. Using experimental data of swimmer geometry and kinematics, we employ the regularized stokeslet boundary element method to simulate the swimming of this worm outside the regime of slender-body theory. Simulated flow fields are then compared with experimentally extracted values confined to the swimmer beat plane, demonstrating good agreement. We finally address the question of how to estimate three-dimensional flow information from two-dimensional measurements.

Maneuverability and chemotaxis of Caenorhabditis elegans in three-dimensional environments. Jerzy Blawzdziewicz, Alejandro Bilbao, Amar Patel, Siva Vanapalli, Texas Tech University — Locomotion of the nematode C. elegans in water and complex fluids has recently been investigated to gain insight into neuromuscular control of locomotion and to shed light on nematode evolutionary adaptation to environments with varying mechanical properties. Previous studies focused mainly on locomotion efficiency and on adaptation of the nematode gait to the surrounding medium. Much less attention has been devoted to nematode maneuverability, in spite of its crucial role in the survival of the animal. Recently [Phys. Fluids 25, 081902 (2013)] we have provided a quantitative analysis of turning maneuvers of crawling and swimming nematodes on flat surfaces and in 2D fluid layers. Based on this work, we follow with the first full 3D description of how C. elegans moves in complex 3D environments. We show that by superposing body twist and 2D undulations, a burrowing or swimming nematode can rotate the undulation plane and change the direction of motion within that plane by varying undulation-wave parameters. A combination of these corkscrew maneuvers and 2D turns allows the nematode to explore 3D space. We conclude by analyzing 3D chemotaxis of nematodes burrowing in gel and swimming in water, which demonstrates an important application of our maneuverability model.

Amplitude transitions of swimmers and flexors in viscoelastic fluids. Robert Guy, Becca Thomas, University of California Davis — In both theoretical and experimental studies of the effect of fluid elasticity on micro-organism swimming, very different behavior has been observed for small and large amplitude strokes. We present simulations of an undulatory swimmer in an Oldroyd-B fluid and show that the resulting viscoelastic stresses are a nonlinear function of the amplitude. Specifically, there appears to be an amplitude dependent transition that is key to obtaining a speed-up over the Newtonian swimming speed. To understand the physical mechanism of the transition, we examine the stresses in a time-symmetric oscillatory bending beam, or flexor. We compare the flow in a neighborhood of the flexor tips with a large-amplitude oscillatory extensional flow, and we see similar amplitude dependent transitions. We relate these transitions to observed speed-ups in viscoelastic swimmers.

This work was supported by NSF grant CBET-1059745.
9:31AM M23.00008 Flexibility, stroke, and dimensionless parameters: the importance of telling the whole story for swimming micro-organisms in complex fluids. BECCA THOMASES, ROBERT GUY, University of California, Davis — The question of how fluid elasticity affects the swimming performance of micro-organisms is complicated and has been the subject of many recent experimental and theoretical studies. The Deborah number, $De = \frac{\lambda_0}{\gamma}$, is typically used to characterize the strength of the fluid elasticity in these studies, and for swimmers is expressed as the product of the elastic relaxation time and the frequency of the swimmer stroke. In simulations of undulatory flexible swimmers in an Oldroyd-B-type fluid, we find that varying the frequency of the stroke and varying the relaxation time separately results in a significantly different dependence of swimming speed for the same $De$. Thus the elastic effects on swimming cannot be characterized by a single dimensionless number. The Weissenberg number, defined as the product of elastic relaxation time and characteristic strain rate ($Wi = \frac{\lambda_0}{\gamma}$), is another dimensionless parameter useful for describing complex fluids. For a fixed swimmer frequency, varying the relaxation time will also vary the Weissenberg number. We conjecture that the different behavior is a consequence of a Weissenberg-number transition in the fluid, which additionally depends on the amplitude of the swimmer stroke.

9:44AM M23.00009 Swimming sheet in a Newtonian fluid confined by a Brinkman medium. SEYED AMIR MIRBAGHERI, HENRY FU, University of Nevada (Reno) — Many microorganisms swim through complex materials such as viscoelastic mucus in their natural habitats. As microorganisms move through complex materials, they may induce spatial heterogeneity in the medium, which can affect swimming properties. For example, the rotating flagella of bacteria may deplete polymer concentration near the flagella, while H pylori can turn nearby mucin gel into sol by elevating the pH. Here we examine a simple model of swimming in such scenarios, by investigating Taylor’s two-dimensional swimming sheet swimming in a layer of Newtonian fluid. The Newtonian fluid is bounded above by a Brinkman medium, which represents the complex material that has been locally depleted or dissolved near the swimmer. We analytically derive the velocity for a small amplitude wave of an infinite sheet using a perturbation series to second order in the wave amplitude. For a fixed swimmer geometry, we explore the dependence of the velocity on the thickness of the Newtonian fluid and the permeability and porosity of the Brinkman medium.

9:57AM M23.00010 Swimming Speeds of Filaments in Viscous Fluids with Resistance. NGUYENHO HO, SARAH OLSON, Worcester Polytechnic Institute — Spermatozoa and bacteria can utilize lateral and spiral bending waves to propagate in a fluid. Often, they encounter different fluid environments filled with mucus, cells, hormones, and other large proteins. These extra materials act as friction, possibly preventing or enhancing forward progression of swimmers. To understand these effects, we employ Taylor’s techniques to calculate the asymptotic swimming speeds of a cylinder of infinite extent in a viscous fluid with resistance known as a Brinkman fluid. We find that, up to the second order expansion, the swimming speeds are enhanced as resistance increases. The Stokes limit can also be also recovered from this result as resistance goes to zero. In addition, we show numerical results for a Lagrangian algorithm of a rod waving in a porous medium and compare numerical results to asymptotic swimming speeds.

Tuesday, November 24, 2015 8:00AM - 10:10AM – Session M24 Biofluids: Cardiovascular Disease II 302 - Zahra Keshavarz-Motamed, MIT

8:00AM M24.00001 Fluid dynamics of coarctation of the aorta: analytical solution, in vitro validation and in vivo evaluation. ZAHRA KESHAVARZ-MOTAMED, Institute for Medical Engineering and Science, Massachusetts Institute of Technology; Harvard-MIT Division of Health Sciences and Technology — Coarctation of the aorta (COA) is a congenital heart disease corresponding to a narrowing in the aorta. Cardiac catheterization is considered to be the reference standard for definitive evaluation of COA severity, based on the peak-to-peak trans-coarctation pressure gradient (PtoP TCPG) and instantaneous systolic value of trans-CAO pressure gradient (TCPG). However, invasive cardiac catheterization may carry high risks given that undergoing multiple follow-up cardiac catheterizations in patients with COA is common. The objective of this study is to present an analytical description of the COA that estimates PtoP TCPG and TCPG without a need for high risk invasive data collection. Coupled Navier-Stokes and elastic deformation equations were solved analytically to estimate TCPG and PtoP TCPG. The results were validated against data measured in vitro (e.g., 90% COA: TCPG: root mean squared error (RMSE)= 3.93 mmHg; PtoP TCPG: RMSE= 7.9 mmHg). Moreover, the estimated PtoP TCPG resulted from the suggested analytical description was validated using clinical data in twenty patients with COA (maximum RMSE: 8.3 mmHg). Very good correlation and concordance were found between TCPG and PtoP TCPG obtained from the analytical formulation and in vitro and in vivo data. The suggested methodology can be considered as an alternative to cardiac catheterization and can help preventing its risks.

8:13AM M24.00002 Optimization of the assisted bidirectional Glenn for single ventricle palliation. ALISON MARSDEN, JESSICA SHANG, MAHDI ESMAILY-MOGHADAM, Stanford University, RICHARD FIGLIOLA, Clemson University, OLAF REINHARTZ, Stanford University, TAIN-YEN HSIA, University College London — For neonates with single ventricle physiology, a systemic-pulmonary shunt (e.g., a modified Blalock-Taussig shunt (mBTS)) is typically employed as an early-stage procedure in preparation for a later-stage bidirectional Glenn (BDG). Mortality rates with the mBTS are high, yet the BDG has poorer outcomes in neonates. The assisted bidirectional Glenn (ABG) augments the inadequate pulmonary flow associated with early BDG implementation in neonates through an additional shunt between the innominate artery and the superior vena cava (SVC). The shunt uses a nozzle to inject high-velocity flow to the SVC, elevating downstream pulmonary pressure. Previous simulations and animal studies verified feasibility and higher pulmonary flow rates. In numerical simulations, we explore shunt geometries and placements implanted into a 3D model of the aorta and pulmonary arteries, coupled with a lumped parameter network describing the remaining circulatory system. We seek an ABG shunt that optimizes hemodynamic variables such as pulmonary flow rate and oxygenation and constrains SVC pressure. The optimized ABG will be evaluated against the mBTS and the BDG in simulations and experiments. A successful implementation of the ABG would replace the mBTS and BDG procedures and reduce mortality rates.

1 Burroughs Wellcome Fund, Leducq Foundation
8:26AM M24.00003 Computational fluid dynamics study of commercially available stents inside an idealised curved coronary artery , WINSON XIAO CHEN, ANDREW OOL, NICHOLAS HUTCHINS, ERIC POON, VIKAS THONDAPU, University of Melbourne, Australia; PETER BARLIS, Northern Health, Australia — Stent placement restores blood flow in diseased coronary arteries and is the standard treatment for obstructive coronary atherosclerosis. Analysis of the hemodynamic characteristics of stented arteries is essential for better understanding of the relationship between key fluid dynamic variables and stent designs. Previous computational studies have been limited to idealised stents in curved arterial segments or more realistic stents in straight segments. In clinical practice, however, it is often necessary to place stents in geometrically complex arterial curvatures. Thus, numerical simulations of the incompressible Navier–Stokes equations are carried out to investigate the effects of curvature on hemodynamics using detailed, commercially available coronary stents. The computational domain is a 3mm curved coronary artery model and simulations are conducted using a physiologically realistic inlet condition. The averaged flow rate is about 80 ml/min, similar to the normal human resting condition. The examination of hemodynamic parameters will assess the performance of several commercially available stents in curved arteries and identify regions that may be at risk for restenosis. It is anticipated that this information will lead to improvements in future stent design and deployment.

8:39AM M24.00004 Lagrangian coherent structures and turbulence characteristics downstream of prosthetic aortic valves , MARCO D. DE TULLIO, Politecnico di Bari — The flowfield through prosthetic heart valves is investigated by means of direct numerical simulations, considering the fully coupled fluid-structure interaction problem. Two different aortic valve models are modeled: a bileaflet mechanical and a biological one. In order to reveal fluid flow structures and to better understand the transport mechanisms, Lagrangian coherent structures (LCS) are used. LCS are distinguished material surfaces that can be identified as boundaries to regions with dynamically distinct behavior, and are revealed as hypersurfaces that locally maximize the finite-time Lyapunov exponent (FTLE) fields. Post-processing the flow simulation data, first FTLE fields are calculated integrating dense meshes of Lagrangian particles backward in time, and then attracting LCS are extracted. A three-jet configuration is distinctive of bi-leaflet mechanical valves, with higher turbulent shear stresses immediately distal to the valve leaflets, while a jet-like flow emerges from the central orifice of bio-prosthetic valves, with high turbulent shear stresses occurring at the edge of the jet. Details of the numerical methodology along with a thorough analysis of the different flow structures developing during the cardiac cycle for the two configurations will be provided.

8:52AM M24.00005 Experimental Comparison of the Hemodynamic Effects of Bi-furcating Coronary Stent Implantation Techniques , MELISSA BRINDISE, PAVLOS VLACHOS, Purdue Univ, AETHER LAB TEAM — Stent implantation in coronary bifurcations imposes unique effects to the blood flow patterns and currently there is no universally accepted stent deployment approach. Despite the fact that stent-induced changes can greatly alter clinical outcomes, no concrete understanding exists regarding the hemodynamic effects of each implantation method. This work presents an experimental evaluation of the hemodynamic differences between implantation techniques. We used four common stent implantation methods including the currently preferred one-stent provisional side branch (PSB) technique and the crush (CRU), Culotte (CUL), and T-stenting (T-PR) two-stent techniques, all deployed by a cardiologist in coronary models. Particle image velocimetry was used to obtain velocity and pressure fields. Wall shear stress (WSS), oscillatory shear index, residence times, and drag and compliance metrics were evaluated and compared against an un-stented case. The results of this study demonstrate that while PSB is preferred, both it and T-PR yielded detrimental hemodynamic effects such as low WSS values. CRU provided polarizing and unbalanced results. CUL demonstrated a symmetric flow field, balanced WSS distribution, and ultimately the most favorable hemodynamic environment.

9:05AM M24.00006 Numerical modeling of the fetal blood flow in the placental circulatory system , ALEXANDER SHANNON, SERGIO GALLUCCI, PARISA MIRBOD, Clarkson University — The placenta is a unique organ of exchange between the growing fetus and the mother. It incorporates almost all functions of the adult body, acting as the fetal lung, digestive and immune systems, to mention a few. The exchange of oxygen and nutrients takes place at the surface of the villous tree. In this study we investigate the geometry of the fetal villous tree, and a 3D computational domain was established to allow for the unsteady fetal blood flow, gas, and nutrient transport over the chorionic plate. The fetal blood was treated as an incompressible Newtonian fluid, and the oxygen and nutrient were treated as a passive scalar dissolved in blood plasma. The flow was laminar, and a commercial CFD code (COMSOL Multiphysics) has been used for the simulation. COMSOL has been selected because it is multi-physics FEM software that allows for the seamless coupling of different physics represented by partial differential equations. The results clearly illustrate that the specific branching pattern and the in-plane curvature of the fetal villous trees affect the delivery of blood, gas and nutrient transport to the whole placenta.

9:18AM M24.00007 Hemodynamics in an Aorta with Bicuspid and Trileaflet Valves† , ANVAR GILMANOV, FOTIS SOTIROPOULOS, Univ of Minn - Minneapolis — Bicuspid aortic valve (BAV) is a congenital heart defect that has been associated with serious aortic congenital, aortic stenosis, aortic regurgitation, infective endocarditis, aortic dissection, calcific aortic valve and dilatation of ascending aorta. Two main hypotheses - the genetic and the hemodynamic are discussed in literature to explain the development and progression of aortopathies in patients with BAV. In this study we seek to investigate the possible role of hemodynamic factors as causes of BAV-associated aortopathy. We employ the Curvilinear Immersed Boundary (CURVIB) method coupled with an efficient thin-shell finite element (TS-FE) formulation for tissues to carry out fluid-structure interaction simulations of a healthy tri-leaflet aortic valve (TAV) and a BAV placed in the same anatomic aorta. The computed results reveal major differences between the TAV and BAV flow patterns. These include: the dynamics of the aortic valve vortex ring formation and break up; the large scale flow patterns in the ascending aorta; and the shear stress magnitude on the aortic wall. The computed results are in qualitative agreement with in vivo Magnetic Resonance Imaging (MRI) data and suggest that the linkages between BAV aortopathy and hemodynamics deserve further investigation.

†This work is supported by the Lillehei Heart Institute at the University of Minnesota and the Minnesota Supercomputing Institute.

9:31AM M24.00008 A numerical investigation of a simplified human birth model , ROSEANNA PEALATER, Tulane University, ALEXA BAUMER, George Washington University, LISA FAUCI, Tulane University, MEGAN C. LEFTWICH, George Washington University — This work uses simplified models and numerical computations to explore the effects of both the fetal velocity and the viscosity of the surrounding fluid on the forces associated with human birth. The numerical results are compared with the results of an experimental model representing the fetus moving through the birth canal using a rigid cylinder (fetus) that moves at a constant velocity through the center of a passive elastic tube (birth canal). The entire system is immersed in highly viscous fluid. Due to low Reynolds number, the Stokes equations can be used to describe the relationship between velocity and forces in the system. The mathematical model uses the method of regularized Stokeslets to estimate the pulling force necessary to move the rigid inner cylinder at a constant velocity. The elastic tube through which the rigid cylinder passes is constructed by a discrete network of Hookean springs, with macroscopic elasticity matched to the tube used in the physical experiment. More complex geometries as well as peristaltic activation of the elastic tube can be added to the model to provide more insight into the relationship between force and velocity during human birth.
9:44AM M24.00009 Tubular Heart Pumping Mechanisms in Ciona intestinalis
NICHOLAS BATTISTA, LAURA MILLER, University of North Carolina at Chapel Hill — In vertebrate embryogenesis, the first organ to form is the heart, beginning as a primitive heart tube. However, many invertebrates have tubular hearts from infancy through adulthood. Heart tubes have been described as peristaltic and impedance pumps. Impedance pumping assumes a single actuation point of contraction, while traditional peristalsis assumes a traveling wave of actuation. In addition to differences in flow, this inherently implies differences in the conduction system. It is possible to transition from pumping mechanism to the other with a change in the diffusivity of the action potential. In this work we consider the coupling between the fluid dynamics and electrophysiology of both mechanisms, within a basal chordate, the tunicate. Using CFD with a neuro-mechanical model of tubular pumping, we discuss implications of the both mechanisms. Furthermore, we discuss the implications of the pumping mechanism on evolution and development.

9:57AM M24.00010 Effect of Trabeculae on the Hemodynamics of an Embryonic Left Ventricle1, VIJAY VEDULA, Stanford University, JUHYUN LEE, TZUNG HSIAI, University of California, Los Angeles, ALISON MARSDEN, Stanford University — The left ventricular (LV) endocardium is not smooth, but has “trabeculae” protruding into the LV cavity. Recent studies have indicated that trabeculae significantly influence LV hemodynamics by enhancing the diastolic penetration depth of inflow and facilitating a better apical systolic washout. However, it remains unclear how the role of hemodynamics modulates the initiation of trabeculae during cardiac morphogenesis. While such an assessment of mammalian heart models is hampered by the prolonged duration of cardiac development and complexity of surrounding internal organs, embryonic zebrafish is a genetically tractable model for investigating cardiac morphogenesis. We employ a novel light-sheet fluorescent microscopy to extract 4D LV models of zebrafish and develop an ALE-based moving domain CFD solver to perform flow simulations and extract quantitative data related to flow velocities and pressure gradients. We will compare near-wall flow dynamics between the wild type zebrafish (with trabeculae) and the cloche mutant lines that fail to develop trabeculae, to provide new insights into the flow-induced mechano-transduction relevant to the initiation of trabeculae during cardiac morphogenesis.

Tuesday, November 24, 2015 8:00AM - 9:57AM — Session M25 Biofluids: Transport and Control 304 - Amy Gao, MIT

8:00AM M25.00001 Modeling Self-Induced Effects is Important for Flow-Relative Control, AMY GAO, MICHAEL TRIANTAFYLLIOU, Massachusetts Inst of Tech-MIT — For aquatic animals, self-generated stimulation has the potential to mask signals from external sources. Fish, which sense their near field using their lateral lines, have developed passive and active means of subtracting the flow signals generated by their self motions, which not only mask biologically relevant stimuli, but also render control deficient. While prior work in this field estimates the orientation of a vehicle as a linear function of the difference in pressure between opposite sides, we demonstrate that a high performance controller cannot use simply this linear relationship, because in a hydrodynamic environment, the external flow and the self-induced flow combine in a nonlinear way. A misinterpretation of the hydrodynamic interactions due to simplistic signal manipulation can be catastrophic, leading to instability or collision. Overall, we demonstrate the importance of model-based control in the underwater environment, and propose a robust controller which uses flow-relative feedback from a sensor inspired by the lateral line of fish.

8:13AM M25.00002 Optimal Sensor Layouts in Underwater Locomotory Systems, BRENDAN COLVERT, EVA KANSO, Univ of Southern California — Retrieving and understanding global flow characteristics from local sensory measurements is a challenging but extremely relevant problem in fields such as defense, robotics, and biomimetics. It is an inverse problem in that the goal is to translate local information into global flow properties. In this talk we present techniques for optimization of sensory layouts within the context of an idealized underwater locomotory system. Using techniques from fluid mechanics and control theory, we show that, under certain conditions, precision measurements can inform the submerged body about its orientation relative to the ambient flow, and allow it to recognize local properties of shear flows. We conclude by commenting on the relevance of these findings to underwater navigation in engineered systems and live organisms.

8:26AM M25.00003 Sinusoidal Forcing of Interfacial Films1, FAYAZ RASHEED, ADITYA RAGHU-NANDAN, Rensselaer Polytechnic Institute, AMIR HIRSA, Rensselaer Polytechnic Institute, JUAN LOPEZ, Arizona State University — Fluid transport, in vivo, is accomplished via pumping mechanisms of the heart and lungs, which results in biological fluids being subjected to oscillatory shear. Flow is known to influence biological macromolecules, but predicting the effect of shear is incomplete without also accounting for the influence of complex interfaces ubiquitous throughout the body. Here, we investigated the oscillatory response of the structure of aqueous interfacial films using a cylindrical knife edge viscometer. Vitamin K, was used as a model monolayer because its behaviour has been thoroughly quantified and it doesn’t show any measurable hysteresis. The monolayer was subjected to sinusoidal forcing under varied conditions of surface concentrations, periodic frequencies, and knife edge amplitudes. Particle Image Velocimetry (PIV) data was collected using Brewster Angle Microscopy (BAM), revealing the influence of oscillatory interfacial shear stress on the monolayer. Insights were gained as to how the velocity profile dampens at specific distances from the knife edge contact depending on the amplitude, frequency, and concentration of Vitamin K.1

1Supported by NNX13AQ22G, National Aeronautics and Space Administration

8:39AM M25.00004 Why are there no short circuits in the arterial network?, SHEY-SHEA CHANG, SHENYINYING TU, Department of Mathematics, University of California Los Angeles, YU-HSIU LIU, Institute of Zoology, National Taiwan University, VAN SAVAGE, Department of Biomatics, University of California Los Angeles, SHENG-PING HWANG, Institute of Cellular and Organismic Biology, Academia Sinica, MARCUS ROPER, Department of Mathematics, University of California Los Angeles — Efficient transport within vascular networks requires red blood cells be delivered at the same rate to each capillary, to ensure even oxygen supply throughout an organism. However, real vascular systems are massive networks in which distance from the heart to capillary vessels can vary over several orders of magnitude. Why are there no short-circuits? Why don’t capillaries closer to the heart receive more red blood cells than farther capillaries? We used the trunk arterial network of a zebrafish embryo as a model for understanding the mechanisms underlying red blood cell partitioning within the microvasculature. Using mathematical modeling and experiments in living zebrafish we show that a tuned hydrodynamic feedback mechanism evenly splits red blood cells between trunk vessels. This key design feature comes at a cost to the overall efficiency of the network in that creating a uniform flux means that many red blood cells no longer travel through capillaries.
8:52AM M25.00005 Modeling and Simulation of Cardiogenic Embolic Particle Transport to the Brain1. DEBANJAN MUKHERJEE, NEEL JANI, SHAWN C. SHADDEN, University of California, Berkeley — Emboli are aggregates of cells, proteins, or fatty material, which travel along arteries distal to the point of their origin, and can potentially block blood flow to the brain, causing stroke. This is a prominent mechanism of stroke, accounting for about a third of all cases, with the heart being a prominent source of these emboli. This work presents our investigations towards developing numerical simulation frameworks for modeling the transport of embolic particles originating from the heart along the major arteries supplying the brain. The simulations are based on combining discrete particle method with image based computational fluid dynamics. Simulations of unsteady, pulsatile hemodynamics, and embolic particle transport within patient-specific geometries, with physiological boundary conditions, are presented. The analysis is focused on elucidating the distribution of particles, transport of particles in the head across the major cerebral arteries connected at the Circle of Willis, the role of hemodynamic variables on the particle trajectories, and the effect of considering one-way vs. two-way coupling methods for the particle-fluid momentum exchange. These investigations are aimed at advancing our understanding of embolic stroke using computational fluid dynamics techniques.

1This research was supported by the American Heart Association grant titled “Embolic Stroke: Anatomic and Physiologic Insights from Image-Based CFD.”

9:05AM M25.00006 Three-dimensional simulation of flow-induced platelet activation in artificial heart valves. MOHAMMADALI HEDAYAT, HAFEZ ASGHARZADEH, IMAN BORAZJANI, University at Buffalo SUNY — Since the advent of heart valve, several valve types as such as mechanical and bio-prosthetic valves have been designed. Mechanical Heart Valves (MHV) are durable but suffer from thromboembolic complications that caused by shear-induced platelet activation near the valve region. Bio-prosthetic Heart Valves (BHV) are known for better hemodynamics. However, they usually have a short average life time. Realistic simulations of heart valves with combination with platelet activation models can lead to a better understanding of the potential risk of thrombus formation in such devices. In this study, an aneurysm approach is developed to calculate the platelet activation in three-dimensional simulations of flow through MHV and BHV using a parallel overerset-curvilinear immersed boundary technique. A curvilinear body-fitted grid is used for the flow simulation through the anatomic aorta, while the sharp-interface immersed boundary method is used for simulation of the Left Ventricle (LV) with prescribed motion. In addition, dynamics of valves were calculated numerically using under-relaxed strong-coupling algorithm. Finally, the platelet activation results for BMV and MHV are compared with each other.

9:18AM M25.00007 Platelets aggregation in pathological conditions: role of local shear rates and platelet activation delay time. HE LI, ALIREZA ZARIF KHALILI YAZDANI, GEORGE KARNIADAKIS, Brown Univ — Platelets play an essential role in the initiation and formation of a thrombus, however their detailed motion in blood vessels with complex geometries, such as in the aneurysmal vessel or stenotic vessel in atherosclerosis, has not been studied systematically. Here, we perform spectral element simulations (NEKTAR code) to obtain the 3D flow field in blood vessel with cavities, and we apply the force coupling method (FCM) to simulate the motion of platelets in blood flow. Specifically, simulations of platelets are performed in a 0.25 mm diameter circular blood vessel with 1 mm length. Corresponding coarse-grained molecular dynamics simulations are employed to provide input to the NEKTAR-FCM code. Simulations are conducted at several different Reynolds numbers (Re). An ellipsoid-shaped cavity is selected to intersect with the middle part of the circular vessel to represent the aneurysmal part of the blood vessel. Based on the simulation results, we quantify how the platelets motion and aggregation in the blood vessel cavities depend on Re, platelet activation delay time, and the geometry of the cavities.

9:31AM M25.00008 A simple numerical model for membrane oxygenation of an artificial lung machine. SAI NIKHIL SUBRAVETI, P.S.T. SAI, VINOD KUMAR VISWANATHAN PILLAI, B.S.V. PATNAIK, Indian Inst of Tech-Madras — Optimal design of membrane oxygenators will have far reaching ramifications in the development of artificial heart-lung systems. In the present CFD study, we simulate the gas exchange between the venous blood and air that passes through the hollow fiber membranes on a benchmark device. The gas exchange between the tube side fluid and the shell side venous liquid is modeled by solving mass, momentum conservation equations. The fiber bundle was modelled as a porous block with a bundle porosity of 0.6. The resistance offered by the fiber bundle was estimated by the standard Ergun correlation. The present numerical simulations are validated against available benchmark data. The effect of bundle porosity, bundle size, Reynolds number, non-Newtonian constitutive relation, upstream velocity distribution etc. on the pressure drop, oxygen saturation levels etc. are investigated. To emulate the features of gas transfer past the alveoli, the effect of pulsatility on the membrane oxygenation is also investigated.

9:44AM M25.00009 Three-dimensional flow and vorticity transport in idealized airway model from laminar to turbulent regimes. SAHAR JALAL, TRISTAN VAN DE MOORTELE, ANDRAS NEMES, University of Minnesota, AZAR ESLAM PANABA, Pennsylvania State University, FILIPPO COLETTI, University of Minnesota — The presence and intensity of secondary flows formed by the inhaled air during respiration has important consequences for gas exchange and particle transport in the lungs. Here we focus on formation and persistence of such secondary flows by experimentally studying the steady inspiration in an idealized airway model. The geometry consists of a symmetric planar double bifurcation that respects the geometrical proportions of the human bronchial tree. Physiologically relevant Reynolds numbers from 100 to 5000 are investigated, ranging from laminar to turbulence. The time-averaged, the dimensional velocity fields are obtained from Magnetic Resonance Imaging (MRI), providing detailed distributions of vorticity, circulation, and secondary flow strength. Information on the velocity fluctuations are obtained by Particle Image Velocimetry (PIV). The measurements highlight the effect of the Reynolds number on the momentum transport, flow partitioning at the bifurcations, strength and sense of rotation of the longitudinal vortices. A marked change in topology is found at a specific Reynolds number, above which the influence of the upstream flow prevails over the effect of the local geometry. Finally, turbulence and its role in the mean vorticity transport are also discussed.

Tuesday, November 24, 2015 8:00AM - 9:57AM —
Session M26 Biofluids: Medical Devices
306 — Jeremy Marston, Texas Tech University

8:00AM M26.00001 Hydrodynamics of jets in needle-free injections1. JEREMY MARSTON, MOMENE MORADI, Texas Tech University — We present results from an experimental study of jets used in needle-free injections. Ultra-high-speed imaging at frame rates over 300 kfps was used to study the jet formation time, initial contact stage and penetration depth evolution when fired into gel substrates. Both commercial devices using gas and spring mechanisms and custom-made devices were tested, exhibiting some key differences. We also explored a range of liquid physical properties, in particular viscosity, in order to quantitatively explore the parameter space for this intriguing process.

1Supported by Bioject.
8:13AM M26.00002 A Computational and Mathematical Model for Device Induced Thrombosis, WEI-TAO WU, Carnegie Mellon University, NADINE AUBRY, Northeastern University, MEHRDAD MASSOUDI, National Energy Technology Laboratory, JAMES ANTAKI, Carnegie Mellon University — Based on the Sorenson’s model of thrombus formation[1, 2], a new mathematical model describing the process of thrombus growth is developed. In this model the blood is treated as a Newtonian fluid, and transport and reactions of the chemical and biological species are modeled using CRD (convection-reaction-diffusion) equations. A computational fluid dynamic (CFD) solver for the mathematical model is developed using the libraries of OpenFOAM. Applying the CFD solver, several representative benchmark problems are studied: rapid thrombus growth in vivo by injecting Adenosine diphosphate (ADP) using iontophoretic method and thrombus growth in rectangular microchannel with a crevice which usually appears as a joint between components of devices and often becomes nidus of thrombosis. Very good agreements between the numerical and the experimental results validate the model and indicate its potential to study a host of complex and practical problems in the future, such as thrombosis in blood pumps and artificial lungs. 1. Sorensen, E.N., et al., Computational simulation of platelet deposition and activation: I. Model development and properties. Ann Biomed Eng, 1999. 27(4): p. 436-48. 2. Sorensen, E.N., et al., Computational simulation of platelet deposition and activation: II. Results for Poiseuille flow over collagen. Ann Biomed Eng, 1999. 27(4): p. 449-56.

8:26AM M26.00003 Selective control for helical microswimmers, PANAYIOTA KATSAMBA, ERIC LAUGA, University of Cambridge — One of the greatest aspirations for artificial microswimmers is their application in non-invasive medicine. For any practical use, adequate mechanisms enabling control of multiple artificial swimmers is of paramount importance. Here we propose a multi-helical, freely-jointed motor as a novel selective control mechanism. We show that the nonlinear step-out behavior of a magnetized helix driven by a rotating magnetic field can be exploited, when used in conjunction with helices, to obtain a velocity profile that is non-negligible only within a chosen interval of operating frequencies. Specifically, the force balance between the competing opposite-handed helices is tuned to give the net motion of high frequency frequencies. Because of the swimming velocity is monotonically with the driving frequency if two opposite helices are used, thereby allowing speed adjustment by varying the driving frequency. We illustrate this idea in detail on a two-helix system, and demonstrate how to generalize to N helices, both numerically and theoretically. We finish by explaining how to solve the inverse problem and design an artificial swimmer with an arbitrarily-complex velocity vs. frequency relationship.

8:39AM M26.00004 Effects of bileaflet mechanical heart valve orientation on coronary flow 1. LAURA HAYA, STA/ROS TAVOURLARIS, University of Ottawa — The aortic sinus is approximately tri-radially symmetric, but bileaflet mechanical heart valves (BMHVs), which are commonly used to replace diseased aortic valves, are bilaterally symmetric. This mismatch in symmetry suggests that the orientation in which a BMHV is implanted within the aortic sinus affects the flow characteristics downstream of it. This study examines the effect of BMHV orientation on the flow in the coronary arteries, which originate in the aortic sinus and supply the heart tissue with blood. Planar particle image velocimetry measurements were made past a BMHV mounted at the inlet of an anatomical aorta model under physiological flow conditions. The complex interactions between the valve jets, the sinus vortex and the flow in the right coronary artery were elucidated for three valve orientations. The coronary flow rate was directly affected by the size, orientation, and time evolution of the vortex in the sinus, all of which were sensitive to the valve’s orientation. The total flow through the artery was highest when the valve was oriented with its axis of symmetry intersecting the artery’s opening. The findings of this research may assist surgeons in choosing the best orientation for BMHV implantation.

1The bileaflet valve was donated by St. Jude Medical. Financial support was provided by the Natural Sciences and Engineering Research Council of Canada.

8:52AM M26.00005 Numerical Simulations of the Mechanics of Vitrectomy, ETHAN YOUNG, JEFF ELDREDGE, JEAN-PIERRE HUBSCHMAN, University of California, Los Angeles — Vitreous is the clear, gel-like substance that fills the cavity between the lens and retina in the eye. Treating certain eye diseases requires removing this substance using a minimally-invasive device called a vitreous cutter. Understanding the behavior of this viscoelastic biofluid during surgeries is essential to improving the effectiveness of the procedure. In this study, three-dimensional computational models of vitreous cutters are investigated using an immersed boundary method paired with a viscoelastic constitutive model. The solver uses a fractional-step method to satisfy continuity and traction boundary conditions to simulate the applied suction. The current work extends previous efforts to accurately model the rheological parameters measured by Sharif-Kashani et al. using the Giesekus constitutive equation [Retina, 2013]. The simulations were used to quantify both the average and time-varying flow rate through the device. Values for flow rate are compared with experimental results from Hubschman et al. [Retina, 2009]. Flow features associated with the cutting dynamics are of particular interest, as is the geometry of the cutter itself. These operational and design changes are a target for improving cutter efficacy while minimizing potential tissue damage.

9:05AM M26.00006 Correlation between Hemodynamics and Treatment Outcome of Intracranial Aneurysms after Intervention with Flow Diverters 1, NIKHIL PALIWAL, ROBERT DAMIANO, JASON DAVIES, ADNAN SIDDIOQUI, HUI MENG, University at Buffalo, the State University of New York — Endovascular intervention by Flow Diverter (FD) - a densely woven stent - occludes an aneurysm by inducing thrombosis in the aneurysm sac and reconstructing the vessel. Hemodynamics plays a vital role in the thrombotic occlusion of aneurysms and eventual treatment outcome. CFD analysis of pre- and post-treatment aneurysms not only provides insight of flow modifications by FD, but also allows investigation of interventional strategies and prediction of their outcome. In this study 80 patient-specific aneurysms treated with FDs were retrospectively studied to evaluate the effect of intervention. Out of these cases, 16 required retreatment and thus are considered as having unfavorable outcome. Clinical FD deployment in these cases was simulated using an efficient virtual stenting workflow. CFD analysis was carried out on both pre- and post-treatment cases, and changes in hemodynamic parameters were calculated. Support vector machine algorithm was used to correlate the hemodynamic changes with outcome. Results show that cases having higher flow reduction into the aneurysmal sac have a better likelihood of occlusion. This suggests that changes in hemodynamics can be potentially used to predict the outcome of different clinical intervention strategies in aneurysms.

1This work was supported by the National Institutes of Health (R01 NS091075).
9:18AM M26.00007 Computational modeling of Endovascular Chemofiltration device for removing toxins from blood, VITALIY RAYZ, University of Wisconsin - Milwaukee and Medical College of Wisconsin, BEN TOMPKINS, Penumbra, Inc, ALBERT CHIN, Chemofilter, ANAND PATEL, STEVEN HEITTS, University of California San Francisco, UNIVERSITY OF WISCONSIN MILWAUKEE TEAM, UNIVERSITY OF CALIFORNIA SAN FRANCISCO TEAM, PENUMBRA, INC TEAM — Purpose: Chemotherapy drugs injected intra-arterially in order to destroy tumor cells can cause systemic toxic effects. A catheter-based filtering device temporarily inserted into the veins downstream of the tumor can remove chemotherapy drugs out of the blood stream right after these drugs have had their effect on the tumor. CFD modeling can help optimize hemodynamic performance of the chemofilter membrane, which chemically binds the toxins. Methods: Two alternative designs of the chemofilter were evaluated in order to increase the contact area of the membrane, while minimizing its obstruction to the flow. The Navier-Stokes equations were solved with a finite-volume solver Fluent. Virtual contrast injections were computed by solving the advection-diffusion equation in order to determine the effect of the chemofilter configuration on the flow residence time. Results: The results demonstrated that one of the chemofilter configurations, while having a 10-fold larger contact area, is substantially less obstructive to the flow. Additional considerations, such as feasibility of deployment and re-sheathing of the device, will affect its final design. The optimization of the chemofilter hemodynamic performance will help minimize drug toxicity, thus allowing to use high-dose therapy.

9:31AM M26.00008 Flow diversion and coil embolization may perform best in conjunction for treatment of intracranial aneurysms: a hemodynamic investigation, ROBERT DAMIANO, DING MA, ADNAN SIDDQUI, HUI MENG, University at Buffalo, State University of New York — Coiling and flow diversion is the current standard for treatment of intracranial aneurysms (IAs). Coils deployed into the IA sac trigger its thrombotic occlusion, while flow diverters (FDs) deployed across the IA ostium redirect blood flow and reconstruct the parent vessel. Despite the wide adoption of these interventions, poor treatment outcomes have been reported. Recent clinical reports indicate that IA patients treated with both coils and FDs had better outcomes, compared to individual strategies alone. To better understand the hemodynamic mechanisms underlying coiling and flow diversion, we applied our advanced FEM-based device modeling toolset in conjunction with CFD to investigate 3 clinical strategies: coiling, FD, and FD with adjunctive coiling. Using 3 patient-specific IAs as test beds, we assessed the hemodynamic modifications induced by each strategy. Hemodynamic modifications in inflow rate, velocity, and wall shear stress revealed that coils were most effective at reducing intra-aneurysmal flow, while FD worked best at reducing flow into the IA sac. When coils were combined with FD, these effects appeared to be synergistically enhanced. Our modeling results support clinical observations that flow diversion and coiling may work best in conjunction for treating IAs.

9:44AM M26.00009 Computational fluid dynamics evaluation of incomplete stent apposition in a tapered artery, ERIC POON, VIKAS THONDAPU, ANDREW OOI, University of Melbourne, Australia, UMAIR HAYAT, PETER BARLIS, Northern Health, Australia, Australia, STEPHEN MOORE, IBM Research, Australia — Coronary stents are deployed to prop open blocked arteries and restore normal blood flow, however in-stent restenosis (ISR) and stent thrombosis (ST) remain possible catastrophic complications. Computational fluid dynamics (CFD) analyses can elucidate the pathological impact of alterations in coronary hemodynamics and correlate wall shear stress (WSS) with atherosclerotic processes. The natural tapering of a coronary artery often leads to proximal incomplete stent apposition (ISA) where stent struts are not in contact with the vessel wall. By employing state-of-the-art computer-aided design (CAD) software, generic open-cell and closed-cell coronary stent designs were virtually deployed in an idealised tapered coronary artery. Pulsatile blood flow (80 mL/min at 75 beats/min) was carried out numerically on these CAD models using a finite volume solver. CFD results reveal significant fluctuations in proximal WSS and large recirculation regions in the setting of proximal ISA, resulting in regions of high wall shear stress gradient (WSSG) that have been previously linked to poor endothelial cell coverage and vascular injury. The clinical significance of these proximal high WSSG regions will be correlated with findings from high-resolution in-vivo imaging.

1Supported by the Australian Research Council (LP120100233) and Victorian Life Sciences Computation Initiative (VR0210).

Tuesday, November 24, 2015 8:00AM - 10:10AM
Session M27 Experiments: Analysis, Image Processing and Algorithms

8:00AM M27.00001 On the Singular Value Decomposition of Measured Data, BRENDEEN EPPS, Thayer School of Engineering, Dartmouth College — Singular value decomposition (SVD) is a well-known mathematical tool that can be used to decompose an ensemble of velocity field data into spatiotemporal modes that may reveal coherent flow structures. The proper orthogonal decomposition (POD) is a special case of the SVD, used when the data are uncorrelated in time (as in a turbulent flow). Although the SVD and POD have been widely used in fluid mechanics, Epps and Techet (2010, ExpFluids 48:355-367) were among the first to consider how experimental error affects the results of the SVD. This talk revisits that paper and provides mathematically-rigorous bounds on the errors in the computed singular values and spatio-temporal mode shapes. Given experimental data with unknown error, a procedure is presented to (i) determine the root mean square measurement error and (ii) determine error bars for the singular values and vectors.

8:13AM M27.00002 Iterative Blind Deconvolution Algorithm for Deburring PSP Image of Rotating Surfaces, ANSHUMAN PANDEY, JAMES GREGORY, The Ohio State University — Fast Pressure-Sensitive Paint (PSP) is used in this work to measure unsteady surface pressures on rotating bodies, with iterative image deblurring schemes being developed to correct for image blur at high rotation rates. A significant amount of rotational blur can occur in PSP images acquired in the lifetime mode when the time scale of luminescent decay is long relative to the rotational speed. Image deblurring schemes have been developed to address this problem, but are not currently able to handle strong pressure gradients. Since the local point spread function at each point on the rotor depends on the unknown pressure, restoring such an image is a spatially-varying blind deconvolution problem. An iterative scheme based on the lifetime decay characteristics of PSP has been developed for restoring this image. The scheme estimates the spatially-varying blur kernel without filtering the blurred image and then restores it using classical iterative regularization tools. The resulting scheme is evaluated using computationally-generated pressure fields with strong gradients, as well as experimental data with strong gradients in luminescent lifetime due to a nitrogen jet. Factors such as convergence, image noise, and regularization-iteration count are studied in this work.

1Supported by the U.S. Government under Agreement No. W911W6-11-2-0010 through the Georgia Tech Vertical Lift Research Center of Excellence.
8:26AM M27.00003 Machine learning and synthetic aperture refocusing approach for more accurate masking of fish bodies in 3D PIV data, LOGAN FORD, ABHISHEK BAJPAYEE, ALEXANDRA TECHET, Massachusetts Inst of Tech-MIT — 3D particle image velocimetry (PIV) is becoming a popular technique to study biological flows. PIV images that contain fish or other animals around which flow is being studied, need to be appropriately masked in order to remove the animal body from the 3D reconstructed volumes prior to calculating particle displacement vectors. Presented here is a machine learning and synthetic aperture (SA) refocusing based approach for more accurate masking of fish from reconstructed intensity fields for 3D PIV purposes. Using prior knowledge about the 3D shape and appearance of the fish along with SA refocused images at arbitrarily oriented four different location and orientation and orientation of a fish in a reconstructed volume can be accurately determined. Once the location and orientation of a fish in a volume is determined, it can be masked out.

8:39AM M27.00004 The Anatomy of Fourier-Based Correlation Image Velocimetry and Sources of Decorrelating Errors, MATTHEW GIARRA, Virginia Tech, PAVLOS VLACHOS, Purdue University — Particle image velocimetry (PIV) algorithms have recently been applied to photographs captured using a variety of techniques including schlieren, synchrotron x-ray, and microscope imaging. While the characteristics of these types of images differ greatly from those of particle images, virtually no analysis has been done to determine how these differences affect the performance of Fourier-based cross correlation (CC) algorithms. Here, we analyze schlieren, x-ray, and traditional PIV images to show that the signal-to-noise ratios (SNR) of their CCs vary across spectral wavenumbers, and that the assignment of a single SNR to the CC is an oversimplification that obscures the underlying source of the decorrelating errors. We will show that the failure of traditional algorithms to distinguish correlated from uncorrelated wavenumbers introduces secondary CC peaks that increase measurement uncertainty by decreasing the correlation peak-height ratio, and that cause the measurement to fail by overtaking the true peak. Finally, we introduce a new algorithm that mitigates these issues and increases measurement accuracy by automatically discriminating correlated wavenumbers with no a priori information about the images’ contents.

8:52AM M27.00005 Empirical mode decomposition profilometry: small scale capabilities and comparison to Fourier Transform Profilometry, GUILLAUME LAGUBEAU, Departamento de Física, Universidad de Santiago de Chile, PABLO COBELL1, Departamento de Física, Facultad de Ciencias Exactas y Naturales, Universidad de Buenos Aires and IFIBA, TOMASZ BOBINSKI, PPMH-ESPCI, AGNES MAUREL, Institut Langevin, VINCENT PAGNEUX, Laboratoire d’Acoustique et des Ultrasons, Université du Maine, PHILIPPE PETITJEANS, PPMH-ESPCI — Fringe projection profilometry is an instrument of choice for the instantaneous measurement of the full height map of a free-surface. It is useful to capture interfacial phenomena such as droplet impact and propagation of water waves. We present the Empirical Mode Decomposition Profilometry (EMDP) for the analysis of fringe projection profilometry images. It is based on an iterative filter, using empirical mode decomposition, that is free of spatial filtering and adapted for sources characterized by a broadband spectrum of deformation. Examples of such surfaces can be found in nonlinear wave interaction regimes such as wave turbulence in gravity-capillary water waves. We show both numerically and experimentally that using EMDP improves strongly the profilometry small scale capabilities compared to traditionally used Fourier Transform Profilometry. Moreover, the height reconstruction distortions is much lower: the reconstructed height field is now both spectrally and statistically accurate.

9:05AM M27.00006 New Reconstruction Accuracy Metric for 3D PIV, ABHISHEK BAJPAYEE, ALEXANDRA TECHET, MIT — Reconstruction for 3D PIV typically relies on recombining images captured from different viewpoints via multiple cameras/apertures. Ideally, the quality of reconstruction dictates the accuracy of the derived velocity field. A reconstruction quality parameter Q is commonly used as a measure of the accuracy of reconstruction algorithms. By definition, a high Q value requires intensity peak levels and shapes in the reconstructed and reference volumes to be matched. We show that accurate velocity fields rely only on the peak locations in the volumes and not on intensity peak levels and shapes. In synthetic aperture (SA) PIV reconstructions, the intensity peak shapes and heights vary with the number of cameras and due to spatial/temporal particle intensity variation respectively. This lowers Q but not the accuracy of the derived velocity field. We introduce a new velocity vector correlation factor Qc as a metric to assess the accuracy of 3D PIV techniques, which provides a better indication of algorithm accuracy. For SAPIV, the number of cameras required for a high Qc is lower than that for a high Q. We discuss Qc in the context of 3D PIV and also present a preliminary comparison of the performance of TomoPIV and SAPIV based on Qc.

9:18AM M27.00007 How To Efficiently Sample Data For Computation Of Statistics, BARTON SMITH, Utah State University, DOUGLAS NEAL, LaVision Inc. — The mean of a sample is a random variable with a variance that is the variance of the measured variable divided by the number of samples, assuming the samples are independent. Ensuring independent samples requires that the sampling period is greater than twice the integral time scale T_u. This time scale is the integral over all time of the autocorrelation ρ of the signal. Three signals are analyzed to investigate the convergence of the mean and other statistics. One is the velocity in a turbulent jet measured using a hot-wire. The other two are pressure signals generated by flow through a confined array of cylinders. To determine T_u, 11 sets of 100,000 data points were acquired at high rate. The value of ρ for each record were averaged together. For the chaotic signal from the jet, ρ was generally positive or slightly negative. The pressure signals contain a coherent component that caused the autocorrelation to become a damped oscillation. In this case, the scheme of integrating ρ from 0 to the time where ρ became negative gives poor results. Instead, an exponential was fit to the envelope of ρ and integration was carried out to the point where this exponential function became small. Data were acquired in the jet and cylinder array at rates above and below those consistent with the 2T_u criterion. Even at 8 times the recommended rate the mean converged at the predicted rate [although the error magnitude was larger than the theory] and that at sampling period of 2T_u, the error in the mean was well predicted by theory. Similar results were found for convergence of the variance.

9:31AM M27.00008 Modal Energy Flow Analysis the Highly Modulated Wake of a Wall-mounted Square-based Pyramid, ROBERT MARTINUZZI, ZAHRRA HOSSEINI, Schulich School of Engineering, University of Calgary, BERND NOACK, Institute PPRIME / CNRS, France — We present the first modal energy flow analysis of a time-resolved 3D velocity field from experimental PIV data for a highly modulated wake of a square-based pyramid protruding a boundary layer. The underlying low-order representation is optimized for resolving the base flow variation as well as the first and second harmonics associated with vortex shedding, thus generalizing the triple decomposition of Reynolds & Hussain (1972). The analysis comprises not only a detailed modal balance of turbulent kinetic energy as pioneered by Rempfer & Fasel (1994) for POD models, but also the companion mean-flow energy balance. The experimental results strikingly demonstrate how constitutive elements of mean-field theory (Stuart, 1958) near laminar Hopf bifurcations are still strongly expressed in a turbulent wake characterized by highly modulated, quasi-periodic shedding. The results emphasize, for instance, the stabilizing role of the mean-field manifolds, as explored in the POD model of Aubry et al. (1998). The proposed low-order representation of the flow and modal energy analysis may provide a novel framework for characterizing highly anisotropic wakes and vortex interactions; yielding important insights and reference data for computational turbulence modeling, e.g. URANS.

1The authors wish to recognize the support of the National Science Foundations and Engineering Research Council (Discovery Grant and Scholarship programs) and the ANR Chair of Excellence TUCOROM.
9:44AM M27.00009 Flow classification using machine learning on sparsely sampled experimental flow visualization data. ZHE BAI, STEVEN L. BRUNTON, BINGNI W. BRUNTON, J. NATHAN KUTZ, University of Washington, EURIKA KAISER, ANDREAS SPOHN, BERNDO R. NOACK, Institute PPRIME, France — In this work, we consider a data-driven approach for characterizing the transitional separation bubble using video images and dimensionality reduction with supervised classification techniques to discriminate between an actuated and an unactuated flow. Flow visualizations are captured using the hydrogen bubble technique along a smooth ramp in a low-speed water tunnel, and instabilities are excited in the actuated case by oscillating a thin horizontal wire inside the boundary layer upstream of the separation. We apply clustering techniques, including the linear discriminant analysis (LDA) in a POD/PCA reduced subspace, to classify the baseline and controlled cases of the flow field from image data. With sparse subsampled pixel measurements, similar classification performance is obtained compared to that of the full-resolution images. Next, we demonstrate a sparse sensor optimization algorithm to locate a small set of pixels that optimally inform the classification task. With 5-10 specially selected sensors, the median cross-validated classification accuracy is ≥ 97%, as opposed to a random set of 5-10 pixels, which result in classification accuracy of 70-80%. The methods developed here apply broadly to high-dimensional data from fluid dynamics experiments.

9:57AM M27.00010 Image Processing Method of the Motion-Capturing PSP/TSP for the Measurement of a Free-Flight Object, MASATO ISHII, National Research Institute of Police Science, Japan, HIDEKI GOYA, TAKESHI MIYAZAKI, University of Electro-Communications, HIROTAKA SAKAUE, University of Notre Dame — The motion-capturing PSP/TSP system consists of a two-color PSP/TSP and a high-speed color camera. Red and green luminescent images are acquired simultaneously as signal and reference outputs by this system. Simply by rationing the red and the green images, we can obtain a pressure/temperature distribution on the surface of a target object. This system is applied to measure the surface pressure/temperature of a free-flight object. However, an acquired image includes motion blur, focus blur and random noise around the object. We discuss image processing methods and evaluations to optimize those uncertainties. Three types of the edge detect methods are used, which are the sobel, the laplassian and the canny. We will also show the evaluation results to discuss an optimized image processing for the motion-capturing PSP/TSP system.

Tuesday, November 24, 2015 8:00AM - 10:10AM – Session M28 Experiments: Visualization, Tagging and Tracking

8:00AM M28.00001 Time-Resolved Visualization of Görtler Vortices in a Pulsed Convex Wall Jet using Fast Pressure-Sensitive Paint, JAMES GREGORY, The Ohio State University, RON DANON, DAVID GREENBLATT, Technion - Israel Institute of Technology — The time-resolved formation and structure of Görtler vortices in a pulsed convex wall jet are studied in this work. While the presence of Görtler vortices in laminar boundary layers on concave surfaces can be clearly observed, their presence in wall jets flowing over convex surfaces is difficult to discern due to transition to turbulence in the outer part of the jet. This work employed fast-response pressure-sensitive paint (PSP), which has a documented flat frequency response greater than 5 kHz, to visualize the time-resolved formation of the wall jet and the details of the Görtler vortices. The radius of curvature of the wall jet was 8 cm, and the Reynolds number (based on slot height and jet exit velocity) was varied between 5×10^2 and 4×10^4. The characteristic spanwise wavelength of the vortices was studied as a function of jet Reynolds number. Furthermore, as the Reynolds number was increased, various secondary instabilities were observed that led to laminar-turbulent transition.

8:13AM M28.00002 Spectral Measurements from the Optical Emission of the A.C. Plasma Anemometer, ERIC MATLIS, CURTIS MARSHALL, THOMAS CORKE, University of Notre Dame, SIVARAM GOGINENI, Spectral Energies, LLC — The optical emission properties of a new class of AC-driven flow sensors based on a glow discharge (plasma) is presented. These results extend the utility of the plasma sensor that has recently been developed for measurements in high-enthalpy flows. The plasma sensor utilizes a high frequency (1MHz) AC discharge between two electrodes as the main sensing element. The voltage drop across the discharge correlates to changes in the external flow which can be calibrated for mass-flux (p\(\rho U\)) or pressure depending on the design of the electrodes and orientation relative to the free-stream flow direction. Recent experiments examine the potential for spectral analysis of the optical emission of the discharge to provide additional insight to the flow field. These experiments compare the optical emission of the plasma to emission from breakdown due to an ND:YAG laser. The oxygen 777.3 nm band in particular is a focus of interest as a marker for the determination of gas density.

8:26AM M28.00003 High-Speed OH* Chemiluminescence Imaging of Shock Tube End-Wall, V.A. TROUTMAN, V.A. MILLER, C.S. STRAND, A.M. TULGESTRE, M.F. CAMPBELL, D.F. DAVIDSON, R.K. HANSON, Stanford University — We have developed a high-speed OH* chemiluminescence imaging diagnostic and a transparent end-wall for the Stanford Aerosol Shock Tube to better understand the structure and homogeneity of the combustion event behind a reflected shock wave. We use an intensified high repetition rate imaging system to acquire images of OH* chemiluminescence (near 308 nm) at 10-33 kHz from n-heptane combustion. Case studies are presented to illustrate the power of this novel imaging diagnostic: first, we infer the temperature homogeneity of the ignition event; then we image the effect of surface imperfections in the wall of the shock tube; lastly, we visualize the effect of particulates in the shock tube and verify the importance of shock tube cleaning routines.
8:39AM M28.00004 The 3D flow structures generated by a pair of cubic roughness elements in a turbulent channel flow resolved using holographic microscopy\(^1\), JIAN GAO, JOSEPH KATZ, Johns Hopkins University — In studies of turbulent flows over rough walls, considerable efforts have been put on the overall effects of roughness parameters such as roughness height and spanwise arrangement on the mean profiles and turbulence statistics. However, there is very little experimental data on the generation, evolution, and interaction among roughness-initiated turbulent structures, which are essential for elucidating the near-wall turbulence production. As a first step, we approach this problem experimentally by applying digital holographic microscopy (DHM) to measure the flow and turbulence around a pair of cubic roughness elements embedded in the inner part of a high Reynolds number turbulent channel flow (Re, \(\approx 2000 – 5000\)). The ratio of half-channel height (\(h\)) to cube height (\(a\)) is 25, and the cubes are aligned in the spanwise direction, and separated by 1.5\(a\). DHM provides high-resolution three-dimensional (3D) three-component (3C) velocity distributions. The presentation discusses methods to improve the data accuracy, both during the hologram acquisition and particle tracking phases. First, we compare and validate velocity fields obtained from a two-view DHM system. Subsequently, during data processing, the seven criteria used for particle tracking is validated and augmented by planar tracking of particle image projections. Sample results reveal instantaneous 3D velocity fields and vortical structures resolved in fine details of several wall units.

\(^1\)Funded by NSF and ONR.

8:52AM M28.00005 Color gradient background oriented schlieren imaging, FRANK AUSTIN MIER, MICHAEL HARGATHER, New Mexico Tech — Background oriented schlieren (BOS) imaging is a method of visualizing refractive disturbances through the comparison of digital images. By comparing images with and without a refractive disturbance visualizations can be achieved via a range of image processing methods. Traditionally, backgrounds consist of random distributions of high contrast speckle patterns. To image a refractive disturbance, a digital image correlation algorithm is used to identify the location and magnitude of apparent pixel shifts in the background pattern. Here a novel method of using color gradient backgrounds is explored as an alternative. The gradient background eliminates the need to perform an image correlation between the two digital images, as simple image subtraction can be used to identify the location, magnitude, and direction of the image distortions. This allows for quicker processing. Two-dimensional gradient backgrounds using multiple colors are shown. The gradient backgrounds are demonstrated to provide quantitative data limited only by the camera’s pixel resolution, whereas speckle backgrounds limit resolution to the size of the random pattern features and image correlation window size. Additional results include the use of a computer screen as a background.

9:05AM M28.00006 Femtosecond laser flow tagging in non-air flows, YIBIN ZHANG, NATHAN CALVERT, Princeton Univ — The Femtosecond Laser Electronic Excitation Tagging (FLEET) [Michael, J. B. et. al. Applied optics, 50(26), 2011] method is studied in nitrogen-containing gaseous flows. The underlying mechanism behind the FLEET process is the dissociation of molecular nitrogen into atomic nitrogen, which produces long-lived florescence as the nitrogen atoms recombine. Spectra and images of the resulting tagged line provide insight into the effects of different atmospheric gases on the FLEET process. The ionization cross-section, conductivity and energy states of the gaseous particles are each brought into consideration. These experiments demonstrate the feasibility for long-lived flow tagging on the order of hundreds of microseconds in non-air environments. Of particular interest is the enhancement of the FLEET signal with the addition of argon gas, and the non-monotonic quenching effect of oxygen on the length, duration and intensity of the resulting signal and spectra. FLEET is characterized in number of different atmospheric gases, including that simulating Mar’s atmospheric composition.

9:18AM M28.00007 A method for extracting the turbulence intensity and integral length scale form single-component molecular tagging velocimetry\(^1\), DAVID OLSON, AHMED NAGUIB, MANOOCHREH KOOCHESFAHANI, Michigan State University — This study demonstrates a method to extract the turbulence intensity and integral length scale from single-component molecular tagging velocimetry (1c-MTV) measurements of freestream turbulence. These measurements are challenging because of the very small magnitude of the fluctuating velocities compared to the freestream velocity, and the presence of low-frequency facility unsteadiness as well as measurement white noise. The approach takes advantage of the inherent capabilities of 1c-MTV to measure the streamwise velocity at a very high spatial resolution of 52 \(\mu\)m over a line that extends 5.3 cm in the cross-stream direction. The resulting data set is equivalent to that which would result from a linear sensor array of 1024 tightly-spaced hot wires, enabling computation of the cross-stream autocorrelation function efficiently. The high-spatial resolution of the measurements allows removal of the white noise contribution to the autocorrelation function, whereas the extended domain of the measurements facilitates rejection of the influence of the low-frequency facility unsteadiness. The “noise-removed” autocorrelation function is used to compute the intensity and integral length scale of turbulence. The procedure is applied to grid-generated freestream turbulence.

\(^1\)This work was supported by AFOSR grant number FA9550-10-1-0342.

9:31AM M28.00008 Study of gas-liquid flow in model porous media for heterogeneous catalysis\(^1\), MARIE FRANCOIS, HUGUES BODIGUEL, PIERRE GUILLOT, Laboratory of the Future, LABORATORY OF THE FUTURE TEAM — Heterogeneous catalysis of chemical reactions involving a gas and a liquid phase is usually achieved in fixed bed reactors. Four hydrodynamic regimes have been observed. They depend on the total flow rate and the ratio between liquid and gas flow rate. Flow properties in these regimes influence transfer rates. Rather few attempts to access local characterization have been proposed yet, though these seem to be necessary to better describe the physical mechanisms involved. In this work, we propose to mimic slices of reactor by using two-dimensional porous media. We have developed a two-dimensional system that is transparent to allow the direct observation of the flow and the phase distribution. While varying the total flow rate and the gas/liquid flow rate ratio, we observe two hydrodynamic regimes: at low flow rate, the gaseous phase is continuous (trickle flow), while it is discontinuous at higher flow rate (pulsed flow). Thanks to some image analysis techniques, we are able to quantify the local apparent liquid saturation in the system. Its fluctuations in time are characteristic of the transition between the two regimes: at low liquid flow rates, they are negligible since the liquid/gas interface is fixed, whereas at higher flow rates we observe an alternation between liquid and gas. This transition between trickle to pulsed flow is in relative good agreement with the existing state of art. However, we report in the pulsed regime important flow heterogeneities at the scale of a few pores. These heterogeneities are likely to have a strong influence on mass transfers.

\(^1\)We acknowledge the support of Solvay.
9:44AM M28.00009 Measuring turbulent fluid dispersion using laser induced phosphorescence\(^1\). DENNIS VAN DER VOORT, NICO DAM, WILLEM VAN DE WATER, RUDIE KUNNEN, HERMAN CLERCX, GERT JAN VAN HELJST, Eindhoven Univ of Tech — Fluid dispersion due to turbulence is an important subject in both natural and engineering processes, from cloud formation to turbulent mixing and liquid spray combustion. The combination of small scales and often high velocities results in few experimental techniques that can follow the course of events. We introduce a novel technique, which measures the dispersion of “tagged” fluid particles by means of laser-induced phosphorescence, using a solution containing a europium-based molecular complex with a relatively long phosphorescence half-life. This technique is used to measure transport processes in both the dispersion of droplets in homogeneous isotropic turbulence and the dispersion of fluid of near-nozzle spray breakup processes. By tagging a small amount of droplets/fluid via laser excitation, the tagged droplets can be tracked in a Lagrangian way. The absolute dispersion of the droplets can be measured in a variety of turbulent flows. Using this technique it is shows that droplets around $St = \frac{\tau_p}{\tau_\eta} \approx 1$ (Stokes number) disperse faster than true fluid tracers in homogeneous isotropic turbulence, as well as differences between longitudinal and radial dispersion in turbulent sprays.

\(^1\)This work is part of the research programme of the Foundation for Fundamental Research on Matter (FOM), which is part of the Dutch Organisation for Scientific Research (NWO)

9:57AM M28.00010 Improvements of a nano-scale crossed hot-wire for high Reynolds number measurements\(^1\). YUYANG FAN, MARCUS HULTMARK, Princeton University — Hot-wire anemometry, despite its limited spatial and temporal resolution, is still the preferred tool for high Reynolds number flow measurements, mainly due to the continuous signal. To address the resolution issues, the Nano-Scale Thermal Anemometry Probe (NSTAP) was developed at Princeton University. The NSTAP has a sensing volume more than one order of magnitude smaller than conventional hot-wires, and it has displayed superior performance. However, the NSTAP can only measure a single component of the velocity. Using a novel combining method, a probe that enables two-component velocity measurements has been created (the $x$-NSTAP). The measurement volume is approximately $50 \times 50 \times 50\mu m$, more than one order of magnitude smaller in all directions compared to conventional crossed hot-wires. The $x$-NSTAP has been further improved to allow more accurate measurements with the help of flow visualization using a scaled model but matching Reynolds number. Results from turbulent flow measurements with the new $x$-NSTAP are also presented.

\(^1\)Supported under NSF grant CBET-1510100 (program manager Dimitrios Papavassiliou)

Tuesday, November 24, 2015 8:00AM - 10:10AM —
Session M29 DFD GPC: Geophysical Fluid Dynamics: Cryosphere and Ice-Ocean Interactions 310 - Colin Meyer, Harvard University

8:00AM M29.00001 Moffatt eddies at the base of ice sheets\(^1\). COLIN R. MEYER, Harvard Univ, TIMOTHY T. CREYTS, LDEO-Columbia Univ, JAMES R. RICE, Harvard Univ — Despite extensive radar surveys of the deep ice, the conditions at the base of ice sheets remain uncertain. Complex structures that include large stratigraphic folds and basal freeze-on ice appear to be related to topography. In many locations beneath both Greenland and Antarctica, ice flows across deep valleys, potentially forming viscous eddies that can confound the interpretation of ice-bed processes. To understand the formation of these eddies, we use a set up analogous to that of Moffatt (1964), where our domain is a subglacial valley. We numerically solve the non-Newtonian Stokes equations with a shear-thinning power-law rheology to determine the critical valley angle for the eddies to form. The shear-thinning nature of ice allows for greater shear localization and, therefore, ice requires smaller valley angles (steeper slopes) to form eddies than a Newtonian fluid. Due to the significant variation of temperature from the warm base to the cold surface of the ice sheet, we analyze eddy formation when the rheology is temperature dependent. The warmer basal ice is less viscous and eddies form in larger valley angles (shallower slopes). Finally, we solve for the ice flow over topography from the Gamburtsev subglacial mountains and show Moffatt eddies in the subglacial valleys.

\(^1\)NSF GRF No. DGE1144152

8:13AM M29.00002 Forced convective melting at an evolving ice-water interface\(^1\). ESHWAN RAMUDU, BENJAMIN HIRSH, PETER OLSON, ANAND GNANADESIKAN, Department of Earth and Planetary Sciences, Johns Hopkins University — The intrusion of warm Circumpolar Deep Water into the ocean cavity between the base of ice shelves and the sea bed in Antarctica causes melting at the ice shelves’ basal surface, producing a turbulent melt plume. We conduct a series of laboratory experiments to investigate how the presence of forced convection (turbulent mixing) changes the delivery of heat to the ice-water interface. We also develop a theoretical model for the heat balance of the system that can be used to predict the change in ice thickness with time. In cases of turbulent mixing, the heat balance includes a term for turbulent heat transfer that depends on the friction velocity and an empirical coefficient. We obtain a new value for this coefficient by comparing the modeled ice thickness against measurements from a set of nine experiments covering one order of magnitude of Reynolds numbers. Our results are consistent with the altimetry-inferred melting rate under Antarctic ice shelves and can be used in climate models to predict their disintegration.

\(^1\)This work was supported by NSF grant EAR-110371

8:26AM M29.00003 Direct numerical simulation of convection and dissolution at a vertical ice-seawater interface. BISHAKHDATTA GAYEN, ROSS W. GRIFFITHS, ROSS C. KERR, The Australian National University — Direct numerical simulations are performed to investigate the convection generated when a wall of ice dissolves into seawater under Antarctic ocean conditions. The ambient water temperatures are kept between $-1^\circ C$ and $6^\circ C$ and salinities around 35 ppm, where diffusion of salt to the ice-water interface depresses the freezing point and further enhances heat diffusion to the ice. We use three coupled interface equations, along with the Boussinesq approximation and the equation of state for seawater, to solve for interface temperature, salinity and melt rate. Fluxes of both heat and salt to the interface play a significant role in governing the rate of dissolution of ice. At the presently achievable Grashof numbers turbulence is equally produced from both buoyancy and velocity shear, which indicates the importance of shear production at geophysical scales.
8:39AM M29.00004 Subglacial hydrology as a control on ice stream shear margin locations. THIABAUT PEROL, JAMES R. RICE, Harvard University, JOHN D. PLATT, Carnegie Institution for Science, JENNY SUCKALE, Stanford University — Ice streams are fast-moving bands of ice separated from the nearly stagnant ice in the adjacent ridge by zones of highly localized deformation known as shear margins. However, it is presently unclear what mechanisms can control the location of shear margins. Within the shear margin, the transition from a slipping bed beneath the ice stream to a locked bed beneath the ridge concentrates stresses. We show that subglacial hydrology can select the shear margin location by strengthening the till within the margin. Our study uses a two-dimensional thermo-mechanical model in a cross-section perpendicular to the directional flow. We show that the intense straining at the shear margins can generate large temperate regions within the deforming ice. Assuming that the melt generated in the temperate ice collects in a drainage channel at the base, we show that the channel locally decreases the pore pressure in the till. For a Coulomb-plastic rheology, this depressed pore pressure leads to a basal strength substantially higher than that inferred under the majority of the shear margin. Our results show that the additional basal resistance produced by the channel can reduce the stresses concentrated on the locked bed. Matching the model to surface velocity data at Whillans ice stream margin, we show that a stable shear margin occurs when the slipping-to-locked bed transition is less than 500 m away from a channel operating at an effective pressure of 200 kPa if the basal hydraulic transmissivity is equivalent to that of a water-film 0.2 mm thick.

8:52AM M29.00005 A Theoretical and Experimental Investigation of Ice-Shelf Flow Dynamics. MARTIN WEARING, GRAE WORSTER, Univ of Cambridge, RICHARD HINDMARSH, British Antarctic Survey — Ice-shelf buttressing is a major control on the rate of ice discharged from fast-moving ice streams that drain the Antarctic Ice Sheet. The magnitude of the force that an ice shelf exerts on the ice stream depends on its shape and position. This geometry is determined by the ice shelf environment, resulting from retreat due to iceberg calving and shelf advance due to flow. In contrast to large-scale ice-sheet models, which require high resolution datasets, we aim to gain insight using simple idealized models, focusing on the transition from lateral confinement to non-confinement. By considering a confined shelf with lateral shear stresses controlling the flow, steady-state analytical solutions can be calculated. These solutions are then compared to a numerical model for a confined flow, which incorporates both shear and extensional stresses. A boundary layer close to the calving front is identified as having dominant extensional and shear stresses. We test these idealized models against fluid-mechanical laboratory experiments, designed to simulate the flow of an ice shelf in a narrow channel. From these experiments velocity fields and altimetry for the ice-sheet are collected, allowing for comparison with the theoretical models and geophysical data.

9:05AM M29.00006 Stability of lubricated ice sheets. KATARZYNA N. KOWAL, M. GRAE WORSTER, Institute of Theoretical Geophysics, DAMTP, University of Cambridge — A significant amount of the Antarctic ice sheet drains towards the ocean through a network of ice streams, fast-moving regions of ice that are generally well lubricated at their base by a layer of water-saturated, sub-glacial sediment known as till. Although till has a complex, nonlinear rheology, it deforms viscously over large spatial scales with an effective viscosity much lower than that of ice. Its dynamical interaction with the overlying ice can initiate a spontaneous instability of ice flow resulting in the formation of ice streams. We examine this interaction both mathematically and experimentally by considering the viscous coupling between two layers of fluid spreading under gravity. A series of our recent fluid-mechanical experiments reveal a novel cross-flow fingering instability if the lower layer is less viscous. We perform a linear stability analysis and explain the instability mechanism in terms of a jump in hydrostatic pressure gradient, stabilised by horizontal shear at large wave numbers, and assess the possibility of this mechanism leading to ice-stream formation.

9:18AM M29.00007 Free fingering at the contact between spreading viscous fluids. JEROME NEUFELD, BP Institute, Department of Earth Sciences, Department of Applied Mathematics and Theoretical Physics, University of Cambridge, LAURA GELL, Department of Physics, University of Cambridge, FINN BOX, BP Institute, University of Cambridge — The spreading of viscous fluids is an everyday phenomena with large-scale applications to the flow of glaciers and the dynamics of mountain formation in continental collisions. When viscous fluids spread on an undeformable base the contact line is stable to perturbations. In contrast, when less viscous fluids displace more viscous fluids in a Hele-Shaw flow or convection, the contact line is unstable to fingering instabilities. Here, we show, experimentally and theoretically, that when a viscous fluid spreads on a pre-existing layer of fixed depth and differing viscosity the geometry of the contact line depends sensitively on the ratio of fluid viscosities, the input flux and the initial layer depth. When the injected fluid is less viscous the contact line may become unstable to an alternating finger mechanism reminiscent of Saffman-Taylor fingering. We explore the parameter space of this new instability, and highlight its applicability to understanding mountain formation and glacial ice streams.

9:31AM M29.00008 Formation of snow penitentes by radiative instabilities. WILKO ROHLS, RWTH - Aachen — Penitents are large scale snow or ice structures covering snow and glacier fields in the tropics or subtropics at centimeters up to 30m, with their walls preferentially orientated from east to west. Although the distribution of penitents on a snow or ice field appears chaotic, they exhibit a characteristic pattern with a distinct separation distance, for which their formation can be associated to some kind of instability. The instability results from the process of radiative trapping, which is caused by higher absorbed solar radiation in the field. We present results for idealized test cases motivated by our scaling analysis.

9:44AM M29.00009 The dynamics of a suspension of solidifying, buoyant ice crystals. DAVID REES JONES, ANDREW WELLS, University of Oxford — In a wide range of geophysical and industrial situations, the solidification of a liquid melt occurs through the growth of solid crystals suspended in the melt. For example, so-called frazil ice crystals form by freezing of the sub-glacial ocean. Ice crystals also form in the interior of solidifying magma chambers. The growth of these crystals is dynamically coupled to the flow. Advection enhances the transport and removal of latent heat that controls crystal growth, whilst the particles provide hydrodynamic feedbacks on the flow. The crystal density is typically different to the liquid density, which induces relative motion, and crystals may also induce density gradients within the fluid itself through the temperature field. We develop scaling arguments for the relative importance of crystal growth, accretion, nucleation and transport as a function of particle size and properties of the fluid flow. We introduce a new framework for the direct numerical simulation of the coupling of solidifying, buoyant particles to the fluid flow using a Lattice Boltzmann Method and present results for idealized test cases motivated by our scaling analysis.

9:57AM M29.00010 Turbulent plumes from ice melting into a linearly stratified ocean. ANDREW WELLS, University of Oxford, SAMUEL MAGORRIAN, University of Manchester — The melting of submerged marine glacier termini and ice shelves floating atop the ocean has important implications for ice sheet dynamics and sea level rise. When vertical or inclined ice faces melt into a warm salty ocean, the fresh meltwater rises in a buoyant plume along the ice-ocean interface and the resulting turbulent heat flux transfer provides a feedback to the climate system. By matching the dynamics of well-mixed meltwater plumes rising along planar ice faces through a linearly stratified ocean, with vertical gradients of background ocean temperature and salinity. When the driving buoyancy force is dominated by salinity differences, the flow develops in a repeating series of layers, with the meltwater plume accelerating along the slope, rising past its neutral density level, and then separating from the ice face and intruding into the background ocean. We determine approximate scaling laws for the layer heights, melting rates and flow properties as a function of the background ocean temperature and salinity. These scaling laws provide a good collapse across a range of numerical solutions of the plume model, and may prove useful as a simple parameterisation of glacial melting in stratified Greenland fjords.
8:00AM M30.00001 Lock-release gravity currents over a sparse and dense rough bottom

Claudia Cenedese, Woods Hole Oceanographic Institution, Roger Nokes, University of Canterbury, Jason Hyatt, Massachusetts Maritima Academy — Dense oceanic overflows mix with surrounding waters along the descent down the continental slope. The amount of entrainment and dilution dictates the final properties of these overflows, and thus is of fundamental importance to the understanding of the formation of deep water masses. We will discuss laboratory experiments investigating the mechanisms by which bottom roughness enhances or inhibits entrainment and dilution in a lock-release dense gravity current. The bottom roughness has been idealized by an array of cylinders. Both spacing (sparse vs. dense configuration) and height of the roughness elements compared with the height of the current have been varied. Both density and velocity fields have been obtained. Experimental results suggest that enhancement of the entrainment/dilution of the current can occur due to two different mechanisms. For a sparse configuration the dense current propagates between the cylinders and the entrainment is enhanced by the vortices generated in the wake of the cylindrical obstacles. For a dense configuration the dense current rides on top of the cylinders and the dilution is enhanced by the onset of convective instability between the dense current above the cylinders and the ambient lighter water between the cylinders. For large values of the ratio of the lock height to the cylinder height, H/hc, the dense current behavior approaches that of a current over a smooth bottom, while the largest deviations from the smooth bottom case are observed for small values of H/hc.

1Support was given by NSF Project OCE-1333174.

8:13AM M30.00002 Dynamics of double-diffusive lock-exchange gravity currents

Nathan Konopliv, Eckart Meiburg, Univ of California - Santa Barbara — The dynamics of double-diffusive gravity currents exhibiting the fingering instability were examined using 2D simulations of a lock exchange initial configuration. Both the initial stability ratio and the diffusivity ratio were varied. It was found that although the spreading of the currents was governed by a balance of buoyancy and turbulent drag forces, currents with more intense fingering spread faster than those with less intense or no fingering. This was due to an increase in the buoyancy of the current with stronger fingering, which had a stronger effect than the increased drag. The fingering also affected the thickness of the currents, with more fingering corresponding to thinner currents. The mechanism that caused the thinner currents was also responsible for the creation of secondary and tertiary currents after a long time in a simulation that had intense fingering. If no secondary or tertiary currents formed, the density of the current was governed by a balance of double-diffusive and diffusive fluxes. An energy budget analysis revealed that double-diffusive currents released more potential energy, had more dissipation and converted a significant amount of internal energy into potential energy via the diffusion of heat and salinity.

8:26AM M30.00003 Mixing Induced by Colliding Gravity Currents

Christopher Hocut, University of Notre Dame, Faizle Hussain, Texas Tech University, Harindra Fernando, University of Notre Dame — Colliding gravity current is a widespread phenomenon in complex-terrain meteorology. Nevertheless, only a few detailed studies have been conducted on the mixing and turbulent transport processes during collision, and no parameterization exists to incorporate the mixing effects of collision in mesoscale models. To this end, controlled laboratory experiments were conducted in a double lock-exchange configuration. Velocity and density measurements were made simultaneously using a PIV/PLIF system. Phase aligned ensemble-averaging was employed to elicit mean and turbulent quantities. Collisions cause localized instabilities both along the density interface and in the interior of gravity currents. The turbulence near the density interface induces strong mixing and, along with ambient fluid entrainment, produces strong fluctuations of buoyancy flux. A time scale for the evolution of ensuing turbulence as well as scaling for the entrainment velocity was delineated for the high Reynolds number cases. The flow was replete with turbulent vortices generated by the collisions, and they decayed exponentially with time.

1Funded by Office of Naval Research Award # N00014-11-1-0709, Mountain Terrain Atmospheric Modeling and Observations (MATER-HORN) Program.

8:39AM M30.00004 Circulation-based modeling of gravity currents propagating into ambients with arbitrary shear and density stratification

Mohammad Nasr-Azadani, Eckart Meiburg, University of California, Santa Barbara — We develop a vorticity-based approach for modeling quasisteady gravity currents propagating into arbitrary density and velocity stratification. The model enforces the conservation of mass, horizontal and vertical momentum, and in contrast to previous approaches it does not rely on empirical, energy-based closure assumptions. Instead, the effective energy loss of the flow can be calculated a posteriori. The present model results in the formulation of a second order, nonlinear ODE that can be solved in a straightforward fashion to determine the gravity current velocity, along with the downstream ambient velocity and density profiles. Comparisons between model predictions and DNS simulations show excellent agreement. They furthermore indicate that for high Reynolds numbers the gravity current height adjusts itself so as to maximize the loss of energy.

8:52AM M30.00005 Entrainment dynamics in self-adjusting gravity currents using simultaneous velocity-density measurements

Sridhar Balasubramanian, Department of Mechanical Engineering, Indian Institute of Technology Bombay, Qiang Zhong, Harindra Fernando, Department of Civil and Environmental Engineering and Earth Sciences, University of Notre Dame, USA — Gravity currents can modify their flow characteristic by entraining and mixing with the ambient fluid. The entrainment in such systems may depend on a variety of intrinsic parameters such as, initial density difference, ∆ρ, total height of the fluid, H, and slope of the terrain, α. Thus, it is imperative to study the entrainment dynamics of a gravity current in order to have a clear understanding of the mixing transitions that govern the flow physics such as the shear layer thickness, ∆u, and the mixing layer thickness, δm. Experiments were conducted in a lock-exchange type facility, where a self-adjusting gravity current is formed, for which the only governing parameter is the Reynolds number, Re = u_f H / ν, where u_f = 0.4 √ g H is the frontal velocity. Simultaneous PIV-PLIF technique is employed to get the velocity and density statistics. A control volume based flux method is used to calculate the flux entrainment coefficient, E_J, for a Reynolds number of Re = 400–12000 used in our experiments. The results show transition at Re ≈ 4x10^3, where the mixing occurs due to Kelvin-Helmholtz billows that promote small scale local mixing, and cause a spike in the flux entrainment velocity.

1SB acknowledges funds from University of Notre Dame, IIT Bombay
2Aerospace and Mechanical Engineering, University of Notre Dame
9:05AM M30.00006 Gravity currents down a slope in the acceleration phase\textsuperscript{1}, YU-LIN HUANG, ALBERT DAI, National Taiwan University — Gravity currents generated from an instantaneous buoyancy source propagating down a slope in the range of $0^\circ \leq \theta < 90^\circ$ have been investigated. Front velocity history shows that, after the heavy fluid is released from rest, the flow goes through the acceleration phase, reaching a maximum front velocity $U_{f,\text{max}}$, and followed by the deceleration phase. The existence of a maximum of $U_{f,\text{max}}$ is found near $\theta = 40^\circ$, which is supported by the theory. It is identified that the time of acceleration decreases as the slope angle increases, when the slope angle is approximately greater than $10^\circ$, and the time of acceleration increases as the slope angle increases for gravity currents on lower slope angles. A fundamental difference in flow patterns, which helps explain the distinct characteristics of gravity currents on high and low slope angles using scaling arguments, is revealed. Energy budgets further show that, as the slope angle increases, the ambient fluid is more easily engaged in the gravitational convection and the potential energy loss is more efficiently converted into the kinetic energy associated with ambient fluid.

\textsuperscript{1}Supported by Taiwan Ministry of Science and Technology

9:18AM M30.00007 Vorticity models for gravity currents propagating into two-layer stratified ambients, MOHAMMAD AMIN KHODKAR, PhD student, MOHAMAD NASR-AZADANI, Postdoctoral student, ECKART MEIBURG, Professor — We investigate the propagation of Boussinesq gravity currents into two-layer stratified ambients by means of vorticity models and two-dimensional Navier-Stokes simulations. The control volume-based vorticity model enforces the conservation of vertical momentum by balancing the in- and outflow of vorticity with the baroclinic vorticity generation inside the control volume. In this way, it avoids the need for energy-based closure assumptions, such as those invoked in earlier modeling efforts. We find that for flow fields both with and without upstream propagating bores, the model predictions regarding the gravity current and bore velocities are in good agreement with the simulation results.

9:31AM M30.00008 On the propagation of a gravity current into a fluid with horizontal and vertical density gradient, HIEU PHAM, SUTANU SARKAR, University of California, San Diego — Large-eddy simulations are used to investigate the dynamics of a rotating gravity current propagating in the ocean surface mixed layer on top of a pycnocline. Two simulations with different conditions in the surface mixed layer are performed: one with a homogeneous mixed layer and one with a horizontal density gradient. In the latter case, the density in the mixed layer decreases with propagating distance. In both cases, a nonlinear bore forms at the front of the gravity current with Kelvin-Helmholtz billows that develop below and in the region behind the bore. In the case with a homogeneous mixed layer, the bore propagates at a constant speed which is proportional to $\sqrt{g' H}$ where $g'$ is the reduced gravity and $H$ is the mixed layer depth. In the case with the horizontal gradient, the speed decreases in time. It is found that the horizontal density gradient influences the propagation of the bore in the following ways: (1) It reduces the buoyancy difference which drives the bore; (2) It generates a horizontal pressure gradient which drives a counter gravity current opposing the bore. The counter current creates a flow-converging zone ahead of the bore. The speed of the bore is found to be dependent of the horizontal density gradient and the traveling distance of the bore.

9:44AM M30.00009 Intrusive gravity currents interacting with obstacles in a continuously stratified environment\textsuperscript{1}, JIAN ZHOU, SUBHAS VENAYAGAMOORTHY, Colorado State Univ — The flow dynamics of intrusive gravity currents past a surface-mounted obstacle was investigated using large eddy simulations. The propagation dynamics of a classical intrusive gravity current in the absence of an obstacle was first simulated to validate the numerical simulations. The numerical results showed good agreement with experimental measurements. An obstacle with a dimensionless height of $\tilde{D} = D/H$ ($H$ the total fluid depth) was then introduced and acted as a controlling factor of the downstream flow pattern. It is found that for short obstacles, the intrusion re-established itself downstream in a form similar to the classical intrusion (in the absence of an obstacle). However, for tall obstacles, the downstream flow was found to be a joint effect of horizontal advection, overshoot-springback phenomenon, and the Kelvin-Helmholtz instability. Three regimes of downstream obstacle-affected propagation speed were identified depending on values of $\tilde{D}$, i.e., a retarding regime ($\tilde{D} \approx 0 \sim 0.3$), an impounding regime ($\tilde{D} \approx 0.3 \sim 0.6$), and a choking regime ($\tilde{D} \approx 0.6 \sim 1.0$).

\textsuperscript{1}Funded by the Office of Naval Research and the National Science Foundation

9:57AM M30.00010 Front conditions for gravity currents in channels of general cross-section: some general conclusions, MARIUS UNGARISH, Technion, Israel Institute of Technology — We consider the propagation of a high-Reynolds-number gravity current in a horizontal channel with general cross-section of width $f(z), 0 \leq z \leq H$; the gravity acceleration $g$ acts in $\pm z$ direction. (The rectangular case is $f(z) = \text{const}$.) We assume a two-layer system of fluids of densities $\rho_i$ (current, of height $h$) and $\rho_o$ (ambient, filling the remaining part of the channel). We revisit the derivation of the nose Froude-number condition $Fr = U_f/(g' h)^{1/2}$; $U_f$ is the speed of propagation of the current and $g' = (\rho_i/\rho_o - 1)g$. We present compact insightful expressions of $Fr$ and energy dissipation as a functions of $\varphi$ (= area fraction occupied by the current in the cross-section), and show that a degree of freedom is present. We demonstrate that the extension of the closure suggested by Benjamin for the rectangular cross-section, namely that the bottom is a perfect stagnation line, produces $Fr$ solutions which are optimal with respect to several useful criteria. However, the energy conserving closure yields problematic $Fr$ results, as manifest in particular by invalidity for deep currents (small $h/H$). Connection with realistic time-dependent gravity currents is discussed.

Tuesday, November 24, 2015 8:00AM - 10:10AM –
Session M31 Waves: Internal and Interfacial Waves 312 - Michael R. Allshouse, University of Texas at Austin
8:00AM M31.00001 Internal wave bolus detection and analysis by a Lagrangian coherent structure method1, MICHAEL R. ALLSHOUSE, G. SALVADOR-VIEIRA, HARRY L. SWINNEY, Center for Nonlinear Dynamics and Department of Physics, University of Texas at Austin — The shoaling of vertical mode internal waves on a continental shelf produces boluses, which are trapped regions of fluid that travel up the shelf with the wave. Unlike a propagating solitary wave, these boluses can transport material with the wave. Boluses have been observed to transport oxygen-depleted water and induce rapid changes in temperature both of which have potential ramifications for marine biology. We extend a number of two-layer studies by investigating bolus generation and material transport in continuously stratified fluids. Laboratory experiments are conducted in a 4 m long tank and are complemented by 2-dimensional numerical simulations of the Navier-Stokes equations. The boundaries of a bolus are identified using a Lagrangian based coherent structure method relying on trajectory clustering. The time evolution of material transport by the bolus is investigated as a function of the stratification, wave properties, and the angle of the sloping topography.

1ONR MURI Grant No. N000141110701

8:13AM M31.00002 Geometric focusing of internal waves: Experimental study2, NATALIA SHMAKOVA, LEGI (CNRS & Université de grenoble), EVGENY ERMANYUK2, Laboratoire de Physique ENS de Lyon (CNRS & Université de Lyon), BRUNO VOISIN, JAN-BERT FLÖR, LEGI (CNRS & Université de grenoble) — Mixing of the abyssal ocean plays a decisive role in large-scale ocean circulation and is believed to be caused by the nonlinear breaking of internal tides. Previous studies of two- and three-dimensional cases considered the generation of diverging waves by simple oscillating bodies such as a cylinder (e.g. Mowbray and Rarity 1967) or a sphere (e.g. King et al. 2009, Ermanyuk et al. 2011). We here consider converging waves as generated by a horizontally oscillating torus. The energy focuses and therefore the waves are more susceptible to overturning and breaking. LIF and PIV techniques are used to measure respectively the isopycnal displacement and the velocity. We have considered linear and nonlinear wave generation as a function of the Keulegan-Carpenter number, here adapted to the focusing waves. For small oscillation amplitude strong velocity amplification is observed in the focal zone, consistent with linear theory. Increasing the oscillation amplitude causes nonlinear effects and in particular the generation of higher harmonics and overturning in the focal zone. In addition, the focal zone acts as a wave source. Increase of the Stokes and Reynolds numbers leads to wave turbulence in the focal zone.

2Supported by LabEx Osug@2020 (Investissements d’avenir - ANR10LABX56)

8:26AM M31.00003 Internal wave focusing by annular forcing: theory1, BRUNO VOISIN, LEGI (CNRS & Université de Grenoble), EVGENY ERMANYUK2, Laboratoire de Physique, ENS de Lyon (CNRS & Université de Lyon), NATALIA SHMAKOVA, JAN-BERT FLÖR, LEGI (CNRS & Université de Grenoble) — Among the various mechanisms susceptible of leading to local intensification of internal wave energy, followed by breaking and ultimately mixing, a specific three-dimensional mechanism has received little attention so far: the geometric focusing of waves emitted by concentrically curved annuli. We present a linear theory of annular forcing, based on the assumption that the annulus is slender, namely of negligible local curvature. Both complete focusing by an axisymmetric annulus and partial focusing by a truncated or horizontally modulated annulus are considered. The case of a torus, either thin (i.e. hula-hoop-like) or thick (i.e. doughnut-like), is studied in detail. Focusing is seen to arise in both cases and to yield significant isopycnal slopes, close to overturning, even at low oscillation amplitude. This effect is all the more pronounced as the Stokes number gets higher and the wave structure changes from unimodal to bimodal. Flat circular Gaussian topography is considered next and compared to the earlier work of Bühler & Muller (JFM 2007) and Grisouard & Bühler (JFM 2012). The oceanic relevance of the analysis is finally discussed.

1Supported by LabEx Osug@2020 (Investissements d’avenir - ANR10LABX56)

8:39AM M31.00004 Nonlinear harmonic generation by diurnal tides, SCOTT WUNSCH, Johns Hopkins University — Recent observations from the South China Sea have demonstrated that diurnal tides sometimes generate higher harmonics. Similar harmonic generation has been found in laboratory experiments and numerical simulations of internal wave beams refracting into a pycnocline. Here, a weakly nonlinear theory of internal wave refraction is applied to oceanic diurnal tides in an idealized stratification profile. The harmonic amplitude is calculated as a function of the tidal frequency and the pycnocline characteristics. The results indicate that harmonic generation by nonlinear refraction of diurnal tides is consistent with the South China Sea observations.

8:52AM M31.00005 Convective Excitation of Internal Waves, DANIEL LECAONET, Univ of California - Berkeley, MICHAEL LE BARS, IRPHE, KEATON BURNS, MIT, GEOFFREY VASIL, University of Sydney, ELIOT QUATAERT, University of California — Berkeley, BENJAMIN BROWN, LASP, JEFFREY OISHI, SUNY Farmingdale — We will present a joint experimental & computational study of internal wave generation by convection. First we describe an experiment using the peculiar property of water that its density maximum is at 4°C. A tank of water cooled from below and heated from above develops a cold, convective layer near 4°C at the bottom of the tank, adjacent to a hot, stable stratified layer at the top of the tank. We simulate this setup in 2D using the open-source Dedalus code (dedalus-project.org). Our simulations show that waves are excited from within the convection zone, opposed to at the interface between the convective and stably stratified regions. Finally, we will present 3D simulations of internal wave excitation by convection in a fully compressible atmosphere with multiple density scaleheights. These simulations provide greater freedom in choosing the thermal equilibrium of the system, and are run at higher Rayleigh number. The simulated waves are then compared to analytic predictions of the bulk excitation model.

9:05AM M31.00006 Simultaneous generation and scattering of internal tides by ocean floor topography1, MANIKANDAN MATHUR, Department of Aerospace Engineering, IIT Madras, Chennai - 600036, — Internal waves play a significant role in the global energy budget of the ocean, with internal tides potentially contributing to the conversion of a large amount of mechanical energy into heat in the deep ocean. Several studies in the past decade have investigated internal tide generation and internal tide scattering by ocean floor topography, but by treating them as two separate, independent processes. In this talk, we use the recently developed Green function model (Mathur et al., J. Geophys. Res. Oceans, 119, 2165-2182, 2014), sans the WKB approximation, to quantify the extent to which internal tide generation (scattering) results from barotropic (baroclinic) forcing on small- and large-scale topography in uniform and nonuniform stratifications is modified by the presence of a background baroclinic (barotropic) tide. Results on idealized topography, stratification and forcing will first be presented, followed by a discussion on the relevance of our studies in the real ocean scenario.

1The author thanks the Ministry of Earth Sciences, Government of India for financial support under the Monsoon Mission Grant MM/2014/IND-002.
9:18AM M31.00007 Linear waves in two-layer fluids over periodic bottoms¹, JIE YU, Department of Civil Engineering, and School of Marine and Atmospheric Sciences, Stony Brook University, USA, LEO MAAS, Royal Netherlands Institute for Sea Research, and Institute for Marine and Atmospheric Research Utrecht, Utrecht University — A new, exact Floquet theory is presented for linear waves in two-layer fluids over a periodic bottom of arbitrary shape and amplitude. A method of conformal transformation is adapted. The solutions are given, in essentially analytical form, for the dispersion relation between wave frequency and generalized wavenumber (Floquet exponent), and for the waveforms of free wave modes. The dispersion relation is the analogue of the classical Lamb’s equation for a two-layer fluid over a flat bottom. For internal modes the interfacial wave shows rapid modulation at the scale of its own wavelength that is comparable to bottom wavelength, whereas for surface modes it becomes a long wave carrier for modulating short waves of bottom wavelength. The approximation using a rigid-lid is given. Sample calculations are shown, including the frequencies that are Bragg resonant.

¹Supports to JY by US National Science Foundation (Grant CBET-0845957) and a visitor’s grant of the Netherlands Organisation for Scientific Research (NWO) during the period of this work, are gratefully acknowledged.

9:31AM M31.00008 An experimental investigation of evanescent wave propagation through a turning depth, ALLISON LEE, JULIE CROCKETT, Brigham Young Univ - Provo — One well known method of internal wave generation is tidal flow over oceanic bathymetry. However, in some locations, the natural frequency of the deep ocean is less than the tidal frequency and thus only evanescent waves are generated. While evanescent waves generally dissipate quickly after formation, it has been observed that if these waves travel into a stronger stratification evanescent waves can become a propagating internal waves. Here we present an experimental investigation of this internal wave generation mechanism. Specifically, internal wave energy transfer through a turning depth for a range of topography shapes, stratification profiles, and turning depth locations is explored. Energy transfer from evanescent to propagating waves is found to occur for both linear and exponential stratifications and increases as the turning depth approaches the topography.

9:44AM M31.00009 Nonlinear effects on internal wave reflection at an interface, JOHN MCHUGH, Univ of New Hampshire — Internal waves impinging on an interface that has a sudden change in the first derivative of density are partially reflected, with a corresponding localized mean flow in the vicinity of the interface. Recent weakly nonlinear results have shown that this mean flow is discontinuous at the interface if the flow is inviscid. This recent theory showed that with continuous density at the interface, the linear interfacial conditions are asymptotically consistent, but only approximately balance overall momentum. The next level of nonlinear interfacial terms are now included here. Only several of the numerous terms are found to contribute significantly to the wave reflection, determined by numerical evaluation of all terms. Including these several terms has resulted in improved conservation of total momentum. The resulting nonlinear reflection coefficient depends strongly on wave parameters.

9:57AM M31.00010 Boundary forced internal waves in a non-uniform stratification: diminution and resonant pathways to instability, SASAN JOHN GAHEMAIADI, THOMAS PEACOCK, MIT — We study the surface forcing of internal waves in a non-uniform stratification comprising a relatively thin, highly stratified upper layer sitting atop a deep, weakly stratified lower layer. Such a system theoretically yields a range of transmission features for harmonic boundary forcing, with the response ranging from diminution to resonant growth depending on the degree of coherence between the constitutive waves in the upper stratification layer. A series of laboratory experiments are performed in order to investigate the role of wave interference in tuning wave transmission. We find that the occurrence of destructive interference in the upper stratification naturally yields diminution of the transmitted wave. Conversely, constructive interference results in a notable amplification of the wave field over time scales on the order of the forcing period; the development of nonlinear wave-wave interactions due to wave amplification is observed over longer time scales. Good agreement is obtained between the experimental results and a weakly viscous, long wave model of our system within the linear regime.

Tuesday, November 24, 2015 8:00AM - 10:10AM – Session M32 Drops: Impact on Surfaces 313 - Shmuel Rubinstein, Harvard University

8:00AM M32.00001 Initiation of liquid-solid contact beneath an impacting drop, SHMUDEL RUBINSTEIN, Harvard University, JOHN KOLINSKI, The Hebrew University of Jerusalem — Before an impacting drop contacts the solid surface it must first drain the air beneath it. The drainage process is influenced by non-continuum effects including the non-continuum correction to the lubrication equation that effectively introduce a lubrication layer beneath the impacting liquid.

8:13AM M32.00002 Non continuum effects influence in splashing dynamics, CHRISTOPHE JOSERAND, MATHIEU GALLAIS, Institut D’Alembert, CNRS & UPMC, LAURENT DUCHEMIN, IRPHE — We study numerically the impact of a liquid drop on a solid substrate using a numerical method that solves both the inviscid dynamics for the drop and the lubrication equation for the gas layer. In this framework, that does not depend on the gas pressure, we first characterize the splashing as function of the impact parameters. Then we introduce in the lubrication equation the correction due to non continuum effect in order to investigate the influence of such an effect on the splashing properties. The non-continuum correction affects the time and position of the jet leading to a change of the splashing dynamics with the gas pressure.

8:26AM M32.00003 Characterization of drop impact based on internal flow quantification, ROHAN DE, ASHISH KARN, JOHN NOONAN, BRETT ROSIEJKA, ROGER ARNDT, JIARONG HONG, University of Minnesota — The impact of drops on solid and liquid surfaces and the post-impact phenomena of drop spreading, recoil, shape oscillations etc. have been discussed in numerous previous studies based on the non-dimensional parameters defined with respect to impact velocity, drop size, surface characteristics and liquid properties. Previous studies have characterized the variation of the external features of the post-impact phenomenon and modelled it based on energy considerations including the drop’s overall kinetic energy and potential energy upon deformation. However, the internal flows induced within a drop upon impact has not yet been quantified and thus, internal kinetic energy has largely been ignored. In this study, we have characterized the flow structures developed inside a drop upon impact through Particle Image Velocimetry (PIV). Our study has shown a substantial difference between the overall kinetic energy and the potential energy at maximum deformation, and this difference is observed to correlate well with the internal kinetic energy estimated from our PIV measurements. Further, distinct regimes of vorticity have been observed and a hypothesis has been proposed to explain the occurrence of such modes. Also, these modes are related to the post-impact drop morphology.
on hydrophilic and hydrophobic grains. The energy that is transferred to the sand from that going into droplet deformation. Moreover, in this talk we will compare craters from impact of different velocities for surface (V_1) and impact velocities (V_d), for example by distinguishing the energy that is transferred to the sand from that going into droplet deformation. Moreover, in this talk we will compare craters from impact on hydrophilic and hydrophobic grains.

9:05AM M32.00006 Spreading behavior of a drop upon impact onto a moving surface, HAMED ALMOHAMMADI, ALIDAD AMIRFAZLI, Department of Mechanical Engineering, York University, Toronto, ON, Canada. — Team — Drop impact on a moving surface is of interest in many applications like ink-jet printing and coating. Aside from the usual drop deposition, splashing and rebound regimes, for drop impact onto a moving surface, new regimes such as asymmetric spreading, and tail-lift-off were also seen. A systematic investigation was performed to understand asymmetric spreading. We present an experimental study for water drops (dia. 2.5 mm) impacting a moving plate, and then spreading on a moving surface. A quantitative analysis of the asymmetric spreading was done for a combination of different velocities for surface (V_s = 0 to 10.2 m/s) and drop (V_d = 0.5 to 3.4 m/s). Results show that the edges of drop act differently, if it spreads in the same (downstream) or opposite (upstream) directions of surface motion. Upon impact the drop apex remains stationary, while the downstream lamella either spreads or moves at V_d. Upstream lamella spreads at lower velocity on the moving surface compared to stationary case. Using hydrophilic and hydrophobic surfaces, the effect of the wettability on the lamella shape was also studied. An empirical model was developed to describe the lamella shape which is functions of V_d, V_s and surface wettability for the spreading regime.

9:18AM M32.00007 Numerical Study of High-Speed Droplet Impact on Surfaces and its Physical Cleaning Effects, TOMOKI KONDO, KEITA ANDO, Keio University — Spurred by the demand for cleaning techniques of low environmental impact, one favors physical cleaning that does not rely on any chemicals. One of the promising candidates is based on water jets that often involve fission into droplet fragments and collide with target surfaces to which contaminant particles (often micrometer-size (smaller) stick. Hydrodynamic forces arising from the droplet impact will play a role to remove the particles, but its detailed mechanism is still unknown. To explore the role of high-speed droplet impact in physical cleaning, we solve compressible Navier-Stokes equations with a finite volume method that is designed to capture both shocks and material interfaces in accurate and robust manners. Water hammer and shear flow accompanied by high-speed droplet impact at a rigid wall is simulated to evaluate lifting force and rotating torque, which are relevant to the application of particle removal. For the simulation, we use the numerical code recently developed by Computational Flow Group lead by Tim Colonius at Caltech. The first author thanks Jomela Meng for her help in handling the code during his stay at Caltech.

9:31AM M32.00008 Droplet Impact on Inclined Surfaces for Forensic Bloodstain Analysis1, MARC SMITH, MICHAEL LOCKARD, G. PAUL NEITZEL, Georgia Institute of Technology — During a crime scene investigation, bloodstains are used to infer the size, impact angle, and velocity of the blood droplet that produced the stain. This droplet impact process was explored using experiments and numerical simulations of droplets impacting planar, inclined surfaces with different roughness and wetting properties over a range of Reynolds numbers (1,000 – 5,500) and Weber numbers (200 – 2,000) typical of some forensic applications. Results will be presented showing how the size and shape of the final elliptical bloodstain varies with impact angle and surface roughness. The common forensic practice to predict the impact angle is fairly accurate for near-normal impacts, but it under-predicts the angle for oblique impacts less than about 40 and this effect worsens for rougher surfaces. The spreading of the droplet normal to the impact plane is shown to follow that of a droplet under normal impact as the impact velocity increases. This effect is also lessened by increased surface roughness. The reasons for these effects will be explored using a new GPU-based wavelet-adaptive flow simulation, which can resolve the flows near the solid surface and near the moving contact line of these droplets for the large Reynolds and Weber numbers of these experiments. The reasons for these effects will be explored using a new GPU-based wavelet-adaptive flow simulation, which can resolve the flows near the solid surface and near the moving contact line of these droplets for the large Reynolds and Weber numbers of these experiments. The reasons for these effects will be explored using a new GPU-based wavelet-adaptive flow simulation, which can resolve the flows near the solid surface and near the moving contact line of these droplets for the large Reynolds and Weber numbers of these experiments.

1Supported by the National Institute of Justice.

9:44AM M32.00009 Drop impact of microbubbles suspensions, JUAN MANUEL FERNANDEZ, FRANCISCO CAMPO-CORTES, University of Seville — Drop impact studies have a wide range of applications, many involving multiphase drops. We present here experiments on liquid drops with a suspension of monodisperse microbubbles inside falling by gravity and impacting, at different speeds, smooth and rough surfaces. We use high speed imaging experiments and different microbubbles volume fraction inside the liquid drop to understand the spreading and retraction of these foam droplets. We try to unravel the relevant physics for various packing densities of the grains ρ and impact velocities V, by distinguishing the energy that is transferred to the sand from that going into droplet deformation. Moreover, in this talk we will compare craters from impact on hydrophilic and hydrophobic grains.

31AM M32.00010 Creating a urine black hole, RANDY HURD, Utah State University, ZHAO PAN, Brigham Young University, ANDREW MERITT, Utah State University, JESSE BELDEN, Naval Undersea Warfare Center, Newport, TADD TRUSCOTT, Utah State University — Since the mid-nineteenth century, both enlisted and fashion-conscious owners of khaki trousers have been plagued by unsightly speckle patterns resulting from splash-back while urinating. In recent years, industrial designers and hygiene-driven entrepreneurs have sought to limit this splashing by creating urinal inserts, with the effectiveness of their inventions varying drastically. From this large assortment of inserts, designs consisting of macroscopic pillar arrays seem to be the most effective splash suppressors. Interestingly this design partially mimics the geometry of the water capturing moss Syntrichia caninervis, which exhibits a notable ability to suppress splash and quickly absorb water from impacting rain droplets. With this natural splash suppressor in mind, we search for the ideal urine black hole by performing experiments of simulated urine streams (water droplet streams) impacting macroscopic pillar arrays with varying parameters including pillar height and spacing, draining and material properties. We propose improved urinary insert designs based on our experimental data in hopes of reducing potential embarrassment inherent in wearing khakis.
8:00AM M33.00001 The interaction of a walking droplet and a pillar\(^1\). JOHN BUSH, MIT, DANIEL HARRIS, UNC, PIERRE-THOMAS BRUN, MIT — Droplets may bounce on the surface of a vibrating fluid bath, propelled forward by their own pilot-wave field. With a view to better understanding the interaction of such walking droplets with boundaries, we consider their impact on a submerged circular pillar. While simple scattering events are the norm, as the Faraday threshold is approached, the drop departs the pillar along a path corresponding to a logarithmic spiral. An effective wave force resulting from the pillar is inferred from the spiral, and takes the form of the Coriolis force that would arise in a frame of reference rotating with the instantaneous angular momentum of the drop about the pillar. An electromagnetic analog is explored.

\(^1\)Thanks to the NSF.

8:13AM M33.00002 Visualization of hydrodynamic pilot-wave dynamics\(^1\). VICTOR PROST, JULIO QUINTELA, MIT, DANIEL HARRIS, UNC, PIERRE-THOMAS BRUN, JOHN BUSH, MIT — We present a low-cost device for examining the dynamics of droplets bouncing on a vibrating fluid bath, suitable for educational purposes. Dual control of vibrational and strobing frequency from a cell phone application allowed us to reduce the total cost to 60 dollars. Illumination with inhomogeneous colored light allows for striking visualization of the droplet dynamics and accompanying wave field via still photography or high-speed videography.

\(^1\)Thanks to the NSF

8:26AM M33.00003 Faraday Pilot-Waves: Generation and Propagation\(^2\). CARLOS GALEANO-RIOS\(^2\), IMPA, PAUL MILEWSKI, University of Bath, ANDRE NACHBIN, IMPA, JOHN BUSH, MIT — We examine the dynamics of drops bouncing on a fluid bath subjected to vertical vibration. We solve a system of linear PDEs to compute the surface wave generation and propagation. Waves are triggered at each bounce, giving rise to the Faraday pilot-wave field. The model captures several of the behaviors observed in the laboratory, including transitions between a variety of bouncing and walking states, the Doppler effect, and droplet-droplet interactions.

\(^2\)Instituto Nacional de Matemática Pura e Aplicada (Rio de Janeiro-Brazil)

8:39AM M33.00004 Schrödinger’s drop in a box: wave-droplet interaction in a circular cavity\(^1\). TRISTAN GILET, University of Liege — A walker is a bouncing droplet on a liquid surface that is horizontally propelled by the Faraday waves it generates. This hydrodynamic wave-particle interaction exhibits many quantum-like behaviors. The horizontal trajectory of a walker becomes chaotic when subject to horizontal confinement. Experiments (Harris et al., PRE 2013) reveal that the statistics of the walker position is shaped by the eigenmodes of the cavity in which it is confined, similarly to a quantum particle in a box. In this talk, I introduce a model of the coupling between a bouncing droplet and a surface wave in a cylindrical container. The resulting iterated map captures many features of experimental observations of walker dynamics under confinement. Moreover, the statistical behavior of the map is shown to be surprisingly similar to the solution of Schrödinger equation for a particle in an infinite potential well. This yields an analogy between the Planck constant and the hydrodynamic wave-particle coupling constant.

\(^1\)This work was financially supported by the Actions de Recherches Concertees (ARC) of the Belgium Wallonia-Brussels Federation under Contract No. 12-17/02.

8:52AM M33.00005 Simulations of walking droplets in a harmonic potential\(^1\). KRISTIN DETTMERS, MIT, DAN HARRIS, UNC, Chapel Hill, ANAND OZA, Courant Institute, NYU, RODOLFO ROSALES, JOHN BUSH, MIT — We present the results of a theoretical investigation of the dynamics of droplets walking on the surface of a vibrating bath while subjected to a radial spring force. This system was first explored by Perrard et al. (2014) and Labousse et al. (in preparation), who reported a number of orbital states characterized by a double quantization, in mean radius and angular momentum. Particular attention is given here to characterizing the dependence of the system behavior, specifically the quantization, on the vibrational forcing and the spring force. A number of new exotic orbital states are identified.

\(^1\)Thanks to the NSF

9:05AM M33.00006 Stochastic trajectories of a walking drop in a harmonic potential\(^1\). MASON BIAMONTE, Massachusetts Institute of Technology, ANAND OZA, New York University, ANDRE NACHBIN, IMPA, Rio de Janeiro, JOHN W. M. BUSH, Massachusetts Institute of Technology — Droplets walking on the surface of a vibrating fluid bath have been shown to exhibit certain features of microscopic, quantum systems. Their dynamics is reminiscent of modern extensions of de Broglie’s pilot-wave theory, according to which charged particles interact with a stochastic field. We here present the results of a theoretical investigation of the influence of such a stochastic field on the pilot-wave dynamics of a droplet walking in a simple harmonic potential.

\(^1\)Thanks to the NSF

9:18AM M33.00007 The uncertain trajectory of a pilot-wave\(^1\). ANDRE NACHBIN, IMPA/Brazil — Yves Couder (Paris 7) and coworkers reported on walking droplets on the surface of a vibrating bath. John Bush (MIT) and coworkers also produced laboratory experiments which were compared to theoretical predictions. Both groups discussed the pilot-wave properties previously thought to be peculiar to the microscopic, quantum realm. Of particular interest is the wavelette statistics for pilot-wave dynamics in a confined domain. We present a one dimensional water wave model for a droplet bouncing in a confined domain. The mathematical model makes use of conformal mapping which allows for the presence of submerged barriers. The computational simulations produce tunneling events.

\(^1\)Work supported by CNPq grant 454027/2008-7 and by FAPERJ Cientistas do Nosso Estado grant 102917/2011.
9:31AM M33.00008 Orbiting pairs of walking droplets, EMMANUEL SIEFERT, JOHN W.M. BUSH, MIT, ANAND OZA, Courant Institute, NYU — Droplets may self-propel on the surface of a vibrating fluid bath, pushed forward by their own Faraday pilot-wave field. We present the results of a combined experimental and theoretical investigation of the interaction of pairs of such droplets. Particular attention is given to characterizing the system’s dependence on the vibrational forcing of the bath and the impact parameter of the walking droplets. Observed criteria for the capture and stability of orbital pairs are rationalized by accompanying theoretical developments.

9:44AM M33.00009 Onset of chaos in orbital pilot-wave dynamics, LUCAS TAMBASHCO, MIT, DANIEL HARRIS, UNC, ANAND OZA, Courant Institute - NYU, RODOLFO ROSALES, JOHN BUSH, MIT — We examine the orbital dynamics of droplets self-propelling along the surface of a vibrating bath. Circular orbital motion may arise when the walking droplet is subjected to one of three external force fields, the Coriolis force, a simple harmonic force, and a Coulomb force. Particular attention is given to a theoretical characterization of the onset of chaos that accompanies the destabilization of such circular orbits.

9:57AM M33.00010 Orbiting droplets on a vibrated bath, NARESH SAMPARA, LOIC BURGER, TRISTAN GILET, Univ de Liege, MICROFLUIDICS, UNIVERSITY OF LIEGE TEAM — A millimeter-sized oil droplet can bounce on a vertically vibrated liquid bath for unlimited time. It may couple to the surface wave it emits; leading to horizontal self-propulsion called walking. When several walkers coexist close to one another, they either repel or attract each other, in response to the superposition of the waves they generate. Attraction leads to various bound states, including droplets that orbit around each other. We have experimentally investigated the variety of quantized orbital motions exhibited by two, three and more identical walkers, as a function of forcing acceleration. Each motion is quantified in terms of droplet and wave energy.

Tuesday, November 24, 2015 8:00AM - 10:10AM
Session M35 Drops: General Ballroom B - Tadd Truscott, Utah State University

9:08AM M35.00002 3D tomographic reconstruction of the internal velocity field of an immiscible drop in a shear flow, PAUL KERDRAON, Ecole polytechnique, STUART B. DALZIEL, RAYMOND E. GOLDSTEIN, JULIEN R. LANDEAU, FRANCOIS J. PEAUDECERF, University of Cambridge — We study experimentally the internal flow of a drop attached to a flat substrate and immersed in an immiscible shear flow. Transport inside the drop can play a crucial role in cleaning applications, but also in internal advective mixing, thus increasing the cleaning rate. We used microlitre water-glycerol drops on a hydrophobic substrate. The drops were spherical and did not deform significantly under the shear flow. An oil phase of relative viscosity 0.01 to 1 was flowed over the drop. Typical Reynolds numbers inside the drops were of the order of 0.1 to 10. Using confocal microscopy, we performed 3D tomographic reconstruction of the flow field in the drop. The in-plane velocity field was measured using micro-PIV, and the third velocity component was computed from incompressibility. To our knowledge, this study gives the first experimental measurement of the three-dimensional internal velocity field of a drop in a shear flow. Numerical simulations and theoretical models published in the past 30 years predict a toroidal internal recirculation flow, for which the entire surface flows streamwise. However, our measurements reveal a qualitatively different picture with a two-lobed recirculation, featuring two stagnation points at the surface and a reverse surface flow closer to the substrate. This finding appears to be independent of Reynolds number and viscosity ratio in the ranges studied; we conjecture that the observed flow is due to the effect of surfactants at the drop surface.

9:34AM M35.00004 Harnessing Nanoparticles to Control Evaporation at Liquid-Vapor Interfaces, XIN YONG, Binghamton University, State University of New York — It is well known that nanoparticles with appropriate size and surface chemistry adsorb to liquid-vapor interfaces and consequently modify the mechanical properties of the interfaces. However, little has been explored about the effect of nanoparticles on the heat transfer occurring at the interfaces. Using many-body dissipative particle dynamics (MDPD), we model an evaporating interface with adsorbed nanoparticles. Homogeneous and amphiphilic Janus nanoparticles, which contain hydrophobic and hydrophilic surface regions, are considered in this study. We measure the variation in the evaporation rates of the interface by gradually increasing particle loading until a hexagonal-close-packed monolayer is achieved. We explore the effect of surface chemistry and surface composition of the particles and demonstrate that evaporation can be readily adjusted by tuning the interaction parameters and amphiphilic ratio. Importantly, we observe that the evaporation suppression by adsorbed nanoparticles occurs only when the ambient vapor pressure is low. This study provides a fundamental understanding of the phase transition in multiphase interfacial systems and opens up new routes to additional control over evaporating interfaces.

1 Thanks to the NSF
8:52AM M35.00005 Dissolution of a multicomponent droplet in an immiscible ambient fluid: Application of the distribution law. SHIGAN CHU, ANDREA PROSPERETTI, Johns Hopkins University, ANDREA PROSPERETTI COLLABORATION — A liquid droplet will shrink in an undersaturated ambient liquid medium due to mass transfer across the interface even when the drop liquid is only sparingly soluble in the host liquid. The dissolution rate of a single-component droplet can be accurately predicted by an adaptation of the Epstein-Plesset theory, in which it is assumed that the droplet surface remains at saturation. This hypothesis may be violated in the case of a multi-component droplet, as the more soluble component dissolves faster than the other(s). As a consequence, the droplet surface cannot remain saturated with this component in the later stages of the process. To account for this feature a modified Epstein-Plesset theory is developed on the basis of the distribution law of liquid-liquid solutions. The implications of the theory are illustrated with several examples.

1This study was supported by a grant from BP/The Gulf of Mexico Research Initiative through the University of Texas Marine Science Institute (DROPPS consortium: “Dispersion Research on Oil: Physics and Plankton Studies”). The funders had no role in study

2Also: Department of Applied Sciences and Burgerscentrum, University of Twente, Enschede, The Netherlands

9:05AM M35.00006 Marangoni stresses and drop breakup due to wall shear in a partially filled rotating cylinder. ANDREW WHITE, AZEEZ ODESANYA, THOMAS WARD, Department of Aerospace Engineering, Iowa State University — Drop deformation and breakup in a rotating cylinder partially filled with oil is studied. Experiments using a rotating cylinder are relatively new but we will demonstrate that they are analogous to studies involving tubes and other geometries. Surfactants are added to the drop phase in concentrations at and below the CMC while the rotation rate of the cylinder is varied. Of interest is the effect of interfacial surfactant transport on changes in oil film thickness, drop shape and the onset of tail streaming. Two Biot numbers comparing the importance of surfactant adsorption and desorption to convection of surfactant on the interface are estimated. As shown in previous work on drops and bubbles in tubes, the balance between surface convection, diffusion and adsorption can affect the placement of Marangoni stresses, resulting in thicker or thinner films than with clean surfaces. When surface convection is large, surfactant builds up at the tail and Marangoni stresses can lead to tail streaming when surface tensions are sufficiently small. Experimental results are compared to numerical simulations and to previous work on drops and bubbles in tubes.

1National Science Foundation (#1262718)

9:18AM M35.00007 Optical whispering gallery mode induced interface deformation of liquid droplets. PENG ZHANG, SUNGHWAN JUNG, Department of Biomedical Engineering and Mechanics, Virginia Tech, Blacksburg, VA 24061, YONG XU, ARAM LEE, Department of Electrical and Computer Engineering, Virginia Tech, Blacksburg, VA 24061 — In this study, we analyze nonlinear processes associated with a high-Q factor whispering gallery mode (WGM) in micro-sized liquid droplets, which can be induced by Kerr nonlinearity, thermal effect, and optical radiation pressure. Optical WGM can produce a radiation pressure on the droplet and induce droplet deformations. In our analysis, the droplet deformation will be obtained both analytically by force balance and numerically by the boundary element method. We will show that the nonlinear optofluidic effect is stronger than the Kerr effect and thermal effect. Time scales of these three nonlinear processes will also be estimated and compared. The feasibility of single photon level nonlinearities will be analyzed.

1This research was supported by the National Science Foundation (CBET 1438112).

9:31AM M35.00008 Washing wedges: a capillary instability in a gradient of confinement. LUDOVIC KEISER, REMY HERBAUT, JOSE BICO, ETIENNE REYSSAT, PMMH, ESPCI, CNRS UMR 7636, Paris, France — When a drop of oil is introduced into a gradient of confinement (two glass plates forming a sharp wedge) capillary forces drive it toward the most confined regions, where the solid-fluid contact area is maximal. A surfactant solution subsequently introduced into the wedge undergoes a similar movement until it reaches the oil previously added. If the aqueous phase wets the solid better than the oil, a complex exchange process occurs. The water-oil interface destabilizes, oil fingers grow in the water phase, pinch-off and lead to the formation of the same movement until it reaches the oil previously added. The water-oil interface destabilizes, oil fingers grow in the water phase, pinch-off and lead to the formation of most confined regions, where the solid-fluid contact area is maximal. A surfactant solution subsequently introduced into the wedge undergoes a similar movement until it reaches the oil previously added. If the aqueous phase wets the solid better than the oil, a complex exchange process occurs. The water-oil interface destabilizes, oil fingers grow in the water phase, pinch-off and lead to the formation of the same movement until it reaches the oil previously added. If the aqueous phase wets the solid better than the oil, a complex exchange process occurs. The water-oil interface destabilizes, oil fingers grow in the water phase, pinch-off and lead to the formation of the same movement until it reaches the oil previously added. If the aqueous phase wets the solid better than the oil, a complex exchange process occurs. The water-oil interface destabilizes, oil fingers grow in the water phase, pinch-off and lead to the formation of the same movement until it reaches the oil previously added. If the aqueous phase wets the solid better than the oil, a complex exchange process occurs. The water-oil interface destabilizes, oil fingers grow in the water phase, pinch-off and lead to the formation of the same movement until it reaches the oil previously added. If the aqueous phase wets the solid better than the oil, a complex exchange process occurs. The water-oil interface destabilizes, oil fingers grow in the water phase, pinch-off and lead to the formation of the same movement until it reaches the oil previously added. If the aqueous phase wets the solid better than the oil, a complex exchange process occurs. The water-oil interface destabilizes, oil fingers grow in the water phase, pinch-off and lead to the formation of the same movement until it reaches the oil previously added. If the aqueous phase wets the solid better than the oil, a complex exchange process occurs. The water-oil interface destabilizes, oil fingers grow in the water phase, pinch-off and lead to the formation of the

9:44AM M35.00009 Generation of highly-viscous microjets. YOSHIYUKI TAGAWA, HAJIME ONUKI, YUTO OI, Tokyo Univ of Agr & Tech — An ink-jet printing system (or a liquid-dispensing device) has ecological and cost advantages compared to other printing systems such as offset printing and gravure printing since it requires a small amount of liquids. However, most ink-jet printers are not able to eject high-viscous liquids more than 10 cSt. This limitation severely restricts applications of the ink-jet system. Here we present a novel jet-generation system, discharging jets of high-viscous liquids up to 1,000 cSt. The system employs an impulsive force and converges the force efficiently in order to accelerate the liquid-air interface strongly for generating viscous jets: It consists of a liquid container and a thin tube partially inserted in the liquid. The liquid-air interface inside the thin tube is set deeper than that outside of the tube. We then add an impulsive force on the bottom of the container, leading to the microjet generation inside the thin tube. The pressure field under the impulsive force is estimated using pressure-impulse approach, deriving the jet velocity. The jet velocity is experimentally measured with varying the impulsive force and liquid levels in the tube and the container. It is found that the measured velocities agree with the estimation. Owing to the simple structure of the generation system and an ability for ejecting viscous liquids, it could extend the limits of existing ink-jet printers and may be applicable for next-generation technologies such as 3D printing systems and needle-free injection devices.

1JSPS KAKENHI Grant Number 26709007
9:57AM M35.00001 From bubble bursting to droplet evaporation in the context of champagne aerosols
THOMAS SEON, CNRS, Institut d'Alembert, Paris, ELISABETH GHABACHE, Institut d'Alembert, Paris, ARNAUD ANTKOWIAK, Institut d'Alembert, UPME, Paris, GERARD LIGER-BELAIR, Université de Reims Champagne-Ardenne, France — As champagne or sparkling wine is poured into a glass, a myriad of ascending bubbles collapse and therefore radiate a multitude of tiny droplets above the free surface into the form of very characteristic and refreshing aerosols. Because these aerosols have been found to hold the organoleptic essence of champagne they are believed to play a crucial role in the flavor release in comparison with that from a flat wine for example. Based on the model experiment of a single bubble bursting in idealized champagnes, the velocity, radius and maximum height of the first jet drop following bubble collapse have been characterized, with varying bubble size and liquid properties in the context of champagne aerosols. Using the experimental results and simple theoretical models for drop and surface evaporation, we show that bubble bursting aerosols drastically enhance the transfer of liquid in the atmosphere with respect to a flat liquid surface. Contrary to popular opinion, we exhibit that small bubbles are negative in terms of aroma release, and we underline bubble radii enabling to optimize the droplet height and evaporation in the whole range of champagne properties. These results pave the road to the fine tuning of champagne aroma diffusion, a major issue of the sparkling wine industry.

Tuesday, November 24, 2015 8:00AM - 10:10AM —
Session M36 Bubbles: Cavitation, Acoustics and Biomedical
Ballroom C - Philip L. Marston, Washington State University

8:00AM M36.00001 Direct visualization of microalgae rupture by ultrasound-driven bubbles
ANGELO POMMELLA, IRINA HARUN, ANTONIS POULIPOULOS, JAMES J. CHOI, KLAUS HELLGARDT, VALERIA GARBIN, Imperial College London — Cell rupture induced by ultrasound is central to applications in biotechnology. For instance, cell disruption is required in the production of biofuels from microalgae (unicellular species of algae). Ultrasound-induced cavitation, bubble collapse and jetting are exploited to induce sufficiently large viscous stresses to cause rupture of the cell membranes. It has recently been shown that seeding the flow with bubbles that act as cavitation nuclei significantly reduces the energy cost for cell processing. However, a fundamental understanding of the conditions for rupture of microalgae in the complex flow fields generated by ultrasound-driven bubbles is currently lacking. We perform high-speed video microscopy to visualize the microscopic details of the interaction of Chlamydomonas reinhardtii, microalgae of about 10 μm in size, with ultrasound-driven microbubbles of 2-200 μm in diameter. We investigate the efficiency of cell rupture depending on ultrasound frequency and pressure amplitude (from 10 kPa up to 1 MPa), and the resulting bubble dynamics regimes. In particular we compare the efficiency of membrane rupture in the acoustic microstreaming flow induced by linear oscillations, with the case of violent bubble collapse and jetting.

1V. G. acknowledges partial support from the European Commission (FP7-PEOPLE-2013-CIG), Grant No. 618333.

8:13AM M36.00002 Cavitation-induced damage in soft tissue phantoms by focused ultrasound bursts
POOYA MOVAHED, Univ of Washington — Umbra, WAYNE KREIDER, ADAM D. MAXWELL, MICHAEL R. BAILEY, Univ of Washington, SHELBY B. HUTCHENS, JONATHAN B. FREUND, Univ of Illinois - Urbana — Cavitation in soft tissues, similar to that in purely hydrodynamic configurations, is thought to cause tissue injury in therapeutic ultrasound treatments. Our goal is to generalize bubble dynamics models to represent this phenomenon, which we pursue experimentally with observations in tissue-mimicking polyacrylamide and agarose phantoms and semi-analytic generalization of Rayleigh–Plesset-type bubble dynamics models. The phantoms were imaged with high-speed cameras while subjected to a series of multiple pressure wave bursts, of the kind being considered specifically for burst-wave lithotripsy (BWL). The experimental observations show bubble activation at multiple sites during the initial pulses. After multiple pulses, a further onset of cavitation is observed at some new locations suggesting material failure due to fatigue under cyclic loading. A nonlinear strain-energy with strain hardening is used to represent the elasticity of the surrounding medium. Griffith’s fracture criterion is then applied in order to determine the onset of material damage. The damaged material is then represented as a Newtonian fluid. By assuming that such a decrease in the fracture toughness occurs under cyclic loading, the fatigue behavior observed in the experiments can be reproduced by our model.

1This work was supported by NIH grant NIDDK PO1-DK043881.

8:26AM M36.00003 Toward the development of erosion-free ultrasonic cavitation cleaning with gas-supersaturated water
TATSUMI YAMASHITA, KEITA ANDO, Department of Mechanical Engineering, Keio University — In ultrasonic cleaning, contaminant particles attached at target surfaces are removed by liquid flow or acoustic waves that are induced by acoustic cavitation bubbles. However, the inertial collapse of such bubbles often involve strong shock emission or water hammer by re-entrant jets, thereby giving rise to material erosion. Here, we aim at developing an erosion-free ultrasonic cleaning technique with the aid of gas-supersaturated water. The key idea is that (gaseous) cavitation is triggered easily even with low-intensity sonication in water where gases are dissolved beyond Henry’s saturation limit, allowing us to buffer violent bubble collapse. In this presentation, we report on observations of the removal of micron/submicron-sized particles attached at glass surfaces by the action of gaseous cavitation bubbles under low-intensity sonication.

8:39AM M36.00004 Bubble Cloud Dynamics in a Focused Ultrasound Field
KAZUKI MAEDA, TIM COLONIUS, California Institute of Technology, WAYNE KREIDER, ADAM MAXWELL, BRYAN CUNITZ, MICHAEL BAILEY, University of Washington — In order to characterize and control cloud cavitation in burst wave lithotripsy, modeling and experimental analysis of the acoustic radiation from a spherical bubble cloud interacting with a traveling ultrasound wave of amplitude \(O(10)\) MPa in water are presented. In modeling, bubbles are treated as spherical, radially oscillating cavities under mutual interactions dispersed in continuous liquid phase. We solve the bubble radius evolution and continuous flow field using a WENO-based compressible flow solver. In the solver, Lagrangian point bubbles are coupled with the continuous phase, defined on an Eulerian grid, at the sub-grid-scale through volume averaging techniques. In the experiment, we use a passive cavitation detector to measure acoustic radiation from a cavitation bubble cloud initiated by a focused, traveling ultrasound wave that is generated from a 335 kHz piezoelectric transducer in a water tank. The evolution of the bubble cloud is concurrently captured by a high-speed camera. Based on comparison of modeling and the experiment, we will discuss the effect of initial size and the bubble void fraction of the cloud to the directivity of resulting acoustic radiation.
9:05AM M36.00006 Modes of elastic plates and shells in water driven by modulated radiation pressure of focused ultrasound\(^1\). PHILIP L. MARSTON, TIMOTHY D. DANIEL, Washington State University, AHMAD T. ABAWI, HLS Research, IVARS KIRSTEINS, NUWC — The modulated radiation pressure (MRP) of ultrasound has been used for decades to selectively excite low frequency modes associated with surface tension of fluid objects in water [1, 2]. Much less is known about the excitation of low frequency modes of less compliant metallic objects. Here we use MRP of focused ultrasound to excite resonant flexural vibrations of a circular metal plate in water. The source transducer was driven with a double-sideband suppressed carrier voltage as in [1]. The response of the target (detected with a hydrophone) was at twice the modulation frequency and proportional to the square of the drive voltage. Since the radiation pressure of focused beams is spatially localized, mode shapes could be identified by scanning the source along the target while measuring the target’s response. Additional measurements were done with an open-ended water-filled copper circular cylindrical shell in which resonant frequencies and mode shapes were also identified. These experiments show how focused ultrasound can be used to identify low-frequency modes of elastic objects without direct contact. [1] P. L. Marston and R. E. Apfel, J. Acoust. Soc. Am. 67, 27–37 (1980). [2] S. F. Morse, D. B. Thiesen, and P. L. Marston, Phys. Fluids 8, 3-5 (1996).

\(^1\)Supported by ONR.

9:18AM M36.00007 Adding Some Gas Can Completely Change How an Object in a Liquid-Filled Housing Responds to Vibration J.R. TORCZYNSKI, T.J. O’HERN, J.R. CLAUSEN, Sandia National Laboratories — Adding a little gas can completely change the motion of an object in a liquid-filled housing during vibration. A common system exhibiting this behavior is a spring-supported piston in a liquid-filled cylinder, where the gaps between them are narrow and depend on the piston position. When gas is absent, the piston’s vibrational response is highly overdamped due to forcing viscous liquid through narrow gaps. When a small amount of gas is added, Bjerknes forces cause some of the gas to migrate below the piston. The resulting two gas regions form a pneumatic spring that enables the liquid to move with the piston, with the result that very little liquid is forced through the narrow gaps. This “Couette mode” has low damping and thus has a strong resonance near the frequency given by the pneumatic spring constant and the piston mass. At this frequency, the piston response is large, and the nonlinearity from the gap geometry produces a net force on the piston. This “rectified” force can be many times the piston’s weight and can cause the piston to compress its supporting spring. Sandia National Laboratories is a multi-program laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy’s National Nuclear Security Administration under contract DE-AC04-94AL85000.

9:31AM M36.00008 Linear Stability Analysis of an Acoustically Vaporized Droplet\(^1\), JUNAID SIDDQUI, ADNAN QAMAR, RAVI SAMTANEY, King Abdullah University of Science and Technology — Acoustic droplet vaporization (ADV) is a phase transition phenomena of a superheat liquid (Dodecfluoropentane, \(C_{12}\, \text{F}_{26}\)) droplet to a gaseous bubble, instigated by a high-intensity acoustic pulse. This approach was first studied in imaging applications, and applicable in several therapeutic areas such as gas embolotherapy, thrombus dissolution, and drug delivery. High-speed imaging and theoretical modeling of ADV has elucidated several physical aspects, ranging from bubble nucleation to its subsequent growth. Surface instabilities are known to exist and considered responsible for evolving bubble shapes (non-spherical growth, bubble splitting and bubble droplet encapsulation). We present a linear stability analysis of the dynamically evolving interfaces of an acoustically vaporized micro-droplet (liquid A) in an infinite pool of a second liquid (liquid B). We propose a thermal ADV model for the base state. The linear analysis utilizes spherical harmonics \((\text{Y}_l^m, \text{of degree } m \text{ and order } n)\) and under various physical assumptions results in a time-dependent ODE of the perturbed interface amplitudes (one at the vapor/liquid A interface and the other at the liquid A/liquid B interface). The perturbation amplitudes are found to grow exponentially and do not depend on \(m\).

\(^1\)Supported by KAUST Baseline Research Funds.

9:44AM M36.00009 Shock-acoustic wave emission and cavitation from structured optical fiber tips MILAD MOHAMMADZADEH, SILVESTRE ROBERTO GONZALEZ AVILA, Nanyang Technological University, YIN CHI WAN, XINCAI WANG, HONGYU ZHENG, Singapore Institute of Manufacturing Technology, CLAUS-DIETER OHL, Nanyang Technological University — Fiber optics are used in medicine to deliver laser pulses for microsurgery. Upon absorption of a high-power laser pulse, a thermoelastic wave is emitted from the fiber tip. If a flat cleaved fiber is used, the photoacoustic field comprises a planar compressive shock wave and a tensile diffraction wave from the tip edge. Here we demonstrate that by modifying the geometry of a fiber tip, multiple shock waves can be generated from a single laser pulse. Flat cleaved fibers generate tension only along the fiber axis and with one compression-tension cycle from a laser pulse; however, structured fiber tips cause significant tension both along and off-axis, and generate multiple pressure cycles from a single laser pulse. Fast flash photography reveals that diffraction waves from the edges of the tip structures overlap and generate enough tension to form cavitation clouds. We numerically solve the linear wave equation to model the acoustic transients of structured fiber tips and achieve good agreement with pressure measurements from a fiber optic hydrophone. Multiple shock wave emission from a single laser pulse introduces structured fiber tips as a candidate to deliver histotripsy effects via a surgical catheter for micro-scale ablation of soft tissue.
9:57AM M36.00010 Drop fragmentation by laser-induced cavitation bubbles. S. ROBERTO GONZALEZ-A, Nanyang Technological University, P. JOTR KERRSSSENS, University of Twente, CLAUS-DIETER OHL. Nanyang Technological University — The fragmentation of water droplets by a short laser pulse has received significant attention since the 70’s. The fundamental understanding of droplet vaporization/fragmentation is of interest in laser beam propagation in the atmosphere, in situ analysis of combustion products -a great concern due to its ecological implications- and more recently driven by a better understanding of the drop shaping by a laser pulse which is of interest in the development of extreme ultraviolet (EUV) machines. In this presentation we discuss about the incipient events that lead to the fragmentation of a droplet. When the bubble expands, it stretches the drop into a thin liquid film; this liquid film is eventually ruptured and a shockwave and small droplets are ejected as fast as 4 times the speed of sound in air. Interestingly, we also observe bubbles appear when counter propagating shock waves that rebound from the walls of the drop meet. We also show different fragmentation scenarios recorded with high-speed video, one of them being a jelly fish like liquid film that eventually fragments into smaller drops.

Tuesday, November 24, 2015 8:00AM - 9:44AM
Session M39 General Fluid Dynamics: Obstacles and Boundaries Sheraton Back Bay C - Michael Plesniak, George Washington University

8:00AM M39.00001 Vortex propagation around a wall-mounted obstacle in pulsatile flow1. IAN A CARR, MICHAEL W PLESNIAK, George Washington University — Wall-mounted obstacles are prevalent in nature and engineering applications. Physiological flows observed in human and animal pathologies, such as polyps, can be modeled by flow over a wall-mounted protuberance. Despite their prevalence, studies of wall-mounted obstacles have been restricted to steady (constant velocity) freestream flow. In biological and geophysical applications, pulsatile flow is much more common, yet effects of pulsatility on the wake of a wall-mounted obstacle remain to be extensively studied. This study aims to characterize the complex physics produced in this unsteady, separated flow. Experiments were performed in a low-speed wind tunnel with a set of rotating vanes, which produce the pulsatile inflow waveform. Instantaneous and phase-averaged particle image velocimetry (PIV) results acquired around a hemispherical obstacle are presented and compared. A mechanism based on self-induced vortex propagation, analogous to that in vortex rings, is proposed to explain the observed dynamics of coherent structures. Predictions of the propagation velocity based on analytical expressions for vortex rings in a viscous fluid are compared to the experimentally measured propagation velocity. Effects of the unsteady boundary layer on the observed physics are explored.

1This material is based in part upon work supported by the National Science Foundation under Grant Number CBET-1236351, and GW Center for Biomimetics and Bioinspired Engineering (COBRE).

8:13AM M39.00002 Pressure-Velocity Correlations in the Cove of a Leading Edge Slat. STEPHEN WILKINS, PATRICK RICHARD, JOSEPH HALL, University of New Brunswick — One of the major sources of aircraft airframe noise is related to the deployment of high-lift devices, such as leading-edge slats, particularly when the aircraft is preparing to land. As the engines are throttled back, the noise produced by the airframe itself is of great concern, as the aircraft is low enough for the noise to impact civilian populations. In order to reduce the aeroacoustic noise sources associated with these high lift devices for the next generation of aircraft an experimental investigation of the correlation between multi-point surface-mounted fluctuating pressures measured via flush-mounted microphones and the simultaneously measured two-component velocity field measured via Particle Image Velocimetry (PIV) is studied. The development of the resulting shear-layer within the slat cove is studied for Re≤80,000, based on the wing chord. For low Mach number flows in air, the major acoustic source is a dipole acoustic source tied to fluctuating surface pressures on solid boundaries, such as the underside of the slat itself. Regions of high correlations between the pressure and velocity field near the surface will likely indicate a strong acoustic dipole source. In order to study the underlying physical mechanisms and understand their role in the development of aeroacoustic noise, Proper Orthogonal Decomposition (POD) by the method of snapshots is employed on the velocity field. The correlation between low-order reconstructions and the surface-pressure measurements are also studied.

8:26AM M39.00003 Structural Affects on the Slamming Pressures of High-Speed Planing Craft1. CHRISTINE IKEDA, BRANDON TARAVELLA, University of New Orleans, CAROLYN JUDGE, United States Naval Academy — High-speed planing craft are subjected to repeated slamming events in waves that can be very extreme depending on the wave topography, impact angle of the ship, forward speed of the ship, encounter angle, and height out of the water. The current work examines this fluid-structure interaction problem through the use of wedge drop experiments and a CFD code. In the first set of experiments, a rigid 20-degree deadrise angle wedge was dropped from a range of heights (H≤0.6 m) and while pressures and accelerations of the slam event were measured. The second set of experiments involved a flexible-bottom 15-degree deadrise angle wedge that was dropped from from the same range of heights. In these second experiments, the pressures, accelerations, and strain field were measured. Both experiments are compared with a non-linear boundary value flat cylinder theory code in order to compare the pressure loading. The code assumes a rigid structure, therefore, the results between the code and the first experiment are in good agreement. The second experiment shows pressure magnitudes that are lower than the predictions due to the energy required to deform the structure.

1Funding from University of New Orleans Office of Research and Sponsored Programs and the Office of Naval Research

8:39AM M39.00004 Experimental Investigations of Flow past Spinning Cylinders1. PASQUALE CARLUCCI, LIAM BUCKLEY, IBGAL MEHMEDAGIC, DONALD CARLUCCI, U. S. Army ARDEC, Picatinny Arsenal, NJ, SIVA THANGAM, Stevens Institute of Technology, Experimental investigations of flow past spinning cylinders is presented in the context of their application and relevance to flow past projectiles. A supersonic wind tunnel is used to perform experiments on flow past spinning cylinders that are sting-mounted and oriented such that their axis of rotation is aligned with the mean flow. The experiments cover a Reynolds number range of up to 30000 and rotation numbers of up to 2 (based on cylinder diameter). The experimental validation of the tunnel characteristics and the benchmarking of the flow field in the tunnel are described. The experimental results for spinning cylinders with both rear-mounted and fore-mounted stings are presented along with available computational and experimental findings.

1This work was funded in part by U. S. Army ARDEC
The state of a bending and twisting rod is characterized as a beam undergoing a pure bending deformation. The rods have bending-torsion coupling which induces a torsional deformation during asymmetric bending. A mathematical model is developed to pseudo-analytically evaluate these frequencies. In this study, the flow pattern and dynamics of the pendulums are examined by varying distance among the pendulums, free-stream fluid velocity, density ratio of the fluid and the pendulums. The interaction of an upstream bluff body and pendulums was also considered to investigate how the wake of the bluff body affects oscillations of cylinders. With this experimental setup, the pendulums show various patterns such as stationary mode and out-of-phase oscillation mode.

Golf in the Wind: Exploring the Effect of Wind on the Accuracy of Golf Shots. NEDA YAGHOOBIAN, RAJAT MITTAL, Johns Hopkins University — Golf play is highly dependent on the weather conditions with wind being the most significant factor in the unpredictability of the ball landing position. The direction and strength of the wind alter the aerodynamic forces on a ball in flight, and consequently its speed, distance and direction of travel. The fact that local wind conditions on any particular hole change over times-scales ranging all the way from a few seconds to minutes, hours and days introduces an element of variability in the ball trajectory that is not understood. Any such analysis is complicated by the effect of the local terrestrial and vegetation topology, as well as the inherent complexity of golf-ball aerodynamics. In the current study, we use computational modeling to examine the unpredictability of the shots under different wind conditions over Hole-12 at the Augusta National Golf Club, where the Masters Golf Tournament takes place every year. Of course, the presence of complex vegetation canopy around this hole introduces a spatial and temporal variability in wind conditions that causes uncertainty and even fear among professional golfers. We use our model to examine the effect of wind direction and wind-speed on the accuracy of the golf shots at this hole and use the simulations to determine the key aerodynamic factors that affect the accuracy of the shot.

Resonance wave pumping with surface waves. REMI CARMIGNIANI, ENPC, CALTECH-ENPC COLLABORATION — The valveless impedance pump enables the production or amplification of a flow without the use of integrated mobile parts, thus delaying possible failures. It is usually composed of fluid-filled flexible tubing, closed by solid tubes. The flexible tube is pinched at an off-centered position relative to the tube ends. This generates a complex wave dynamic that results in a pumping phenomenon. It has been previously reported that pinching at intrinsic resonance frequencies of the system results in a strong pulsating flow. A case of a free surface wave pump is investigated. The resonance wave pump is composed of a rectangular tank with a submerged plate separating the water into a free surface and a recirculation rectangular section connected through two openings at each end of the tank. A paddle placed at an off-center position above the submerged plate is controlled in a heaving motion with different frequencies and amplitudes. Similar to the case of valveless impedance pump, we observed that near resonance frequencies strong pulsating flow is generated with almost no oscillations. A linear theory is developed to pseudo-analytically evaluate these frequencies. In addition, larger scale applications were simulated using Smoothed Particle Hydrodynamic codes.

Reconfiguration of a flexible flat plate under snow loading. FRDRICK GOsselIN, Ecole Polytechnique de Montreal — In the present work, we seek to understand the fundamental mechanisms of three-dimensional reconfiguration of plants by studying the large deformation of a flexible rod in fluid flow. Flexible rods made of Polyurethane foam and reinforced with Nylon fibers are tested in a wind tunnel. The rods have bending-torsion coupling which induces a torsional deformation during asymmetric bending. A mathematical model is also developed by coupling the Kirchhoff rod theory with a semi-empirical drag formulation. Different alignments of the material frame with respect to the flow direction and a range of structural properties are considered to study their effect on the deformation of the flexible rod and its drag scaling. Results show that twisting causes the flexible rods to reorient and bend with the minimum bending rigidity. It is also found that the drag scaling of the rod in the large deformation regime is not affected by torsion. Finally, using a proper set of dimensionless numbers, the state of a bending and twisting rod is characterized as a beam undergoing a pure bending deformation.
Numerical simulation of the dynamics of a flexible cantilevered plate subjected to a perpendicular or a parallel fluid flow. FABIEN SANSAS, ERIC LAURRENDEAU, FREDERICK COSSelin, Ecole Polytechnique de Montreal — We focus on the dynamic deformation of a cantilevered flexible plate immersed in a fluid flow. The following two-dimensional numerical study is based on a large deformation beam model solved by finite difference. The fluid is computed by an in-house Arbitrary Eulerian-Lagrangian (ALE) compressible CFD solver. After a validation and verification procedures confirming second order accuracy, two different cases are examined. The first case serves as a validation exercise for the coupling procedure with the flow parallel to the plate: its leading edge is clamped and the trailing end is free. This case models a flapping flag for which the stability of the plate as a function of its mass and flow velocity is investigated. Different parameters are compared to previous numerical and experimental results. The second case is that of a plate clamped at its middle, the flow being perpendicular to its initial shape. The plate deforms by bending in the flow direction. Streamlining and projected area reduction lead to fluid forces reduction but, at some point, dynamic instability occurs. Preliminary results of this instability phenomena are presented, namely the various dynamic behaviours and the trade-offs between streamlined and instability.

8:52AM M40.00005 Beds of reconfigurable angled hairs rectify Stokes flows. JOSE ALVARADO, JENIEVRE, ANDRE MARQUIER, INRIA-PIAF, ECOLE POLYTECHNIQUE — Biological tissues such as intestines, blood vessels, kidneys, and tongues are coated with beds of passive, elongated, hair-like protrusions such as microvilli, hyaluronans, primary cilia, and papillae. Stress from fluid flows can bend deformable hairs, but this reconfiguration can in turn affect confined fluid flows. We investigate this elastoviscous coupling by developing a biomechanical model system of elastomer hair beds subject to shear-driven Stokes flows in a Taylor-Couette geometry. We characterize this system with a theoretical model which shows that reconfiguration of hair beds is controlled by a single elastoviscous number. Hair bending results in an apparently thinning because the hair tips lift toward the base and thus widens the gap between hairs. For a constrained mean vertical momentum flux when velocity is low a power-law function of velocity when velocity is above a threshold. For a constrained mean vertical momentum flux when velocity is low a power-law function of velocity when velocity is above a threshold.

Wave-forced reconfiguration of a 2D artificial canopy. SYLVIE BARSU, DELPHINE DOPPLER, NICOLAS RIVIRE, MICHEL LANCE, LFMA, Universit de Lyon, CNRS UMR 5509 — Blades inside aquatic vegetation canopies show collective motion when submitted to a water flow. Coherent deformation waves might be observed under given flow conditions, which might enhance mass and sediment transfers between the canopy and surrounding flow, thus impacting the plants development. However, more is known on this phenomenon, and little detailed study has been performed on the dynamic reconfiguration of a single array of PVC blades in a wave flume. The oscillations of the blades are imaged while the water level is separately measured using resistive probes. A delayed coherent wave motion is observed within the canopy, as a response to the oscillatory flow. The associated transfer function (amplitude, phase, wave speed) is built by correlating blade displacements and water local velocity time series. The canopy-flow interaction is then modelled by a simple linear damped oscillator chain whose parameters are deduced from experiments.

9:18AM M40.00007 Drag reduction of flexible beams in shear flow. TRISTAN LECLERCQ, EMMANUEL DE LANGRE, Ecole Polytechnique — Flexible systems bending in steady flows are known to experience a lesser drag compared to their rigid counterpart. This effect can be quantified by the Vogel exponent $\nu < 0$ such that the total drag force on the structure increases as $U^{1+\nu}$ instead of the classical quadratic drag-velocity relationship. In this work, an analytical expression of the Vogel exponent of cantilever beams in cross-flow is derived by dimensional analysis, in the case of shear flow with vertical self-similarity. Numerical simulations are also performed and show excellent agreement. The results of the self-similar case provides insight regarding the scaling of drag with respect to the magnitude of the flow in more complex situations. The example of reconfiguration in a Blasius boundary layer is discussed.

9:31AM M40.00008 Drag reduction by reconfiguration of a full tree in a wind tunnel. EMMANUEL DE LANGRE, LOIC TADRIST, TRISTAN LECLERCQ, PASCAL HEMON, XAVIER AMANDOLEX, LabHyX, Ecole Polytechnique — Flexible systems bending in steady flows are known to experience a lesser drag compared to their rigid counterpart. This effect can be quantified by the Vogel exponent $\nu < 0$ such that the total drag force on the structure increases as $U^{1+\nu}$ instead of the classical quadratic drag-velocity relationship. In this work, an analytical expression of the Vogel exponent of cantilever beams in cross-flow is derived by dimensional analysis, in the case of shear flow with vertical self-similarity. Numerical simulations are also performed and show excellent agreement. The results of the self-similar case provides insight regarding the scaling of drag with respect to the magnitude of the flow in more complex situations. The example of reconfiguration in a Blasius boundary layer is discussed.

9:44AM M40.00009 Impacts of the Reconfiguration of Flexible Plants on the Structure of Turbulence and Dispersion of Particles. YING PAN, National Center for Atmospheric Research, ELIZABETH FOLLETT, Massachusetts Institute of Technology, MARCELO CHAMECKI, Pennsylvania State University, HEIDI NEPF, Massachusetts Institute of Technology, SCOTT ISARD, Pennsylvania State University — The effect of a canopy of sufficient density on the flow can be parameterized as a distributed drag calculated as the product of the square of velocity, the canopy density and a drag coefficient. Field and laboratory experimental data suggest that the reconfiguration of flexible plants leads to a power-law dependence of the drag coefficient on velocity. For large-eddy simulation (LES) resolving the canopy layer, by modeling the drag coefficient as a constant when velocity is low and a power-law function of velocity when velocity is above a threshold. For a constrained mean vertical momentum flux at the canopy top, changing the power-law exponent (known as the Vogel number) has negligible effects on LES predictions of the total vertical momentum flux. However, skewness of velocity components, the strength of sweeps and ejections and the fractions of vertical momentum flux transported in different event quadrants are highly sensitive to changes in the Vogel number. These changes in the structure of turbulence have profound impacts on the dispersion of particles within and above the canopy.
9:57AM M40.00010 Reconfiguration of tree architecture under the effect of wind, competition for light, and annual growth. CHRISTOPHE ELOY, IRPHE, Centrale Marseille — In general, trees have self-similar architectures with longer and thicker branches near the roots. Yet, branch segments grown each year always have approximately the same length. This hierarchy of branch lengths and the whole self-similar characteristics results in fact from a continuous process of growth of new branches and shedding of old ones. To assess how such a process affects tree architecture, a functional-structural mechanically-based model of virtual trees is developed. In this model, trees grow into fractal structures to promote efficient photosynthesis in a competing environment. In addition, branch diameters increase in response to wind-induced loads. The results of this model suggest that most self-similar characteristics of trees can be explained by considering that tree are growing structures able to resist mechanical loads due to wind efficiently.

Tuesday, November 24, 2015 10:40AM - 11:15AM —
Session N1 Invited Session: Flows Driven by Libration, Precession and Tides in Planetary Cores Auditorium - Patrice Le Gal, CNRS

10:40AM N1.00001 Flows driven by libration, precession, and tides, in planetary cores, MICHAEL LE BARS, IRPHE, CNRS and Aix-Marseille University, France — Because of gravitational interactions with their companions, the rotational dynamics of planets involve periodic perturbations of their shape, the direction of their rotational vector, and their rotation rate. These perturbations correspond in planetary terms to tides, precession, and longitudinal libration. I will review the flows driven by those mechanical forcings in the liquid iron core of planets, as well as their possible consequences on the planetary dynamics. Special focus will be placed on the associated instabilities and on the various routes toward turbulence recently studied, combining laboratory experiments and numerical simulations. The key point is that mechanical forcings do not provide the energy to the excited flows: They convey part of the available rotational energy and generate intense fluid motions through the excitation of localized jets, shear layers, and resonant inertial modes. Hence, even very small forcings - as it is the case in planets - may have large scale consequences. Mechanically driven flows thus play a fundamental role in planets, providing for instance alternative mechanisms to the standard convective models in explaining the puzzling past magnetic fields recorded on Moon and Mars, and in rationalizing the variety of shapes and rotational states observed in exoplanets.

Tuesday, November 24, 2015 10:40AM - 11:15AM —
Session N34 Invited Session: New Methods for State Estimation and Adaptive Observation of Environmental Flow Systems Leveraging Coordinated Swarms of Sensor Vehicles Ballroom BC - J. Kim, University of California, Los Angeles

10:40AM N34.00001 New methods for state estimation and adaptive observation of environmental flow systems leveraging coordinated swarms of sensor vehicles. THOMAS BEWLEY, UCSD Flow Control & Coordinated Robotics Labs — Accurate long-term forecasts of the path and intensity of hurricanes are imperative to protect property and save lives. Accurate estimations and forecasts of the spread of large-scale contaminant plumes, such as those from Deepwater Horizon, Fukushima, and recent volcanic eruptions in Iceland, are essential for assessing environment impact, coordinating remediation efforts, and in certain cases moving folks out of harms way. The challenges in estimating and forecasting such systems include: (a) environmental flow modeling, (b) high-performance real-time computing, (c) assimilating measured data into numerical simulations, and (d) acquiring in-situ data, beyond what can be measured from satellites, that is maximally relevant for reducing forecast uncertainty. This talk will focus on new techniques for addressing (c) and (d), namely, data assimilation and adaptive observation, in both hurricanes and large-scale environmental plumes. In particular, we will present a new technique for the energy-efficient coordination of swarms of sensor-laden balloons for persistent, in-situ, distributed, real-time measurement of developing hurricanes, leveraging buoyancy control only (coupled with the predictable and strongly stratified flowfield within the hurricane). Animations of these results are available at http://flowcontrol.ucsd.edu/3dhurricane.mp4 and http://flowcontrol.ucsd.edu/katrina.mp4. We also will survey our unique hybridization of the venerable Ensemble Kalman and Variational approaches to large-scale data assimilation in environmental flow systems, and how essentially the dual of this hybrid approach may be used to solve the adaptive observation problem in a uniquely effective and rigorous fashion.

Tuesday, November 24, 2015 11:20AM - 11:40AM —
Session P1 Francoise N. Frenkkel Award Lecture Auditorium - Sutanu Sarkar, University of California, San Diego

11:20AM P1.00001 Francois N. Frenkkel Award Talk: Swimming bacteria at complex interfaces. DIEGO LOPEZ, ERIC LAUGA, LMFA, INSA de Lyon, University of Cambridge — Flagellated bacteria exploiting helical propulsion are known to swim along circular trajectories near surfaces. Past fluid dynamic studies predict this circular motion to be clockwise above a rigid surface (when viewed from inside the fluid) and counter-clockwise below a free surface. Recent experimental investigations showed that complex physicochemical processes at the nearby surface could lead to a change in the direction of rotation, both at solid surfaces absorbing slip-inducing polymers and interfaces covered with surfactants. Motivated by these results, we use a far-field hydrodynamic model to predict the kinematics of swimming near three types of interfaces: clean fluid-fluid interface, slipping rigid wall, and a fluid interface covered by incompressible surfactants. Representing the helical swimmer by a superposition of hydrodynamic singularities, we first show that in all cases the surfaces reorient the swimmer parallel to the surface and attract it, both of which are a consequence of the Stokes dipole component of the swimmer flow field. We then show that circular motion is induced by a higher-order singularity, namely, a rotlet dipole, and that its rotation direction is strongly affected by the boundary conditions at the interface and the bacteria shape.

In collaboration with Eric Lauga, DAMTP, University of Cambridge.

Tuesday, November 24, 2015 11:20AM - 11:40AM —
Session P34 Andreas Acrivos Dissertation Award Lecture Ballroom BC - S. Balachandar, University of Florida
Comparative studies inform how simulations can be scaled up to inform design of utility-scale MHK power plants. An aeroelastic model coupled to LES predicts the dynamical response of rotors to upstream wakes and ambient turbulence. These tiers of turbine models are evaluated to discern tradeoffs in fidelity to physics versus cost. Analogous “actuator methods” are included from experiments of 3 turbines are characterized under varying downstream spacing and lateral offsets. To study effects of unsteady hydrodynamics, the turbines are conducted for axial-flow Marine Hydro-Kinetic (MHK) turbines operating in a flume. This study aims to understand the influence of coherent structures in high Reynolds number wakes on energy extraction and dynamical rotor control processes. In experiments, rotor torque and rotational measurements were obtained 2 diameters downstream of the rotor plane. Flow field structures, as well as wave phase averaged mean velocities contained the laser sheet forming optics and the side looking submersible included a camera and remote focus/aperture electronics. Planar wake measurements were obtained with a stereoscopic particle image velocimetry was carried out in the near-wake region of turbine to understand the mechanism responsible for variation of $C_p$, with rotational speed and free surface proximity. Results revealed presence of slower wake at higher rotational velocities and increased asymmetry in the wake at high free surface proximity.

1:03PM R1.00002 The influence of surface waves on performance characteristics and wake measurements of a horizontal axis marine current turbine$^1$, KAREN FLACK, ETHAN LUST, LUKSA LUZNIK, US Naval Academy — Performance characteristics and wake field results are presented for a 1/25 scale, 0.8 m diameter two bladed horizontal axis marine current turbine. The performance data and 2D PIV measurements were obtained in the 380 ft tow tank at the National Naval Oceanic and Atmospheric Administration of USNavy was towed at a constant speed of $1.68$ m/s with turbine loading resulting in a nominal tip speed ratio of 7. Conditions with two regular waves were investigated. The first wave had a 2.3 second period and 0.18 m wave height, while the second wave had a 2.0 second period and 0.20 m wave height. The waves were selected to have the same energy. Flow field measurements were obtained with an underwater PIV system comprised of two submersible housings. The forward looking submersible contained the laser sheet forming optics and the side looking submersible included a camera and remote focus/aperture electronics. Planar wake measurements were obtained 2 diameters downstream of the rotor plane. Flow field structures, as well as wave phase averaged mean velocities turbulence statistics will be presented and compared to the baseline case without surface waves.

$^1$Work supported by the Office of Naval Research

1:16PM R1.00003 The effect of wall proximity on energy harvesting using a pitching and heaving hydrofoil$^1$, YUNXING SU, MICHAEL MILLER, SHREYAS MANDRE, KENNETH BREUER, Brown University — Measurements of energy harvesting using a heaving and pitching hydrofoil with an aspect ratio 4.5 were taken in three different situations: unconfined, in close proximity to one wall and in close proximity to two walls. Measured lift and torque forces were used with the hydrofoil positions to calculate the efficiency of energy extracted from the flow. There was a modest increase in efficiency with one-wall proximity, while a pronounced increase in efficiency (up to 50%) was realized when the hydrofoil operated between two walls with strong confinement. The lift coefficient of the hydrofoil experienced a noticeable increase in two-wall proximity cases with the strong confinement, which directly contributed to the increase in efficiency of energy harvesting. In the case of two-wall confinement, we found that the optimal frequency and pitch amplitude were higher than those for both the free stream and the one-wall proximity cases. The power extracted from the heaving motion was greatly enhanced by two-wall proximity at high frequencies and high pitch amplitudes and these gains exceeded the additional power required to execute the pitching motion, resulting in the net increase in energy harvesting effectiveness.

$^1$ARPA-e

1:29PM R1.00004 Effect of Free Stream Turbulence on the Performance of a Marine Hydrokinetic Turbine, ASHWIN VINOD, ARINDAM BANERJEE, Lehigh University — The effects of controlled and elevated levels of free stream turbulence on the performance characteristics of a three bladed, constant chord, untwisted marine hydrokinetic turbine is tested experimentally. Controlled homogenous free stream turbulence levels ranging from 3% to ~20% are achieved by employing an active grid turbulence generator that is placed at the entrance of the water channel test section and is equipped with motor controlled winglet shafts. In addition to free stream turbulence, various (turbine) operating conditions such as the free stream velocity and rotational speed are varied. A comparison of performance characteristics that includes the mean and standard deviations of the power coefficient ($C_p$), and thrust coefficient ($C_T$) will be presented and compared to the case of a laminar free stream with FST levels <1%.

1:42PM R1.00005 Experimental/Numerial Comparison of Turbine Efficiency and Wake Structure in an Array of 3 Scale-Model Marine Hydrokinetic Turbines, DANNY SALE, JOHN BATES, BRIAN POLAGYE, ALBERTO ALISEDA, University of Washington, Seattle — Numerical simulations and experiments are conducted for axial-flow Marine Hydro-Kinetic (MHK) turbines operating in a flume. This study aims to understand the influence of coherent structures in high Reynolds number wakes on energy extraction and dynamical rotor control processes. In experiments, rotor torque and rotational position measurements are collected, and the flow field characterized by simultaneous imaging with particle image velocimetry. The performance of 3 turbines are characterized under varying downstream spacing and lateral offsets. To study effects of unsteady hydrodynamics, the turbines are outfitted with open-loop and close-loop feedback controls and compared to the case of uncontrolled rotor. In numerical simulations, different tiers of turbine models are evaluated to discern tradeoffs in fidelity to physics versus cost. Analogous “actuator methods” are included from Large-Eddy-Simulations and Reynolds-Averaged-Navier-Stokes, where the models impose body forces upon the flow field in form of disks, lines, or surfaces. An aeroelastic model coupled to LES predicts the dynamical response of rotors to upstream wakes and ambient turbulence. These comparative studies inform how simulations can be scaled up to inform design of utility-scale MHK power plants.
1:55PM R1.00006 Flow structure in the near wake of a horizontal axis marine current turbine under steady and unsteady inflow conditions, LUKSA LUZNIK, ETHAN LUST, KAREN FLACK, US Naval Academy — Near wake flow field results are presented for a 1/25 scale, 0.8 m diameter (D) two bladed horizontal axis tidal turbine. The 2D PIV measurements were obtained in the USNA 380 ft tow tank for two inflow conditions. The first case had steady inflow conditions, i.e. the turbine was towed at a constant carriage speed (Ucw = 1.68 m/s) and the second case had a constant carriage speed and incoming regular waves with a period of 2.3 seconds and 0.18 m wave height. The underwater PIV system is comprised of two submersible housings with forward looking submersible containing laser sheet forming optics, and the side looking submersible includes a camera and remote focus/aperture electronics. The resulting individual field of view for this experiment was nominally 30x30 cm. Near wake mapping is accomplished by “tilting” individual fields of view with approximately 5 cm overlap. All measurements were performed at the nominal tip speed ratio (TSR) of 7. The mapping is accomplished in a vertical streamwise plane (x-z plane) centered on the turbine nacelle and the image pair captures were phase locked to two phases: reference blade horizontal and reference blade vertical. Results presented include distribution of mean velocities, Reynolds shear stresses, 2D turbulent kinetic energy. The discussion will focus on comparisons between steady and unsteady case. Further discussion will include comparisons between the current high resolution PIV measurements and the previous point measurements with the same turbine at different lateral planes in the same flow conditions.

2:08PM R1.00007 Wake structure of axial-flow hydrokinetic turbines in tri-frame arrangement, SAURABH CHAWDHARY, Saint Anthony Falls Laboratory, Department of Mechanical Engineering, University of Minnesota, XIAOLEI YANG, CRAIG HILL, ALI KHOSRONEJAD, MICHELE GUALA, FOTIS SOTIROPOULOS, Saint Anthony Falls Laboratory, Department of Civil, Environmental, and Geo-Engineering, University of Minnesota — Marine and hydro-kinetic (MHK) energy hold promise for future of sustainable energy generation. Tri-frame of turbines, three turbines mounted on vertices of a triangle, are an effective way to build a power producing array of hydrokinetic turbines in marine environment. Large eddy simulation (LES) is used to simulate the flow past a tri-frame and characterize its wake. Full geometry of all three turbines in the tri-frame is resolved using the Curvilinear Immersed Boundary (CURVIB) method of Kang et al. (2011). High fidelity solution of flow field is obtained owing to the inclusion of detailed geometry of the turbines. Excellent agreement is obtained with the experiments conducted in a flume at Saint Anthony Falls Laboratory (SAFL). The wake evolution of the three turbines is compared to that of an isolated single turbine. The differences in wake dynamics are highlighted to elucidate the importance of turbine wake interaction in an array. The simulations indicate lower levels of TKE and lower levels of momentum deficit in the wake of the upstream turbine of tri-frame compared to the other turbines. Analysis of the far wake recovery is useful for the optimal MHK array design.

1This work was supported by NSF grant IIP-1318201. The simulations were carried out at the Minnesota Supercomputing Institute.

2:21PM R1.00008 ABSTRACT WITHDRAWN —

2:34PM R1.00009 Development of a towing tank PIV system and a wake survey of a marine current turbine under steady conditions, ETHAN LUST, LUZNIK, KAREN FLACK, U.S. Naval Academy — A submersible particle image velocimetry (PIV) system was designed and built at the U.S. Naval Academy. The system was used to study the wake of a scale-independent horizontal axis marine current turbine. The turbine is a 1/25th scale model of the U.S. National Renewable Energy Laboratory’s Reference Model 1 (RM1) tidal turbine. It is a two-bladed turbine measuring 0.8 m in diameter and featuring a NACA 63-618 airfoil cross-section. The wake survey was conducted over an area extending 0.25D forward of the turbine tip path to 2.0D aft to a depth of 1.0D beneath the turbine output shaft in the streamwise plane. Each field of view was approximately 30 cm by 30 cm, and each overlapped the adjacent fields of view by 5 cm. The entire flow field was then reconstructed by registering the resultant vector fields together into a single field of investigation. Results include the field of investigation from a representative case, for the mean velocity field averaged over approximately 1,000 realizations, and turbulent statistics including turbulence intensities, Reynolds shear stresses, and turbulent kinetic energy.

1This research was funded by the Office of Naval Research.

2:47PM R1.00010 Experimental and Numerical Study on Performance of Ducted Hydrokinetic Turbines with Pre-Swirl Stator Blades, ANDREW GISH, United States Naval Academy — Ducts (also called shrouds) have been shown to improve performance of hydrokinetic turbines in some situations, bringing the power coefficient (Cp) closer to the Betz limit. Here we investigate optimization of the duct design as well as the addition of stator blades upstream of the turbine rotor to introduce pre-swirl in the flow. A small scale three-bladed turbine was tested in a towing tank. Three cases (bare turbine, with duct, and with duct and stators) were tested over a range of flow speeds. Important parameters include duct cross-sectional shape, blade-duct gap, stator cross-sectional shape, and stator angle. For each test, Cp was evaluated as a function of tip speed ratio (TSR). Experimental results were compared with numerical simulations. Results indicate that ducts and stators can improve performance at slower flow speeds and lower the stall speed compared to a bare turbine, but may degrade performance at higher speeds. Ongoing efforts to optimize duct and stator configurations will be discussed.

3:00PM R1.00011 Design of Bi-Directional Hydrofoils for Tidal Current Turbines, IVAYLO NEDYALKOV, MARTIN WOSNIK, University of New Hampshire — Tidal Current Turbines operate in flows which reverse direction. Bi-directional hydrofoils have rotational symmetry and allow such turbines to operate without the need for pitch or yaw control, decreasing the initial and maintenance costs. A numerical test-bed was developed to automate the simulations of hydrofoils in OpenFOAM and was utilized to simulate the flow over eleven classes of hydrofoils comprising a total of 700 foil shapes at different angles of attack. For promising candidate foil shapes physical models of 75 mm chord and 150 mm span were fabricated and tested in the University of New Hampshire High-Speed Cavitation Tunnel (HiCaT). The experimental results were compared to the simulations for model validation. The numerical test-bed successfully generated simulations for a wide range of foil shapes, although, as expected, the k-ω-SST turbulence model employed here was not adequate for some of the foils and for large angles of attack at which separation occurred. An optimization algorithm is currently being coupled with the numerical test-bed and additional turbulence models will be implemented in the future.
Phase Resolved Angular Velocity Control of Cross Flow Turbines

Benjamin Strom, Steven Brunton, Brian Polagye, Univ of Washington — Cross flow turbines have a number of operational advantages for the conversion of kinetic energy in marine or fluvial currents, but they are often less efficient than axial flow devices. Here a control scheme is presented in which the angular velocity of a cross flow turbine with two straight blades is prescribed as a function of azimuthal blade position, altering the time-varying effective angle of attack. Flume experiments conducted with a scale model turbine show approximately an 80% increase in turbine efficiency versus optimal constant angular velocity and constant resistive torque control schemes. Torque, drag, and lateral forces on one- and two-bladed turbines are analyzed and interpreted with bubble flow visualization to develop a simple model that describes the hydrodynamics responsible for the observed increase in mean efficiency. Challenges associated with implementing this control scheme on commercial-scale devices are discussed. If solutions are found, the performance increase presented here may impact the future development of cross flow turbines.

Session R2 Detonation and Explosion 101 - Ju Zhang, Florida Institute of Technology

12:50PM R2.00001 Detonation Initiation with Thermal Deposition due to Pore Collapse in Energetic Materials - Towards the Coupling between Micro- and Macroscale

Ju Zhang, Florida Institute of Technology, Thomas Jackson, University of Florida — Initiation of detonation through thermal power deposition due to pore collapse in energetic materials (such as HMX) is studied numerically by solving the reactive Euler equations. The thermal power deposition model is partially based on previous results of direct simulations of pore collapse. The thermal deposition time scales obtained from the pore collapse model are significantly longer than acoustic time scale. It is found here that a critical size of hot spots exists, and when hot spots exceed the critical size, direct initiation of detonation upon ignition seems independent of power input, and is achieved even with low power input. On the other hand, when hot spots are below the critical size, the ignition does not lead to detonation. However, if the thermal deposition time scale is increased, a scenario different than pore collapse, such that it is on the acoustic time scale, detonation does arise, a scenario corresponding to the so-called “explosion in explosion”. A time scale criterion for direct initiation of detonation is then proposed and demonstrated with numerical simulations. It is proposed that if the chemical reaction time scale is shorter than the acoustic time scale at ignition, the ignition will lead to a direct initiation of detonation.

1:03PM R2.00002 Experimental investigation of turbulent mixing in post-explosion environment

Josh Smith, Michael Hargather, New Mexico Tech — Experiments are performed to investigate the turbulent mixing of product gases and the ambient environment in a post-explosion environment. The experiments are performed in a specially constructed shock tunnel where thermitite-enhanced explosions are set off. The explosives are detonated at one end of the tunnel, producing a one-dimensional shock wave, and product gas expansion which moves toward the open end of the tunnel. Optical diagnostics are applied to study the shock wave motion and the turbulent mixing of the gases after the detonation. Results are presented for schlieren, shadowgraph, and interferometry imaging of the expanding gases with simultaneous pressure measurements. An imaging spectrometer is used to identify the motion of product gas species. Results show varying shock speed with thermitite mass and the identification of turbulent mixing regions.

1:16PM R2.00003 Exhaust Gas Emissions from a Rotating Detonation-wave Engine

Kazhikathra Kailasanath, Douglas Schwer, U.S. Naval Research Laboratory — Rotating detonation-wave engines (RDE) are a form of continuous detonation-wave engines. They potentially provide further gains in performance than an intermittent or pulsed detonation-wave engine (PDW). The overall flow field in an idealized RDE, primarily consisting of two concentric cylinders, has been discussed in previous meetings. Because of the high pressures involved and the lack of adequate reaction mechanisms for this regime, previous simulations have typically used simplified chemistry models. However, understanding the exhaust species concentrations in propulsion devices is important for both performance considerations as well as estimating pollutant emissions. Progress towards addressing this need will be discussed in this talk. In this approach, an induction parameter model is used for simulating the detonation but a more detailed finite-chemistry model including NOx chemistry is used in the expansion flow region, where the pressures are lower and the uncertainties in the chemistry model are greatly reduced. Results show that overall radical concentrations in the exhaust flow are substantially lower than from earlier predictions with simplified models. The performance of a baseline hydrogen/air RDE increased from 4940 s to 5000 s with the expansion flow chemistry, due to recombination of radicals and more production of H2O, resulting in additional heat release.

1:29PM R2.00004 Investigation of detonation propagation through an array of random discrete energy sources using the reactive Burgers’ analog

Giuseppe Di Labbio, Charles Basenga Kiyanda, Concordia University, Xiaocheng Mi, Andrew Jason Huggins, McGill University, Nikolaos Nikiforakis, University of Cambridge, HoI Dicker Ng, Concordia University — For a homogeneous reactive medium such as a combustible gaseous mixture, the Chapman-Jouguet (CJ) condition is closely approximated by a local Chapman-Jouguet (CJ) condition. Although the CJ condition was originally formulated for a wave propagating in homogenous media at constant velocity, it has been posited that this condition may also determine the average detonation velocity in heterogeneous media. This work aims to test the applicability of the CJ condition to heterogeneous media on the one-dimensional reactive Burgers equation, a tractable analog to the reactive Euler equations, with the reaction governed by a generic rate law. In this work, the entire energy source, originating from random energy content, randomly distributed throughout space such that the total energy release is equivalent to that of a homogeneous medium with constant energy density. The equations are solved using a second-order finite volume approach with an exact Riemann solver. The evolution of the discrete detonation is tracked over a long duration and its average propagation velocity is computed. In all cases, the average detonation velocity was found to be in agreement with the velocity predicted by the CJ condition for the equivalent homogeneous system.

1:42PM R2.00005 Numerical investigation of the density effect in modeling detonation propagation in high explosives

Carlos Chiquete, Chad D. Meyer, Mark Short, Los Alamos National Laboratory — Detonation Shock Dynamics (DSD) is an asymptotically-derived detonation propagation model used in engineering models of high explosive (HE) performance. The method is based on the limit where the detonation reaction zone length and time scales are small in relation to the much larger geometry in which the HE is embedded. The intrinsic DSD propagation law (functionally relating the surface normal velocity and curvature) for each HE is typically calibrated to simplified geometry tests where steady-state front velocities and shapes are measured. This relationship is necessarily a function of the experimental conditions and is thus limited in scope. For HEs with variable pressing or casting density, a particular need exists for calibrations sensitive to this variability. However, there is little constraint on how the density effect is specifically incorporated into the fitting procedure. To investigate this issue, shock-attached calculations in simple slab or cylindrical geometries are performed for varying initial density for a numerical explosive model with a realistic equation of state. The steady-state detonation velocities, front shapes and the resulting DSD calibration of this generated data are analyzed as function of the applied HE density.
1:55PM R2.00006 Dynamics of galloping detonations: inert hydrodynamics with pulsed energy release

MATEI I. RADULESCU, University of Ottawa, JOSEPH E. SHEPHERD, Caltech — Previous models for galloping and cellular detonations of Ulyanitski, Vasil’ev and Higgins assume that the unit shock decay or cell can be modeled by Taylor-Sedov blast waves. We revisit this concept for galloping detonations, which we model as purely inert hydrodynamics with periodically pulsed energy deposition. At periodic time intervals, the chemical energy of the non-reacted gas accumulating between the lead shock and the contact surface separating reacted and non reacted gas is released nearly instantaneously. In between these pulses, the gas evolves as an inert medium. The resulting response of the gas to the periodic forcing is a sudden gain in pressure followed by mechanical relaxation accompanied by strong shock waves driven both forward and backwards. It is shown that the decay of the lead shock in-between pulses follows an exponential decay, whose time constant is controlled by the frequency of the energy deposition. Moreover, the average speed of the lead shock is found to agree with 2 percent to the ideal Chapman-Jouguet value, while the large scale dynamics of the wave follows closely the ideal wave form of a CJ wave trailed by a Taylor expansion. When friction and heat losses are accounted for, velocity deficits are predicted, consistent with experiment.

1Work performed while MIR was on sabbatical at Caltech

2:08PM R2.00007 Detonation Propagation through Nitromethane Embedded Metal Foam

BRANDON LIEBERTHAL, University of Illinois at Urbana-Champaign, WARREN R. MAINES, Sandia National Laboratories, D. SCOTT STEWART, University of Illinois at Urbana-Champaign — There is considerable interest in developing a better understanding of dynamic behaviors of multicomponent systems. We report results of Eulerian hydrodynamic simulations of shock waves propagating through metal foam at approximately 20% relative density and various porosities using a reactive flow model in the ALE3D software package. We investigate the applied pressure and energy of the shock wave and its effects on the fluid and the inert material interface. By varying pore sizes, as well as metal impedance, we predict the overall effects of heterogeneous material systems at the mesoscale. In addition, we observe a radically expanding blast front in these heterogeneous models and apply the theory of Detonation Shock Dynamics to the convergence behavior of the lead shock.

2:21PM R2.00008 Theory of weakly nonlinear multidimensional detonations

LUIZ FARIA, KAUST, MIT, ASLAN KASIMOV, KAUST, RODOLFO ROSALES, MIT — We derive an asymptotic model for the dynamics of weakly nonlinear multi-dimensional detonations from the compressible reactive Navier-Stokes equations. It is assumed that activation energy is large, heat release is small, evolution is slow, and $\gamma - 1$ is small. The resultant model in 2D in dimensionless form is given by

\begin{align*}
\frac{du}{dx} + u\frac{dv}{dx} + v\frac{dw}{dx} &= -\frac{1}{2}T_x + \frac{\mu}{\kappa} \frac{d^2u}{d\lambda^2} \\
\frac{dv}{dx} &= \frac{u}{\kappa} \\
\lambda_x &= -k(1 - \lambda)e^{\theta T} - d\lambda_x \\
\kappa T_x + T_k &= u + q\lambda + g(\lambda_x)
\end{align*}

where $u, v$ is the velocity field, $T$ is the temperature, and $\lambda \in [0, 1]$ is the reaction progress variable, $q$ heat release, and $\mu, \kappa, d$ are coefficients of viscosity, heat conduction, and diffusion, respectively. This system is a generalization of the models of small disturbance unsteady transonic flow, weakly nonlinear acoustics (Zabolotskaya-Khokhlov (ZK) equation), and water waves (dispersionless Kadomtsev-Petviashvili (KP) equation). The model predicts regular and irregular multi-dimensional patterns, and in 1D exhibits transition from steady and stable traveling waves to oscillatory traveling waves through a Hopf bifurcation as $\theta$ is increased. Period-doubling bifurcations leading to chaos are also observed.

2:34PM R2.00009 On the quasi-one dimensional structure of the cellular detonation in a two-dimensional duct

C.M. UYEDA, M. KUROSAKA, A. FERRANTE, William E. Boeing, Dept. of Aeronautics & Astronautics, University of Washington, Seattle — We performed numerical simulations of cellular detonations in a 2D duct to establish the validity of the one-dimensional ZND model. The detonation was calculated by solving the Euler equations with a WENO-TVD numerical method using adaptive mesh refinement and a detailed chemical reaction mechanism. The results show that the properties of the ZND model of a 2H2-O2-7Ar reaction are very close to the results of the simulations initiated using three different methods for the area-averaged properties and the properties of particles tracked along their pathlines. Disagreements between the particle properties and the ZND model are greatest near the detonation front where the transverse wave and Mach stem introduce larger jumps in the flow properties than the ZND model predicts. The particle pathlines also exhibit a quasi one-dimensional motion downstream from the detonation front which is supported by the quick decay in the particles’ velocity ratio of the vertical to horizontal velocity components, in the reference frame attached to the detonation front. These findings show the quasi-one-dimensional nature of 2D detonations and the applicability of the ZND model.

2:47PM R2.00010 Flame Acceleration and Transition to Detonation in Channels

GABRIEL GOODWIN, RYAN HOUMI, ELAINE ORAN, Univ of Maryland-College Park — Two-dimensional numerical simulations of a confined, homogeneous, chemically reactive gas were used to compute and catalog interactions leading to deflagration-to-detonation transition (DDT). The geometrical configuration was a long rectangular channel with regularly spaced obstacles and adiabatic boundary conditions on all of the surfaces. The channel contained a stoichiometric mixture of ethylene-oxygen at 300 K and one atom that was ignited with a circular flame. The reactive Navier-Stokes equations were solved on an adapting grid by a high-order Godunov algorithm. The channel height was fixed at 0.32 cm and obstacle heights created blockage ratios ranging from 0.8 to 0.05, where the blockage ratio is defined as the obstacle height divided by the channel height. The computations show the development of a turbulent flame, the creation of shocks, shock-flame interactions, and a host of fluid and chemical-fluid instabilities. The result is an accelerating flame and eventual DDT in unburned, but shock-heated, material. Several DDT mechanisms were observed; these will be shown and discussed, with an emphasis on several new observations related to shock interactions.

This work is supported by the Office of Naval Research.

3:00PM R2.00011 Liquid explosions induced by X-ray laser pulses

CLAUDIU STAN, HARTWAN LAKSMONO, RAYMOND SIERRA, TREVOR MCGUEN, DESPINA MILATHIANAKI, JASON KOGLIN, THOMAS LANE, MARC MESSERSCHMIDT, GARTH WILLIAMS, MATT HAYES, SERGE GUILLET, SLAC National Accelerator Laboratory, SABINE BOTHA, KAROL NASS, ILME SCHLICHTLING, ROBERT SHOEMAN, Max-Planck Institute for Medical Research, HOWARD STONE, Princeton University, Sbastien Boutet, SLAC National Accelerator Laboratory — Sudden generation and release of enough energy to vaporize matter are encountered in systems that range from supernovae explosions and asteroid impacts to applications in fusion energy generation, materials processing, and laser surgery. Understanding these strong explosions is important to both fundamental science and technical applications. We studied a new type of microexplosion, induced by absorption of X-ray pulses from a free-electron laser in micron-sized drops and jets of water. These explosions are related to, but different from, those observed in experiments performed with optical lasers. Unlike explosions caused by optical lasers, X-ray laser explosions produce symmetric expansion patterns that are simpler to rationalize. The release of energy initially concentrated in a small region inside drops and jets leads to ballistic vapor flow and inertial liquid flow. The kinematics of these flows indicates that the conversion of the energy deposited by X-rays into flow has a scaling that is similar to the one encountered in shock waves.
12:50PM R3.00001 Quantifying Numerical Dissipation due to Filtering in Implicit LES, FRANCOIS CADIEUX, Johns Hopkins University, JULIAN ANDRZEJ DOMARADZKI, University of Southern California — Numerical dissipation plays an important role in LES and has gained rise to the widespread use of implicit LES in the academic community. Recent results demonstrate that even with higher order codes, the use of stabilizing filters can act as a source of numerical dissipation strong enough to compare to an explicit subgrid-scale model (Cadieux et al, JFE 136-6). The amount of numerical dissipation added by such filtering operation in the simulation of a laminar separation bubble is quantified using a new method developed by Schranner et al, Computers & Fluids 114. It is then compared to a case where the filter is turned off, as well as the subgrid-scale dissipation that would be added by the σ model. The sensitivity of the method to the choice of subdomain location and size is explored. The effect of different derivative approximations and integration methods is also scrutinized. The method is shown to be robust and accurate for large subdomains. Results show that without filtering, numerical dissipation in the high order code is negligible, and that the filtering operation at the resolution considered adds substantial numerical dissipation in the same regions and at a similar rate as the σ subgrid-scale model would.

1 NSF grant CBET-1233160

1:03PM R3.00002 Pressure-Velocity-Scalar Filtered Mass Density Function for Large Eddy Simulation of Compressible Turbulent Flow, ARASH Nouri GHEIMASSI, PEYMAN GIVI, University of Pittsburgh, MEHDI B. NIK, Stanford University, STEPHEN B. POPE, Cornell University — A new model is developed which accounts for the effects of subgrid scale pressure in the context of the filtered density function (FDF) formulation. This results in a pressure-velocity-scalar filtered mass density function (PVS-FMDF), which is suitable for large eddy simulation of compressible turbulence. Following its mathematical definition, an exact transport equation is derived for the PVS-FMDF. This equation is modeled in a probabilistic manner by a system of stochastic differential equations (SDEs). The consistency and the predictive capability of the model are established by conducting LES of a three-dimensional compressible mixing layer, and comparison with direct numerical simulation (DNS) data.

1:16PM R3.00003 ABSTRACT WITHDRAWN –

1:29PM R3.00004 ABSTRACT WITHDRAWN –

1:42PM R3.00005 Large-eddy simulation of a spatially-evolving turbulent mixing layer, FRANCESCO CAPUANO, PIETRO CATALANO, ANDREA MASTELLONE, Centro Italiano Ricerche Aerospaziali (CIRA) — Large-eddy simulations of a spatially-evolving turbulent mixing layer have been performed. The flow conditions correspond to those of a documented experimental campaign (Delville, Appl. Sci. Res. 1994). The flow evolves downstream of a splitter plate separating two fully turbulent boundary layers, with Reθ = 2900 on the high-speed side and Reθ = 1200 on the low-speed side. The computational domain starts at the trailing edge of the splitter plate, where experimental mean velocity profiles are prescribed; white-noise perturbations are superimposed to mimic turbulent fluctuations. The fully compressible Navier-Stokes equations are solved by means of a finite-volume method implemented into the in-house code SPARK-LES. The results are mainly checked in terms of the streamwise evolution of the vorticity thickness and averaged velocity profiles. The combined effects of inflow perturbations, numerical accuracy and subgrid-scale model are discussed. It is found that excessive levels of dissipation may damp inlet fluctuations and delay the virtual origin of the turbulent mixing layer. On the other hand, non-dissipative, high-resolution computations provide results that are in much better agreement with experimental data.

1:55PM R3.00006 How Many Grid Points Are Required for Time Accurate Simulations?, AYABOE EDOH, ANN KARAGOZIAN, University of California, Los Angeles, NATHAN MUNDIS, ERC, Inc., VENKATESWARAN SANKARAN, Air Force Research Laboratory — Grid resolution is a key element in a numerical discretization scheme’s ability to accurately capture complex fluid dynamics phenomena encountered in LES and DNS calculations. The fundamental question to be asked concerns the minimum number of points required to represent relevant flow phenomena such as vortex and acoustic wave propagation. The answer is naturally dependent upon the choice of numerical scheme but it is also influenced by the modal content of the fluid dynamics. Specifically, this study looks at high-order and optimized spatial stencils and their associated dispersion and dissipation characteristics coupled with several time integration schemes. Scheme stabilization is also addressed with respect to artificial dissipation and filtering techniques. The theoretical investigations based on von Neumann analysis are substantiated by calculations of pure mode and multiple mode wave propagation problems, isentropic vortex propagation and the DNS of Taylor Green vortex transition, all of which are used to establish the accuracy properties of the schemes.

1 Distribution A: Approved for public release, distribution unlimited. Supported by AFOSR (PM: Dru. F. Fahroo and Chiping Li)
3 Edoh, Karagozian, Merkle and Sankaran, AIAA 2015-0284

2:08PM R3.00007 An improved numerical scheme for a dynamic LES model, BRANISLAV BASARA, AVL List GmbH — Dynamic LES models are very popular nowadays. There are clear advantages in computing rather than prescribing the unknown coefficient that appear in a subgrid-scale model for Large Eddy Simulation (LES). Whatever is the origin of the model; these dynamic models usually impair the convergence rate when compared to the standard and well-known Smagorinsky model. Although most of them provide physical bounds for the non-dimensional constant and with that numerically reasonable values for the unknown sub grid-scale stresses, strong gradients of these terms that can appear across the flow may introduce additional difficulties to the numerical simulations. In the present discretization scheme, we use a deferred-correction approach for the subgrid-scale stresses with the additional correction term, which all together ensure a more stable solution, but without negative effects on the accuracy. As a representative dynamic LES model, we choose the coherent structure model of Kobayashi (2005). Nevertheless, the conclusions derived here are applicable to other dynamic models as well.
2:21PM R3.00008 Unstructured finite element simulations of compressible phase change phenomena1 — EHSAN SHAMS, FAN YANG, YU ZHANG, ONKAR SAJNI, MARK SHEPHARD, ASSAD OBERAI, Rensselaer Polytechnic Inst, SCIENTIFIC COMPUTATION & RESEARCH CENTER (SCOREC) TEAM — Modeling interactions between compressible gas flow and multiple combusting solid objects, which may undergo large deformations, is a problem with several challenging aspects that include, compressible turbulent flows, shocks, strong interfacial fluxes, discontinuous fields and large topological changes. We have developed and implemented a mathematically consistent, computational framework for simulating such problems. Within our framework the fluid is modeled by solving the compressible Navier-Stokes equations with a stabilized finite element method. Turbulence is modeled using large eddy simulation, while shocks are captured using discontinuity capturing methods. The solid is modeled as a hyperelastic material, and its deformation is determined by writing the constitutive relation in a rate form. Appropriate jump conditions are derived from conservation laws applied to an evolving interface, and are implemented using discontinuous functions at the interface. The mesh is updated using the Arbitrary Lagrangian Eulerian (ALE) approach, and is refined and adapted during the simulation. In this talk we will present this framework and will demonstrate its capabilities by solving canonical phase change problems.

1We acknowledge the support from Army Research Office (ARO) under ARO Grant # W911NF-14-1-0301.

2:34PM R3.00009 Large Eddy Simulation of Supersonic Cold Flow in Ramp-Cavity Combustor with Fuel Injector — ZIA GHIAHI, DONGRU LI, JONATHAN KOMPERDA, FARZAD MASHAYEK, University of Illinois at Chicago — Numerical simulation of supersonic flows is technologically important in efficient design and development of high-speed propulsion systems. The supersonic flow within the combustor chamber of scramjet is a prime example of multi-scale and multi-physics flow and is generally accompanied by concurrent presence of shock waves and turbulence. Developing a robust numerical method for such simulations leads to various technical challenges due to the presence of complex geometries, shocks, and turbulence, and normally requires massively parallel computation. In the present work, we employ the Discontinuous Spectral Element Method (DSEM) for high-fidelity simulation of supersonic and turbulent flows. The numerical code features an entropy-based artificial viscosity method for capturing shock waves and standard Smagorinsky-Lilly model for turbulence modeling. Two different turbulence sensors are also developed to improve the turbulent viscosity at the shocked areas and the inlet boundary layer. A supersonic cold flow within a ramp-cavity flame holder featuring a round fuel injector at the ramped side of the cavity is simulated. Results are provided and the physics of the flow is studied.

2:47PM R3.00010 Reynolds-constrained large-eddy simulation of compressible flow over a compression ramp — ZUOLI XIAO, LIANG CHEN, Peking University — A novel large-eddy simulation (LES) method is introduced for numerical simulation of wall-bounded compressible turbulent flows. The subgrid-scale (SGS) model in this method is designed to be composed of two parts depending on the distance to the nearest wall. In the near-wall region, both the mean SGS stress and heat flux are constrained by external Reynolds stress and heat flux to ensure the total target quantities, while the fluctuating SGS stress and heat flux are closed in a traditional fashion but using residual model parameterizations. In the far-wall region, the conventional SGS model is directly employed with necessary smoothing operation in the neighborhood of the constrained-unconstrained interface, which might be different for the stress and heat flux depending on the flow configuration. Compressible flow over a compression ramp is numerically studied using the new LES technique. The results are compared with the available experimental and direct numerical simulation (DNS) data, and those from traditional LES and detached-eddy simulation (DES). It turns out that the Reynolds-constrained large-eddy simulation (RCLES) method can predict the size of the separation bubble, mean flow profile, and friction force, etc. more accurately than traditional LES and DES techniques. Moreover, the RCLES method proves to be much less sensitive to the grid resolution than traditional LES method, and makes pure LES of flows of engineering interest feasible with moderate grids.

3:00PM R3.00011 Energy based hybrid turbulence modeling — SIGFRIED HAERING, ROBERT MOSER, University of Texas at Austin — Traditional hybrid approaches exhibit deficiencies when used for fluctuating smooth-wall separation and reattachment necessitating ad-hoc delaying functions and model tuning making them no longer useful as a predictive tool. Additionally, complex geometries and flows often require high cell aspect-ratios and large grid gradients as a compromise between resolution and cost. Such transitions and inconsistencies in resolution detrimentally effect the fidelity of the simulation. We present the continued development of a new hybrid RANS/LES modeling approach specifically developed to address these challenges. In general, modeled turbulence is returned to resolved scales by reduced or negative model viscosity until a balance between theoretical and actual modeled turbulent kinetic energy is attained provided the available resolution. Anisotropy in the grid and resolved field are directly integrated into this balance. A viscosity-based correction is proposed to account for resolution inhomogeneities. Both the hybrid framework and resolution gradient corrections are energy conserving through an exchange of resolved and modeled turbulence.

3:13PM R3.00012 ABSTRACT WITHDRAWN —

Tuesday, November 24, 2015 12:50PM - 3:13PM — Session R4 CFD: Applications II 103 - Chris Pain, Imperial College London

12:50PM R4.00001 Three-dimensional numerical simulations of falling films using an adaptive unstructured mesh1 — CHRIS PAIN, ZHIHUA XIE, OMAR MATAR, Imperial College London — Falling liquid films have rich wave dynamics, often occurring in many industrial applications, such as condensers, evaporators and chemical reactors. A number of numerical studies featuring falling liquid films are available in the literature; the majority of them, however, have focused on two-dimensional falling films. Fewer studies have considered three-dimensional falling films, and those that have only studied the flow in a periodic domain. The objective of this study is to investigate flow dynamics of developing three-dimensional falling films using the Navier-Stokes equations coupled with interface capturing approach over extended domains. An adaptive, unstructured mesh modelling framework is employed here to study this problem, which can modify and adapt three-dimensional meshes to better represent the underlying physics of multiphase problems and reduce computational effort without sacrificing accuracy. Numerical examples of three-dimensional falling films in a long domain are presented and discussed.

1EPSRC Programme Grant, MEMPHIS, EP/K0039761/1
1:03PM R4.00002 Simulation of bubble growth and coalescence in reacting polymer foams

This work concerns with the simulation of reacting polymer foams with Computational Fluid Dynamics (CFD). In these systems, mixing of different ingredients polymerization starts and some gaseous compounds are produced, resulting in the formation of bubbles that grow and coalesce. As the foam expands, the polymerization proceeds resulting in an increase of the apparent viscosity. The evolution of the collective behavior of the bubbles within the polymer foam is tracked by solving a master kinetic equation, formulated in terms of the bubble size distribution. The rate which individual bubbles grow is instead calculated by resolving the momentum and concentration boundary layers around the bubbles. Moreover, since it is useful to track the evolution of the interface between the foam and the surrounding air, a Volume-of-Fluid (VOF) model is adopted. The final computational model is implemented in the open-source CFD code openFOAM by making use of the compressibleInterFoam solver. The master kinetic equation is solved with a quadrature-based moment method (QBMM) directly implemented in openFOAM, whereas the bubble growth model is solved independently and “called” from the CFD code by using an unstructured database. Model predictions are validated against experimental data.

1:16PM R4.00003 A compressible real gas eulerian model for LES of fuel sprays

EDWARD KNUDSEN, ERIC DORAN, Bosch Research And Technology Center — A compressible solver for eulerian multiphase spray simulations is presented. This large eddy simulation solver employs a Peng-Robinson (PR) equation of state to describe mixtures of two species such as liquid docodeane and gaseous nitrogen. Modeling challenges associated with the use of PR are discussed, as are the resource requirements associated with using a compressible formulation to describe liquids when full fuel injector applications are considered. The solver is analyzed using canonical cases and the Spray A experiment from the Engine Combustion Network.

1:29PM R4.00004 ABSTRACT WITHDRAWN

1:42PM R4.00005 Computational Fluid Dynamics Analysis of Canadian Supercritical Water Reactor (SCWR)

MOHAMMAD MOVASSAT, JOANNE BAILEY, METIN YETISIR, Canadian Nuclear Laboratories — A Computational Fluid Dynamics (CFD) simulation was performed on the proposed design for the Canadian SuperCritical Water Reactor (SCWR). The proposed Canadian SCWR is a 1200 MW(e) supercritical light-water cooled reactor with pressurized fuel channels. The reactor concept uses an inlet plenum that all fuel channels are attached to and an outlet header nested inside the inlet plenum. The coolant enters the inlet plenum at 350 C and exits the outlet header at 625 C. The operating pressure is approximately 26 MPa. The high pressure and high temperature outlet conditions result in a higher electric conversion efficiency as compared to existing light water reactors. In this work, CFD simulations were performed to model fluid flow and heat transfer in the inlet plenum, outlet header, and various parts of the fuel assembly. The ANSYS Fluent solver was used for simulations. Results showed that mass flow rate distribution in fuel channels varies radially and the inner channels achieve higher outlet temperatures. At the outlet header, zones with rotational flow were formed as the fluid from 336 fuel channels merged. Results also suggested that insulation of the outlet header should be considered to reduce the thermal stresses caused by the large temperature gradients.

1:55PM R4.00006 Numerics of surface acoustic wave (SAW) driven acoustic streaming and radiation force

NITESH NAMA, Pennsylvania State Univ, RUNE BARNKOB, CHRISTIAN KAHLER, Bundeswehr University Munich, FRANCESCO COSTANZO, TONY JUN HUANG, Pennsylvania State Univ — Recently, surface acoustic wave (SAW) based systems have shown great potential for various lab-on-a-chip applications. However, the physical understanding of the precise acoustic fields and associated acoustophoresis is rather limited. In this work, we present a numerical study of the acoustophoretic particle motion inside a SAW-actuated, liquid-filled polydimethylsiloxane (PDMS) microchannel. We utilize a perturbation approach to divide the flow variables into first- and second-order components. The first-order fields result in a time-averaged acoustic radiation force on suspended particles, as well as the time-averaged body force terms that drive the second-order fields. We model the SAW actuation by a displacement function while we utilize impedance boundary conditions to model the PDMS walls. We identify the precise acoustic fields generated inside the microchannel and investigate a range of particle sizes to characterize the transition from streaming-dominated acoustophoresis to radiation-force-dominated acoustophoresis. Lastly, we demonstrate the ability of SAW devices to tune the position of vertical pressure node inside the microchannel by tuning the phase difference between the two incoming surface acoustic waves.

2:08PM R4.00007 Streaming Potential and Energy Conversion in Nanochannel Grafted With Poly-Zwitterion Brushes

JAHIN PATWARY, University of Maryland, Baltimore County, GUANG CHEN, SIDDHARTHA DAS, University of Maryland — Here we study the streaming potential and electrochemomechanical energy conversion in nanochannels grafted with poly-zwitterion (PZ) brushes. PZs are polymer molecules consisting of negative and positive charge centres simultaneously; depending on the bulk pH, the extent of dissociation differs at each of these charge centres, yielding a particular net charge on the PZ molecule. This PZ charge, therefore, develops a pH dependent electrostatics of the PZ brushes grafted at the nanochannel walls. We develop a self-consistent field theory model to calculate these electrostatics by appropriately accounting for the explicit hydrogen ion concentration. Secondly, we use these electrostatics to calculate the streaming potential and the resulting electrochemomechanical energy conversion in nanochannels grafted with poly-zwitterion (PZ) brushes. Our results indicate distinct influences of pH, bulk ion concentration, and the ionization parameters of the PZs in regulating the nanochannel energy conversion.

2:21PM R4.00008 Polarizable water model for Dissipative Particle Dynamics

IGOR PIVKIN, EMANUEL PETER, Institute of Computational Science, Faculty of Informatics, University of Lugano — Dissipative Particle Dynamics (DPD) is an efficient particle-based method for modeling mesoscopic behavior of fluid systems. DPD forces conserve the momentum resulting in a correct description of hydrodynamic interactions. Polarizability has been introduced into some coarse-grained particle-based simulation methods; however it has not been done with DPD before. We developed a new polarizable coarse-grained water model for DPD, which employs long-range electrostatics and Drude oscillators. In this talk, we will present the model and its applications in simulations of membrane systems, where polarization effects play an essential role.
2:34PM R4.00009 Polarizable protein model for Dissipative Particle Dynamics. EMANUEL PETER, KIRILL LYKOV, IGOR PIVKIN, Institute of Computational Science, Faculty of Informatics, University of Lugano — In this talk, we present a novel polarizable protein model for the Dissipative Particle Dynamics (DPD) simulation technique, a coarse-grained particle-based method widely used in modeling of fluid systems at the mesoscale. We employ long-range electrostatics and Drude oscillators in combination with a newly developed polarizable water model. The protein in our model is represented by a polarizable backbone and a simplified representation of the sidechains. We perform geometry optimizations of the model structures of TrpZip2, TrpCage, and TrpCage. We validate the model on folding of five other proteins and demonstrate that it successfully predicts folding of these proteins into their native conformations. As a perspective of this model, we will give a short outlook on simulations of protein aggregation in the bulk and near a model membrane, a relevant process in several Amyloid diseases, e.g. Alzheimers and Diabetes II.

This work was sponsored by the Collaboratory on Mathematics for Mesoscopic Modeling of Materials (CM4) supported by DOE.

2:47PM R4.00010 Modeling of mesoscopic electrokinetic phenomena using charged dissipative particle dynamics. MINGGE DENG, ZHEN LI, GEORGE KARNIADAKIS, Brown University — In this work, we propose a charged dissipative particle dynamics (cDPD) model for investigation of mesoscopic electrokinetic phenomena. In particular, this particle-based method was designed to simulate micro- or nano-flows which governing by Poisson-Nernst-Planck (PNP) equation coupled with Navier-Stokes (NS) equation. For cDPD simulations of wall-bounded fluid systems, a methodology for imposing correct Dirichlet and Neumann boundary conditions for both PNP and NS equations is developed. To validate the present cDPD model and the corresponding methodology, we perform cDPD simulations of electrostatic double layer (EDL) in the vicinity of a charged wall, and the results show good agreement with the mean-field theoretical solutions. The capacity density of a parallel plate capacitor in salt solution is also investigated with different salt concentration. Moreover, we utilize the proposed methodology to study the electroosmotic and electroosmotic/pressure-driven flow in a micro-channel. In the last, we simulate the dilute polyelectrolyte solution both in bulk and micro-channel, which show the flexibility and capability of this method in studying complex fluids.

1 This work was sponsored by the Collaboratory on Mathematics for Mesoscopic Modeling of Materials (CM4) supported by DOE.

3:00PM R4.00011 Modeling of advection-diffusion-reaction processes using transport dissipative particle dynamics. ZHEN LI, ALIREZA YAZDANI, Division of Applied Mathematics, Brown University, ALEXANDRE TARTAKOVSKY, Computational Mathematics Group, Pacific Northwest National Laboratory, GEORGE EM KARNIADAKIS, Division of Applied Mathematics, Brown University — We present a transport dissipative particle dynamics (tDPD) model for simulating mesoscopic problems involving advection-diffusion-reaction (ADR) processes, along with a methodology for implementation of the correct Dirichlet and Neumann boundary conditions in tDPD simulations. In particular, the transport of concentration is modeled by a Fickian flux and a random flux between tDPD particles, and the advection is implicitly considered by the movements of Lagrangian particles. To validate the proposed tDPD model and the boundary conditions, three simulations of one-dimensional diffusion with different boundary conditions are performed, and the results show excellent agreement with the theoretical solutions. Also, two-dimensional simulations of ADR systems are performed and the tDPD simulations agree well with the results obtained by the spectral element method. Finally, an application of tDPD to the spatio-temporal dynamics of blood coagulation involving twenty-five reacting species is performed to demonstrate the promising biological applications of the tDPD model.

1 Supported by the DOE Center on Mathematics for Mesoscopic Modeling of Materials (CM4) and an INCITE grant.

Tuesday, November 24, 2015 12:50PM - 3:26PM – Session R5 CFD: Uncertainty Quantification

12:50PM R5.00001 A new paradigm for variable-fidelity stochastic simulation and information fusion in fluid mechanics. DANIELE VENTURI, Univ of California-Santa Cruz, LUCIA PARUSSINI, University of Trieste, PARIS PERDIKARIS, Massachusetts Institute of Technology, GEORGE KARNIADAKIS, Brown University — Predicting the statistical properties of fluid systems based on stochastic simulations and experimental data is a problem of major interest across many disciplines. Even with recent theoretical and computational advancements, no broadly applicable techniques exist that could deal effectively with uncertainty propagation and model inadequacy in high-dimensions. To address these problems, we propose a new paradigm for variable-fidelity stochastic modeling, simulation and information fusion in fluid mechanics. The key idea relies in employing recursive Bayesian networks and multi-fidelity information sources (e.g., stochastic simulations at different resolution) to construct optimal predictors for quantities of interest, e.g., the random boundary conditions, the organism and vessels not included in the computational domain. The current practice is to use outflow conditions based on resistance and capacitance, whose values are tuned to obtain a physiological behavior of the patient pressure. However it is not known a priori how this choice affects the results of the simulation. The impact of the uncertainties in these outflow parameters is investigated here by using the generalized Polynomial Chaos approach. This analysis also permits to calibrate the outflow-boundary parameters when patient-specific in-vivo data are available.

1 This research was supported by AFOSR and DARPA.

1:03PM R5.00002 Uncertainty Quantification applied to flow simulations in thoracic aortic aneurysms. ALESSANDRO BOCCADIFUOCO, Scuola Superiore Sant’Anna - Pisa, ALESSANDRO MARIOTTI, DICI - University of Pisa, SIMONA CELI, NICOLA MARTINI, Ospedale del Cuore, Fondazione Toscana G. Monasterio, Massa, MARIA VITTORIA SALVETTI, DICI - University of Pisa — The thoracic aortic aneurysm is a progressive dilatation of the thoracic aorta causing a weakness in the aortic wall, which may eventually cause life-threatening events. Clinical decisions on treatment strategies are currently based on empiric criteria, like the aortic diameter value or its growth rate. Numerical simulations can give the quantification of important indexes which are impossible to be obtained through in-vivo measurements and can provide supplementary information. Hemodynamic simulations are carried out by using the open-source tool SimVascular and considering patient-specific geometries. One of the main issues in these simulations is the choice of suitable boundary conditions, modeling the organs and vessels not included in the computational domain. The current practice is to use outflow conditions based on resistance and capacitance, whose values are tuned to obtain a physiological behavior of the patient pressure. However it is not known a priori how this choice affects the results of the simulation. The impact of the uncertainties in these outflow parameters is investigated here by using the generalized Polynomial Chaos approach. This analysis also permits to calibrate the outflow-boundary parameters when patient-specific in-vivo data are available.
1:16PM R5.00003 Uncertainty quantification of box model and CFD predictions for night-time ventilation in Stanford’s Y2E2 building. CATHERINE GORLÉ, Columbia University, Civil Engineering and Engineering Mechanics, GIANLUCA IACCARINO, Stanford University, Mechanical Engineering — Robust design of natural ventilation systems remains a challenging task, because the simplifications and assumptions introduced in models that predict natural ventilation performance can result in non-negligible uncertainty in the results. The objective of this work is to investigate the predictive capability of a box model and a CFD simulation. We consider night-flush ventilation in the Y2E2 building and indoor air quality using results with available data from air thermometers, representing heat sources and sinks as integral values. The uncertainty in the input parameters is propagated using a non-intrusive polynomial chaos method. The mean result predicts a too fast cooling rate with a maximum air temperature difference of 0.6K, but the measurements are within the predicted 95% confidence interval. The CFD simulation represents a much higher level of detail in the building model, but it also predicts a too high cooling rate with a maximum air temperature difference of 0.9K. Further work will focus on quantifying the uncertainty in the CFD simulation and on using CFD results to determine inputs for the box model, such as discharge and heat transfer coefficients.

1:29PM R5.00004 Quantifying Model-Form Uncertainties in Reynolds Averaged Navier-Stokes Equations: An Open-Box, Physics-Based, Bayesian Approach. HENG XIAO, JINLONG WU, JIANXUN WANG, RUI SUN, CHRISTOPHER J. ROY, Virginia Tech — For many practical flows, the turbulence models are the most important source of uncertainty in Reynolds-Averaged Navier-Stokes (RANS) predictions. In this work, we develop an open-box, physics-informed Bayesian framework for quantifying the model-form uncertainties in RANS simulations. Uncertainties are introduced directly to the Reynolds stresses and are represented with compact parameterization accounting for empirical prior knowledge and physical constraints (e.g., realizability, smoothness, and symmetry). An iterative ensemble Kalman method is used to incorporate the prior information with available observation data in a Bayesian framework to posterior distributions of the Reynolds stresses and other quantities of interest. Two representative cases, the flow over periodic hills and the flow in a square duct, are used to evaluate the performance of the proposed framework. Simulation results suggest that the obtained posterior mean has significantly better agreement with the benchmark data compared to the baseline simulation, even with very sparse observations. At most locations, the posterior distribution adequately represents the model-form uncertainties.

1:42PM R5.00005 Model-Form Uncertainty Quantification in RANS Simulation of Wing-Body Junction Flow. JINLONG WU, JIANXUN WANG, HENG XIAO, Virginia Tech — Junction flow, known as one of the remaining challenges for computational aerodynamics, occurs when a boundary layer encounters an obstacle mounted on the surface. Previous studies have shown that the RANS models are not capable to provide satisfactory prediction. In this work, a novel open-box, physics-informed Bayesian framework is used to quantify the model-form uncertainties in RANS simulation of junction flow. The first objective is to correct the bias in RANS prediction, by utilizing several observation data. The second one is to quantify the model-form uncertainties, which can enable risk-informed decision-making. To begin with a standard RANS simulation, which is performed on a 3.2 elliptic nose and NACA0020 tail cylinder, uncertainties with empirical prior knowledge and physical constraints are directly injected into the Reynolds stresses term, and the unbiased knowledge from observation data is incorporated by an iterative ensemble Kalman method. Current results show that the bias in the quantities of interest (QoIs) of the RANS prediction, e.g., mean velocity, turbulent kinetic energy, etc, can be significantly corrected by this novel Bayesian framework. The probability density distributions of QoIs show that the model-form uncertainty can be quantified as well.

1:55PM R5.00006 Assessment of the DNS Data Accuracy Using RANS-DNS Simulations. JUAN D. COLMENARES F., SVETLANA V. POROSEVA, University of New Mexico, SCOTT M. MURMAN, Nasa Ames — Direct numerical simulations (DNS) provide the most accurate computational description of a turbulent flow field and its statistical characteristics. Therefore, results of simulations with Reynolds-Averaged Navier-Stokes (RANS) turbulence models are often evaluated against DNS data. The goal of our study is to determine a limit of RANS model performance in relation to existing DNS data. Since no model can outperform DNS, this limit can be determined by solving RANS equations with all unknown terms being represented by their DNS data (RANS-DNS simulations). In the presentation, results of RANS-DNS simulations conducted using transport equations for velocity moments of second, third, and fourth orders in incompressible planar wall-bounded flows are discussed. The results were obtained with two solvers: OpenFOAM and an in-house code for fully-developed flows at different Reynolds numbers using different DNS databases.

2:08PM R5.00007 A fast algorithm for the estimation of statistical error in DNS (or experimental) time averages. PAOLO LUCHINI, University of Salerno - DIIN, SERENA RUSSO, University of Naples Federico II — A standard final step in the DNS (but the same can be said of experimental measurements) of turbulence, is the time- and space-averaging of the instantaneous results in order to give their means or correlations or other statistical properties. These averages are necessarily performed over a finite time and space window, and are therefore more correctly just estimates of the “true” statistical averages. The choice of the appropriate window size is most often subjectively based on individual experience, but as sublter statistics enter the focus of investigation, an objective criterion becomes desirable. Classical estimators of the averaging error of finite time series fall in two categories: “batch means” algorithms, fast but not very accurate, and ARMA methods, slower because they estimate the complete correlation function to start with. Here a modification of the batch means algorithm will be presented, which retains its speed while removing its biasing error. As a side benefit, an automatic determination of batch size is also included. Examples will be given involving both an artificial time series of known statistics and an actual DNS of turbulence.

2:21PM R5.00008 Turbulence model form uncertainty quantification in OpenFOAM. ZENGXING HAO, Columbia Univ, STEPHANIE ZEOLI, LAURENT BRIECUTEX, Univ Mons, CATHERINE GORLÉ, Columbia Univ, CFD & UQ TEAM, FLUIDS-MACHINES TEAM — Reynolds-averaged Navier-Stokes (RANS) simulations with a two-equation linear eddy-viscosity turbulence model remain a commonly used computational technique for engineering design and analysis of turbulent flows. The accuracy of the results is however limited by the inability of the turbulence model to correctly predict the complex flow features relevant to engineering applications. To enable supporting critical design decisions based on these imperfect model results it is essential to quantify the uncertainty related to the turbulence model form and define confidence levels for the results. The objective of this study is the implementation and validation of a previously developed approach for quantifying the uncertainty in RANS predictions of a turbulent flow in the open source code OpenFOAM. The methodology is based on two steps: 1. calculate a marker to determine where in the flow the model is plausibly inaccurate, and 2. perturb the modeled Reynolds stresses in the momentum equations. The perturbations are defined in terms of the decomposed Reynolds stress tensor, i.e., the tensor magnitude and the eigenvalues and eigenvectors of the normalized anisotropy tensor. Results for a square duct and the flow over a wavey wall will be presented for validation of the implementation.
2:34PM R5.00009 Representing Model Inadequacy in Combustion Mechanisms of Laminar Flames, REBECCA MORRISON, ROBERT MOSER, TODD OLIVER, Univ of Texas, Austin — An accurate description of the chemical processes involved in the oxidation of hydrocarbons may include hundreds of reactions and thirty or more chemical species. Kinetics models of these chemical mechanisms are often embedded in a fluid dynamics solver to represent combustion. Because the computational cost of such detailed mechanisms is so high, it is common practice to use drastically reduced mechanisms. But, this introduces modeling errors which may render the model inadequate. In this talk, we present a formulation of the model inadequacy in reduced models of combustion mechanisms. Our goal is to account for the discrepancy between the detailed model and its reduced version by incorporating an additive, linear, probabilistic inadequacy model. In effect, it is a random matrix, whose entries are characterized by probability distributions and which displays interesting properties due to conservation constraints. In particular, we investigate how the inclusion of the random matrix affects the prediction of flame inadequacy model. In effect, it is a random matrix, whose entries are characterized by probability distributions and which displays interesting properties due to conservation constraints. In particular, we investigate how the inclusion of the random matrix affects the prediction of flame inadequacy model.

2:47PM R5.00010 Uncertainty Quantification of the Dynamic Mode Decomposition, ANTHONY DEGENNARO, SCOTT DAWSON, CLARENCE ROWLEY, Princeton University — This work explores and quantifies the statistical effect that parameterized uncertainty has on the dynamic mode decomposition (DMD). For the data under consideration, such uncertain parameters could include Reynolds number, geometry, or random sensor/signal noise in the system. The aims of this study are twofold: firstly, to quantify the robustness of the algorithm in terms of pertinent identified quantities (such as DMD modes and eigenvalues), thus expanding upon recent work in this area, and secondly, to present a method for analyzing the underlying dynamic systems from data in an efficient manner. We use polynomial chaos expansions to represent the relevant DMD quantities of interest. This approach can be computationally more efficient than sample-based methods (e.g., Monte Carlo) when the dimensionality of the parameter space is moderate. We demonstrate our methodology on a number of well-studied example systems, including numerical simulations of flow past a circular cylinder.

3:00PM R5.00011 Impact of uncertainties in free stream conditions on the aerodynamics of a rectangular cylinder, ALESSANDRO MARIOTTI, DICI - University of Pisa, PEJMAN SHOEBI OMRAHI, Fluid Dynamics Division, Netherlands Organization for Applied Scientific Research (TNO), JEROEN WITTEVEEN, Scientific Computing Group, Center for Mathematics and Computer Science (CWI), MARIA VITTORIA SALVETTI, DICI - University of Pisa — The BARC benchmark deals with the flow around a rectangular cylinder with a chord-to-depth ratio equal to 4. This flow configuration is of practical interest for civil and industrial structures and it is characterized by massively separated flow and unsteadiness. In a recent review of BARC results, significant dispersion was observed both in experimental and numerical predictions of some flow quantities, which are extremely sensitive to various uncertainties, which may be present in experiments and simulations. Besides modeling and numerical errors, in simulations it is difficult to exactly reproduce the experimental conditions due to uncertainties in the set-up parameters, which sometimes cannot be exactly controlled or characterized. Probabilistic methods and URANS simulations are used to investigate the impact of the uncertainties in the following set-up parameters: the angle of incidence, the free stream longitudinal turbulence intensity and length scale. Stochastic collocation is employed to perform the probabilistic propagation of the uncertainty. The discretization and modeling errors are estimated by repeating the same analysis for different grids and turbulence models. The results obtained for different assumed PDF of the set-up parameters are also compared. Tuesday, November 24, 2015 12:50PM - 2:47PM Session R6 CFD: General 105 - Sandra Sowah, Princeton University

12:50PM R6.00001 Numerical Simulations of Curvature Effects in Laminar Channel Flows, SANDRA S. SOWAH, MICHAEL E. MUELLER, HOWARD A. STONE, Princeton University — Numerical simulations of curvature effects in laminar channel flows are performed by introducing body force terms into the Navier-Stokes equations, which are written in Cartesian coordinates. The advantage of introducing body force terms within a Cartesian framework, compared to performing simulations in native cylindrical coordinates, is the ability to easily transition from straight to curved regions of a channel flow. Using this approach, the onset of Dean vortices for laminar flow is investigated for varying Reynolds numbers and ratios of radius of curvature to channel height. The results are verified against simulations in cylindrical coordinates.

1:03PM R6.00002 An Eulerian-based Bubble Dynamics Model for Computational Fluid Dynamics, ASISH BALU, MICHAEL KINZEL, Pennsylvania State Univ — Cavitation dynamics of nuclei are largely governed by the Rayleigh-Plesset Equation (RPE). This research explores the implementation of a one-way coupling to the solution of the RPE to a computational fluid dynamics (CFD) simulation in an Eulerian-framework. In this work, we used transport equations (i.e., advection) of the bubble radius and bubble growth rate, both of which are governed by advection mechanisms and coupling to the RPE through the CFD pressure field. The method is validated in the context of hypothetical pressure fields by prescribing a temporally varying pressure. Then, it is extended to one-way coupling with cavitation development in three different flow situations: (1) flow over a cylinder, (2) bubble formation during a bottle collapse event, and (3) cavitation in a tip vortex. In the context of these flows, the CFD simulations replicate an equivalent MATLAB-based solution to the RPE, thus validating the model. Additionally, an analytical formulation for appropriate upper and lower bounds for the bubble’s physical properties is prescribed. These bounds serve as initial and ending points for the CFD solver to converge at longer time steps, therefore increasing the rate of convergence as well as maintaining solution accuracy. The results from this work suggest that Eulerian-based RPE cavitation models are practical and have the potential to simulate large numbers of bubbles that challenge Lagrangian methods.

1:16PM R6.00003 ABSTRACT WITHDRAWN —
1:29PM R6.00004 Aiding Design of Wave Energy Converters via Computational Simulations, HEJAR JEBELI AQDAM, PhD Student, University of Massachusetts Dartmouth, BABAK AHMADI, MS Student, University of Massachusetts Dartmouth — With the increasing interest in renewable energy sources, wave energy converters will continue to gain attention as a viable alternative to current electricity production methods. It is therefore crucial to develop computational tools for the design and analysis of wave energy converters. A successful design requires balance between the design performance and cost. Here an analytical solution is used for the approximate analysis of interactions between a flap-type wave energy converter (WEC) and waves. The method is verified using other flow solvers and experimental test cases. Then the model is used in conjunction with a powerful heuristic optimization engine, Charged System Search (CSS) to explore the WEC design space. CSS is inspired by charged particles behavior. It searches the design space by considering candidate answers as charged particles and moving them based on the Coulomb’s laws of electrostatics and Newton’s laws of motion to find the global optimum. Finally the impacts of changes in different design parameters on the power takeout of the superior WEC designs are investigated.

1:42PM R6.00005 An Assessment of Supercavitation Transition using Computational Fluid Dynamics, MELISSA FRONZEO, MICHAEL KINZEL, Penn State ARL — A computational fluid dynamics approach is used to improve the understanding of supercavitation and its physical characteristics. A ventilated disk cavitator is used in several studies to evaluate these physics. The first study focuses on twin vortex cavities, specifically to understand correlation between cavity shape and pressure. The study uses validated measurements (in the CFD model) of the cavity shape and pressure for various ventilation rates and Fr numbers. The data is used to evaluate the semi-empirical formula of L.A Epstein, where results indicate a potentially improved correlation. In addition, the detailed measurements of the CFD model yield insight on improved experimental measurement techniques for cavity pressure. The second study uses unsteady detached eddy simulations (DES) to predict hysteresis in the transition behavior of the cavity closure from toroidal vortex to twin-vortex regimes. The solution is initialized as a toroidal-type cavity (low gas ventilation rate), then the ventilation rate is slowly increased until a twin-vortex cavity is formed. In addition, the opposite process is also performed. The data is analyzed to develop an understanding of the unknown physical mechanisms involved in the transition process.

1:55PM R6.00006 The Immersed Interface Method for Flow Around Non-Smooth Boundaries, YANG LIU, SHENG XU, Southern Methodist University — In the immersed interface method, a boundary immersed in a fluid is generated by a singular force in the Navier-Stokes equations, and the singular force enters a numerical scheme as jump conditions across the boundary. In previous work, the method has been developed for smooth boundaries. In this talk, we present how to extend the method for non-smooth boundaries. We use panels to represent a boundary, compute necessary jump conditions explicitly, and compare two different pressure Poisson solvers. We test our extended method by simulating flows past a circular cylinder, a square cylinder or around a flapping plate. Our results show that the method is robust, accurate and efficient.

2:08PM R6.00007 Computational Framework for a Fully-Coupled, Collocated-Arrangement Flow Solver Applicable at all Speeds, CHEM-NIAN XIAO, FABIAN DENNER, BEREND VAN WACHEM, Imperial College London — A pressure-based Navier-Stokes solver which is applicable to fluid flow problems of a wide range of speeds is presented. The novel solver is based on collocated variable arrangement and uses a modified Rhie-Chow interpolation method to assure implicit pressure-velocity coupling. A Mach number biased modification to the continuity equation as well as coupling of flow and thermodynamic variables via an energy equation and vorticity of state enable the simulation of compressible flows belonging to transonic or supersonic Mach number regimes. The flow equations systems are all solved simultaneously, thus guaranteeing strong coupling between pressure and velocity at each iteration step. Shock-capturing is accomplished via nonlinear spatial discretisation schemes which adaptively apply an appropriate blending of first-order upwind and second-order central schemes depending on the local smoothness of the flow field. A selection of standard test problems will be presented to demonstrate the solvers capability of handling incompressible as well as compressible flow fields of vastly different speed regimes on structured as well as unstructured meshes.

The authors are grateful for the financial support of Shell

2:21PM R6.00008 Lagrangian Proper Orthogonal Decomposition of the Wake Downstream of a Cylinder, JACK ROSSETTI, Ph.D Student, Mechanical and Aerospace Engineering, Syracuse University, MELISSA GREEN, JOHN DANNENHOFER, Associate Professor, Mechanical and Aerospace Engineering, Syracuse University — Proper orthogonal decomposition (POD) has long been utilized by the fluid dynamics community to extract information regarding the energy contained in the structures of turbulent flows. These POD techniques are generally executed in an Eulerian frame, encapsulating all the structures created and destroyed through time. Unfortunately, the mode shapes that Eulerian POD produce are linked to the translation of structures and little is learned about the evolution of individual structures. We overcome this by applying POD in a Lagrangian frame. We first track pertinent features through cross-correlation techniques. Both Eulerian and Lagrangian POD were tested on a CFD simulation of the wake downstream of a cylinder. Eulerian POD focuses on the large-scale von Karman vortex street, whereas the Lagrangian POD allows one to extract physical phenomena associated with each of the individual vortices. This can result in a better understanding of the physics within each vortex.

2:34PM R6.00009 ABSTRACT WITHDRAWN —

Tuesday, November 24, 2015 12:50PM - 3:13PM –

Session R7 CFD: Algorithms 107 - Joseph Powers, University of Notre Dame

12:50PM R7.00001 Physical diffusion suppresses the carbuncle instability, KE SHI, ALEKSANDER JEMCOV, JOSEPH POWERS, Univ of Notre Dame — We demonstrate a simple antidote exists to the numerical carbuncle instability predicted by some shock-capturing schemes: inclusion of physical momentum and energy diffusion via a compressible Navier-Stokes solution to the supersonic flow of a calorically perfect ideal gas past a circular cylinder. We demonstrate the carbuncle phenomenon and its rectification by solving two problems. Both employ the same geometry, initial conditions, computational grid, time step size, advective flux model of a Roe-based scheme without an entropy fix, and time-advancement scheme. For the first problem, we neglect physical diffusion, while for the second we include it. When physical diffusion is neglected, we predict a carbuncle phenomenon; however, when it is included and sufficiently resolved, no carbuncle is predicted, in agreement with experiment.
leveraging new time-stepping and multigrid algorithms in incompressible flow computations on overset grids in a time-stable manner will be briefly discussed. The stability and the accuracy of the methods for solving the Euler equations. The extension of these methods to solve the Navier-Stokes equations assessed against the commonly-used approach of injecting the interpolated data onto each grid. Numerical results will be presented to confirm the contributions of the CGP method to the pressure correction technique are twofold: first, it substantially lessens the computational cost devoted to the Poisson equation, which is the most time-consuming part of the simulation process. Second, it preserves the accuracy of the velocity field. The velocity and pressure spaces are approximated by Galerkin spectral element using piecewise linear basis functions. A restriction operator is designed so that fine data are directly injected into the coarse grid. The Laplacian and divergence matrices are driven by taking inner products of coarse grid shape functions. Linear interpolation is implemented to construct a prolongation operator. A study of the data accuracy and the CPU time for the CGP-based versus non-CGP computations is presented.

A coarse-grid-projection acceleration method for finite-element incompressible flow computations, ALI KASHIF, Engineering Science and Mechanics Program, Department of Biomedical Engineering and Mechanics, Virginia Tech, ANNE STAPLES, Associate Professor, Engineering Science and Mechanics Program, Department of Biomedical Engineering and Mechanics, Virginia Tech, FIN LAB TEAM — Coarse grid projection (CGP) methodology provides a framework for accelerating computations by performing some part of the computation on a coarsened grid. We apply the CGP to pressure projection methods for finite element-based incompressible flow simulations. Based on it, the predicted velocity field data is restricted to a coarsened grid, the pressure is determined by solving the Poisson equation on the coarse grid, and the resulting data are prolonged to the preset fine grid. The Laplacian and divergence matrices are driven by taking inner products of coarse grid shape functions. Linear interpolation is implemented to construct a prolongation operator. A study of the data accuracy and the CPU time for the CGP-based versus non-CGP computations is presented.

A high-order provably stable overset grid methods for hyperbolic problems, with application to the Euler equations, NEK SHARAN, Graduate Student, Department of Aerospace Engineering, University of Illinois at Urbana-Champaign, CARLOS PANTANO, Associate Professor, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, DANIEL BODONY, Associate Professor, Department of Aerospace Engineering, University of Illinois at Urbana-Champaign — Overset grids provide an efficient and flexible framework to implement high-order finite difference methods for simulations of compressible viscous flows over complex geometries. However, prior overset methods were not provably stable and were applied with artificial dissipation in the interface regions. We will discuss new, provably time-stable methods for solving hyperbolic problems on overlapping grids. The proposed methods use the summation-by-parts (SBP) derivative approximations coupled with the simultaneous-approximation-term (SAT) methodology for applying boundary conditions and interface treatments. The performance of the methods will be assessed against the commonly-used approach of injecting the interpolated data onto each grid. Numerical results will be presented to confirm the stability and the accuracy of the methods for solving the Euler equations. The extension of these methods to solve the Navier-Stokes equations on overset grids in a time-stable manner will be briefly discussed.

Study of time-accurate integration of the variable-density Navier-Stokes equations, XIAOYI LU, CARLOS PANTANO, Mechanical Science and Engineering Department, University of Illinois at Urbana-Champaign — We present several theoretical elements that affect time-consistent integration of the low-Mach number approximation of variable-density Navier-Stokes equations. The goal is for velocity, pressure, density, and scalars to achieve uniform order of accuracy, consistent with the time integrator being used. We show examples of second-order (using Crank-Nicolson and Adams-Bashforth) and third-order (using additive semi-implicit Runge-Kutta) uniform convergence with the proposed conceptual framework. Furthermore, the consistent approach can be extended to other time integrators. In addition, the method is formulated using approximate/incomplete factorization methods for easy incorporation in existing solvers. One of the observed benefits of the proposed approach is improved stability, even for large density difference, in comparison with other existing formulations. A linearized stability analysis is also carried out for some test problems to better understand the behavior of the approach.
for Incompressible Viscous Flows on Staggered Cartesian Grids, VINCENT LE CHENADEC, YONG

The new method addresses both of these challenges by utilizing an adaptive anisotropic wavelet transform on curvilinear meshes that can be either algebraically prescribed or calculated on the fly using PDE-based mesh generation. In order to ensure accurate representation of spatial operators in physical space, an additional adaptation on spatial physical coordinates is also performed. It is important to note that when new nodes are added in computational space, the physical coordinates can be approximated by interpolation of the existing solution and additional local iterations to ensure that the solution of coordinate mapping PDEs is converged on the new mesh. In contrast to traditional mesh generation methods, the cost of adding additional nodes is minimal, mainly due to localized nature of iterative mesh generation PDE solver requiring local iterations in the vicinity of newly introduced points.

This work was supported by ONR MURI under grant N00014-11-1-069.

A Fully Conservative and Entropy Preserving Cut-Cell Method for Incompressible Viscous Flows on Staggered Cartesian Grids, VINCENT LE CHENADEC, YONG YI BAY, University of Illinois at Urbana-Champaign — The treatment of complex geometries in Computational Fluid Dynamics applications is a challenging endeavor, which immersed boundary and cut-cell techniques can significantly simplify by alleviating the meshing process required by body-fitted meshes. These methods also introduce new challenges, in that the formulation of accurate and well-posed discrete operators is not trivial. A cut-cell method for the solution of the incompressible Navier-Stokes equation is proposed for staggered Cartesian grids. In both scalar and vector cases, the emphasis is set on the structure of the discrete operators, designed to mimic the properties of the continuous ones while retaining a nearest-neighbor stencil. For convective transport, different forms are proposed (divergence, advective and skew-symmetric), and shown to be equivalent when the discrete continuity equation is satisfied. This ensures mass, momentum and kinetic energy conservation. For diffusive transport, conservative and symmetric operators are proposed for both Dirichlet and Neumann boundary conditions. Symmetry ensures the existence of a sink term (viscous dissipation) in the discrete kinetic energy budget, which is beneficial for stability. The accuracy of method is finally assessed in standard test cases.

This work was supported by ONR MURI on Soil Blast Modeling.

A low-dissipation numerical scheme on Voronoi grids for complex geometries, FRANK HAM, SANJEEB BOSE, BABAHEJAZI, Cascade Technologies, VARUN MITTAL, Bosch RTC — The generation of high quality meshes in complex geometries suitable for multi-scale computations remains difficult and cumbersome. Inevitably, the lack of regularity, skewness, or other undesirable mesh features leads to compromises in the numerical scheme where either accuracy is sacrificed or dissipation is introduced. Reliance on scheme switching based on grid quality complicates grid convergence, especially when utilizing local refinement. We introduce an alternative strategy where the computational meshes are built from the Voronoi diagram of a prescribed point cloud. The use of the Voronoi diagram naturally leads to a mesh with inherent quality (e.g., alignment of face normals and site displacement vectors). Moreover, because the Voronoi diagram is defined uniquely from a set of points, mesh regularity can be achieved from either proper packing of the generating sites or by straightforward mesh smoothing. The efficiency (of both the diagram generation and solution), convergence, and solution quality will be illustrated using canonical and applied configurations.

Tuesday, November 24, 2015 12:50PM - 3:13PM
Session R8 Microscale Flows: Particles 108 - Sascha Hilgenfeldt, University of Illinois at Urbana-Champaign
12:50PM R8.00001 Size-selective sorting in bubble streaming flows: Particle migration on fast time scales. RAQEEB THAMEEM, BHARGAV RALLABANDI, SASCHA HILGENFELDT, Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — Steady streaming from ultrasonically driven microbubbles is an increasingly popular technique in microfluidics because such devices are easily manufactured and generate powerful and highly controllable flows. Combining streaming and Poiseuille transport flows allows for passive size-sensitive sorting at particle sizes and selectivities much smaller than the bubble radius. The crucial particle deflection and separation takes place over very small times (milliseconds) and length scales (20-30 microns) and can be rationalized using a simplified geometric mechanism. A quantitative theoretical description is achieved through the application of recent results on three-dimensional streaming flow field contributions. To develop a more fundamental understanding of the particle dynamics, we use high-speed photography of trajectories in polydisperse particle suspensions, recording the particle motion on the time scale of the bubble oscillation. Our data reveal the dependence of particle displacement on driving phase, particle size, oscillatory flow speed, and streaming speed. Within this mechanism, the effective repulsive force exerted by the bubble on the particle can be quantified, showing for the first time how fast, selective particle migration is effected in a streaming flow.

1:03PM R8.00002 Forces on particles in microstreaming flows. SASCHA HILGENFELDT, BHARGAV RALLABANDI, RAQEEB THAMEEM, Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — In various microfluidic applications, vortical steady streaming from ultrasonically driven microbubbles is used in concert with a pressure-driven channel flow to manipulate objects. While a quantitative theory of this boundary-induced streaming is available, little work has been devoted to a fundamental understanding of the forces exerted on microparticles in boundary streaming flows, even though the differential action of such forces is central to applications like size-sensitive sorting. Contrary to other microfluidic sorting devices, the forces in bubble microstreaming act over millisecond times and micron length scales, without the need for accumulated deflections over long distances. Accordingly, we develop a theory of hydrodynamic forces on the fast time scale of bubble oscillation using the lubrication approximation, showing for the first time how particle displacements are rectified near moving boundaries over multiple oscillations in parallel with the generation of the steady streaming flow. This dependence of particle migration on particle size is compared with experimental data. This theory is applicable to boundary streaming phenomena in general and demonstrates how particles can be sorted very quickly and without compromising device throughput.

1:16PM R8.00003 Drag and diffusion coefficient of a spherical particle attached to a fluid interface. STEFFEN HARDT, AARON DOERR, Center of Smart Interfaces, TU Darmstadt, HASSAN MASOUD, Department of Mechanical Engineering, University of Munich, HOWARD STONE, Department of Mechanical and Aerospace Engineering, Princeton University — We consider a spherical particle attached to the interface between two immiscible fluids of large viscosity contrast. The degree of immersion in the two fluids is determined by the contact angle. For small enough particles and significant contact-angle hysteresis, it can be assumed that the three-phase contact line is pinned at the particle surface. We study the movement of such particles along the fluid interface for the case of small Reynolds and capillary numbers. We solve the Stokes equation based on two geometric perturbation expansions around contact angles of 90 degrees and 180 degrees, the latter corresponding to a particle completely immersed in the less viscous phase. Based on the Lorentz Reciprocity Theorem we obtain expressions for the drag coefficient of an interfacial particle which are analogs of the well-known Stokes drag coefficient for a particle moving in an unbounded medium. Interpolation of the two results gives a relationship which approximates the drag coefficient quite accurately over the entire range of contact angles. A comparison with previously published numerical results for contact angles below 90 degrees shows good agreement. Using the fluctuation-dissipation theorem, we also obtain expressions for the diffusion constant of a small particle attached to a fluid interface.

1:29PM R8.00004 Size-sensitive particle trajectories in three-dimensional microbubble acoustic streaming flows. ANDREAS VOLK, MASSIMILIANO ROSSI, Bundeswehr University Munich, SASCHA HILGENFELDT, BHARGAV RALLABANDI, University of Illinois at Urbana-Champaign, CHRISTIAN KAHLER, ALVARO MARIN, Bundeswehr University Munich — Oscillating microbubbles generate steady streaming flows with interesting features and promising applications for microparticle manipulation. The flow around oscillating semi-cylindrical bubbles has been typically assumed to be independent of the axial coordinate. However, it has been recently revealed that particle motion is strongly three-dimensional [A. Marin et al., Phys. Rev. Appl. 3, 041001, (2015); Rallabandi et al., J. Fluid Mech. 777, (2015)]. Small tracer particles follow vortical trajectories with pronounced axial displacements near the bubble, weaving a toroidal stream surface. A well-known consequence of bubble streaming flows is size-dependent particle migration [C. Wang et al., Biomicrofluidics (2012)], which can be exploited for sorting and trapping of microparticles in microfluidic devices. In this talk, we will show how the three-dimensional toroidal topology found for small tracer particles is modified as the particle size increases up to 1/3 of the bubble radius. Our results show how the size-sensitive particle positioning along the axis of the semi-cylindrical bubble. In order to analyze the three-dimensional sorting and trapping capabilities of the system, experiments with an imposed flow and polydisperse particle solutions are also shown.

1:42PM R8.00005 In-situ Microfluidic Measurement of the Dielectric Constant of Colloidal Particles. SETAREH MANAFIRASI, Department of Chemical Engineering, The City College of New York, THOMAS LEARY, Department of Chemical Engineering and Applied Chemistry, University of Toronto, CHARLES MALDARELLI, Department of Chemical Engineering, The City College of New York — The ability to manipulate micron-sized colloidal particles or biological cells in a liquid medium in microfluidic geometries is necessary in lab on a chip devices for micro scale biological analysis and diagnostics for sorting and directing the trafficking of the particles. In dielectrophoresis, a nonuniform electric (E) field is applied to move the particles along the gradient of the field energy, and the velocity is a function of the particle’s dielectric constant. Measurement of the dielectric constant is necessary in order to scale field strengths for applications, and it is important to undertake this measurement in-situ as the particle’s dielectric constant can be modified by the suspending medium (e.g. adsorption onto the particle surface). In this talk we measure directly the dielectric constant of colloids in a microfluidic channel by applying an electric field with “V”-shaped and planar electrodes on opposite sides of the channel. The cusp of the “V” shape concentrates the field to provide a sufficient field intensity gradient which is designed to be uniform across the height of the channel and to vary only with its width. Optical measurements of the dielectrophoretic velocity of polymer colloids are compared to simulations based on numerical solutions of the E-field and particle hydrodynamics to obtain the particle dielectric constant and investigate the effect of biomolecule adsorption on the particle surface.
1:55PM R8.00006 MicroPIV measurements of flows induced by rotating microparticles near a boundary. JAMEL ALI, MINJUN KIM, Drexel University — We report the hydrodynamics induced by single digit micron sized particles rotating in low Reynolds number environments and analysis of their flow fields using MicroPIV. Magnetic microparticles floating a few nanometers above a glass substrate, in an otherwise quiescent fluid, were actuated wirelessly using a rotating magnetic field controlled using two pairs of orthogonally positioned electromagnetic coils. A highspeed camera was used to sufficiently capture the motion of nanometer sized seeding particles at 500 frames per second as well as track the rotation of the microparticles. Analysis of microPIV data revealed agreement with the analytical solution for flow fields generated by two particles as they approach each other, to form dimers, was also analyzed. It was observed that as two synchronously rotating beads of equal diameter are placed closed together, their flow fields were offset, at their combined center of mass, and superimposed near their outer peripheries. These results suggest that colloidal magnetic particles can be pattered in a manner such that when rotated their generated flow is globally coordinated.

2:08PM R8.00007 Hydrodynamic repulsion of elastic dumbbells. MAREK BUKOWICKI, MARTA GRUCA, Institute of Fundamental Technological Research, Polish Academy of Sciences — Dynamics of two identical elastic dumbbells, settling under gravity in a viscous fluid at low Reynolds number are analyzed within the point-particle model. Initially, the dumbbells are vertical; their centers are aligned horizontally, and the springs which connect the dumbbell’s beads are at the equilibrium. The motion of the beads is determined numerically with the use of the Runge-Kutta method. After an initial relaxation phase, the system converges to a universal time-dependent solution. The elastic dumbbells tumble while falling, but their relative motion is not periodic (as in case of rigid dumbbells or pairs of separated beads). The elastic constraints break the time-reversal symmetry of the motion. As the result, the horizontal distance between the dumbbells slowly increases – they are hydrodynamically repelled from each other. This effect can be very large even though the elastic forces are always much smaller than gravity. [For the details, see M. Bukowicki, M. Gruca, M. L. Ekiel-Jezewska, J. Fluid Mech. 767, p. 95 (2015).] The dynamics described above are equivalent to the motion of a single elastic dumbbell under a constant external force which is parallel to a flat free surface. The dumbbell migrates away from the interface and its tumbling time increases.

2:21PM R8.00008 Collective effects in the flotation of electrically charged particles at an interface. DUCK-GYU LEE, Department of Nature-Inspired Nanoconvergence Systems, Korea Institute of Machinery and Materials, PIETRO CICTUA, Cavendish Laboratory, University of Cambridge, Cambridge CB3 0HE, United Kingdom, DOMINIC VELLA, Mathematical Institute, University of Oxford, Oxford OX2 8G, United Kingdom — We study the flotation of electrically charged line particles at an interface. Motivated by recent work on the localization of charged and magnetic particles at interfaces, we consider the equilibrium of the particles, accounting for the weight of each as well as the electrical and surface tension forces acting on them. Our numerical solution of the force balance equations shows that as the number of particles increases, the particles sink deeper into the liquid and ultimately sink. To understand whether the clumps of particles that are formed are stable, we use a free energy analysis; this shows that as the number of particles N increases, the binding energy per particle increases also. We compare our numerical results with scaling and experimental analyses.

2:34PM R8.00009 High-Throughput, Motility-Based Sorter for Microswimmers and Gene Discovery Platform. JINZHOU YUAN, DAVID RAIZEN, HAIM BAU, University of Pennsylvania — Animal motility varies with genotype, disease progression, aging, and environmental conditions. In many studies, it is desirable to carry out high throughput motility-based sorting to isolate rare animals for, among other things, forward genetic screens to identify genetic pathways that regulate phenotypes of interest. Many commonly used screening processes are labor-intensive, lack sensitivity, and require extensive investigator training. Here, we describe a sensitive, high throughput, automated, motility-based method for sorting nematodes. Our method was implemented in a simplified microfluidic device capable of sorting many hundred animals per module, and ultimately sink. To understand whether the clumps of particles that are formed are stable, we use a free energy analysis; this shows that as the number of particles N increases, the binding energy per particle increases also. We compare our numerical results with scaling and experimental analyses.

This work is supported, in part, by NIH NIA Grant 5R03AG042690-02.

2:47PM R8.00010 Continuous size separation of micro/nano particles using ridged microchannel by controlling particle position in the z-direction. BUSHRA TASADDUQ, GONGHAO WANG, WENBIN MAO, WILBUR LAM, ALEXANDER ALEXEEV, ALI FATIH SARIOGLU, TODD SULCHEK, Georgia Institute of Technology — In the last meeting we presented results that demonstrated that the particle trajectories depend on their z-position inside a microchannel with diagonal ridges. The phenomenon arises due to dipoles created by the diagonal ridges that transport the fluid at the channel center in the negative y-direction, whereas the fluid located near the bottom channel walls moves in the positive y-direction. This effect is harnessed to improve the separation of particles by size. We have incorporated a vertical sheath to improve the z focusing of particles in our device and operated the device at an optimized sample to vertical sheath flow rates. As the vertical sheath flow velocity increases, the sample flow streamlines are pushed towards the bottom channel wall. Due to vortices created by diagonal ridges the small particles are pushed towards the bottom channel and move with positive y-trajectories. Large particles also are pushed towards the bottom of the channel, yet due to their larger sizes and close to the gap size they experience a net negative y-trajectory. We are able to improve the purity of large particle enrichment by over 5 times as compared to our previous work.

3:00PM R8.00011 Control of Lateral Inertial Migration Rate of Particles in Microchannels. ARMIN KARIRI, University of California, Los Angeles, RISHAV ROY, Indian Institute of Technology (IIT), Kharagpur, SAM BRAY, DINO DI CARLO, University of California, Los Angeles — The net inertial lift force acting on particles results in lateral inertial migration across streams. The migration direction and magnitude is strongly dependent on channel geometry, size of the particle, Reynolds number and location of the particle within the channel cross-section. In many chemical and biological applications in which precise temporal control and solution exchange around particles is required, the initial variation in distribution of focusing positions of particles within the channel cross-section becomes a determining factor. This variation is shown to be a limiting factor in achieving precise control over the migration time in previous studies. In order to improve uniformity of the average migration rate, a microfluidic device is designed to aid particles in achieving a single stable equilibrium position by inducing a net helical flow. Using this inertial focusing platform, a comprehensive numerical and experimental study is performed to characterize the range of lateral migration rates for rigid spherical particles as a function of particle size, initial particle position, flow rates of each stream and Reynolds number for a given channel geometry. The tool developed in this study can be used to achieve precise migration characteristics for the microparticles crossing fluid streams in microchannels over millisecond time scales.

1:This work was supported, in part, by NIH NIA Grant 5R03AG042690-02.
12:50PM R9.00001 Simulations of Micropumps Based on Tilted Flexible Fibers

MATT HANCOCK, NAGI ELABBASI, Veryst Engineering, LLC, MELIK DEMIREL, The Pennsylvania State University — Pumping liquids at low Reynolds numbers is challenging because of the principle of reversibility. We report here a class of microfluidic pump designs based on tilted flexible structures that combines the concepts of cilia (flexible elastic elements) and rectifiers (e.g., Tesla valves, check valves). We demonstrate proof-of-concept with 2D and 3D fluid-structure interaction (FSI) simulations in COMSOL Multiphysics® of micropumps consisting of a source for oscillatory fluidic motion, e.g., a piston, and a channel lined with tilted flexible rods or sheets to provide rectification. When flow is against the rod tilt direction, the rods bend backward, narrowing the channel and increasing flow resistance; when flow is in the direction of rod tilt, the rods bend forward, widening the channel and decreasing flow resistance. The 2D and 3D simulations involve moving meshes whose quality is maintained by prescribing the mesh displacement on guide surfaces positioned on either side of each flexible structure. The prescribed displacement depends on structure bending and maintains mesh quality even for large deformations. Simulations demonstrate effective pumping even at Reynolds numbers as low as 0.001. Because rod rigidity may be specified independently of Reynolds number, in principle, rod rigidity may be reduced to enable pumping at arbitrarily low Reynolds numbers.

1:03PM R9.00002 Using micro-3D printing to build acoustically driven microswimmers.1

NICOLAS BERTIN, CNRS Grenoble, OLIVIER STEPHAN, PHILIPPE MARMOTTANT, Université Joseph Fourier, Grenoble, TAMIS SPELLMAN, ERIC LAUGA, University of Cambridge, DYFCOM TEAM2, COMPLEX AND BIOLOGICAL FLUIDS TEAM3 — With no protection, a micron-sized free bubble at room temperature in water has a life span shorter than a few tens of seconds. Using two-photon lithography, which is similar to 3D printing at the micron scale, we can build “armors” for these bubbles: micro-capsules with an opening to contain the bubble and extend its life to several hours in biological buffer solutions. When excited by an ultrasound transducer, a 20 µm bubble performs large amplitude oscillations in the capsule opening and generates a powerful acoustic streaming flow (velocity up to dozens of mm/s). A collaboration with the Dept. of Applied Mathematics and Theoretical Physics, University of Cambridge, is helping us predict the true resonance of these capsules and the full surrounding streaming flow. The present Bubbleboost project aims at creating red blood cell sized capsules (~ 10-20 µm) that can move on their own with a non-contact acoustic excitation for drug delivery applications. Another application of this research is in microfluidics: we are able to fabricate fields of capsules able to generate mixing effects in microchannels, or use the bubble-generated flow to guide passing objects at a junction.

1 ERC Grant Agreement Bubbleboost no. 614655.
2 LPHY, université Joseph Fourier
3 Department of Applied Mathematics and Theoretical Physics, University of Cambridge

1:16PM R9.00003 Inertial microfluidic pump

PAVEL KORNILOVITCH, ALEXANDER GOVYADINOV, DAVID MARKE, ERIK TORNIAIERN, HP Inc — The inertial pump is powered by a microheater positioned near one end of a fluidic microchannel. As the microheater explosively boils the surrounding fluid, a vapor bubble expands and then collapses asymmetrically, resulting in net flow. Such devices become an effective means of transporting fluids at microscale. They have no moving parts and can be manufactured in large numbers using standard batch fabrication processes. In this presentation, physical principles behind pump operation are described, in particular the role of reservoirs in dissipating mechanical momentum and the expansion-collapse asymmetry. An effective one-dimensional dynamic model is formulated and solved. The model is compared with full three-dimensional CFD simulations and available experimental data. Potential applications of inertial micropumps are described.

1:29PM R9.00004 Convective flow reversal in self-powered enzyme micropumps

HENRY SHUM, University of Pittsburgh, ISAMAR ORTIZ-RIVERA, ARJUN AGRAWAL, AYUSMAN SEN, The Pennsylvania State University, ANNA BALAZS, University of Pittsburgh — It was recently shown that a surface-bound patch of enzymes in a fluid filled chamber can drive large scale flow in the presence of the enzyme’s substrate. Evidence suggested that the flow was buoyancy driven but the pumping speed, or even direction, was not always consistent with estimates based on heat released by the reaction. Hence, we develop and analyze a model for even direction, was not always consistent with estimates based on heat released by the reaction. Hence, we develop and analyze a model for dynamic model is formulated and solved. The model is compared with full three-dimensional CFD simulations and available experimental data. Potential applications of inertial micropumps are described.

1:42PM R9.00005 Chemically generated convective transport of micron sized particles

OLEG SHKLAYEV, The University of Pittsburgh, SAMBETTA DAS, ALCIA ALTEMESO, The Pennsylvania State University, HENRY SHUM, ANNA BALAZS, The University of Pittsburgh, AYUSMAN SEN, The Pennsylvania State University — A variety of chemical and biological applications require manipulation of micron sized objects like cells, viruses, and large molecules. Increasing the size of particles up to a micron reduces performance of techniques based on diffusive transport. Directional transport of cargo toward detecting elements reduces the delivery time and improves performance of sensing devices. We demonstrate how chemical reactions can be used to organize fluid flows carrying particles toward the assigned destinations. Convection is driven by density variations caused by a chemical reaction occurring at a catalyst or enzyme-covered target site. If the reaction causes a reduction in fluid density, as in the case of catalytic decomposition of hydrogen peroxide, then fluid and suspended cargo is drawn toward the target along the bottom surface. The intensity of the fluid flow and the time of cargo delivery are controlled by the amount of reagent in the system. After the reagent has been consumed, the fluid pump stops and particles are found aggregated on and around the enzyme-coated patch. The pumps are reusable, being reactivated upon injection of additional reagent. The developed technique can be implemented in lab-on-a-chip devices for transportation of micro-scale object immersed in solution.
1:55PM R9.00006 Optimized open-flow mixing: insights from microbubble streaming
BHARGAV RALLABANDI, Department of Mechanical and Aerospace Engineering, University of Illinois at Urbana-Champaign, CHENG WANG, Department of Mechanical and Aerospace Engineering, Missouri University of Science and Technology. LIN GUO, SASCHA HILGENFELDT, Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — Microbubble streaming has been developed into a robust and powerful flow actuation technique in microfluidics. Here, we study it as a paradigmatic system for microfluidic mixing under a continuous throughput of fluid (open-flow mixing), providing a systematic optimization of the device parameters in this practically important situation. Focusing on two-dimensional advective stirring (neglecting diffusion), we show through numerical simulation and analytical theory that mixing in steady streaming vortices becomes ineffective beyond a characteristic time scale, necessitating the introduction of unsteadiness. By duty cycling the streaming, such unsteadiness is introduced in a controlled fashion, leading to exponential refinement of the advection structures. The rate of refinement is then optimized for particular parameters of the time modulation, i.e. a particular combination of times for which the streaming is turned "on" and "off". The optimized protocol can be understood theoretically using the properties of the streaming vortices and the throughput Poiseuille flow. We can thus infer simple design principles for practical open flow micromixing applications, consistent with experiments.

1Current Address: Mechanical and Aerospace Engineering, Princeton University

2:08PM R9.00007 Magnetically Actuated Cilia for Microfluidic Manipulation
SRINIVAS HANASOGE, DREW OWEN, MATT BALLARD, PETR J HESKETH, ALEXANDER ALEXEEV, Georgia Institute of Technology, WOODRUFF SCHOOL OF MECHANICAL ENGINEERING COLLABORATION, PETIT INSTITUTE FOR BIOENGINEERING AND BIOSCIENCES COLLABORATION — We demonstrate magnetic micro-cilia based microfluidic mixing and capture techniques. For this, we use a simple and easy to fabricate high aspect ratio cilia, which are actuated magnetically. These micro-features are fabricated by evaporating NiFe alloy at room temperature, on to patterned photoresist. The evaporated alloy curls upwards when the seed layer is removed to release the cilia, thus making a free standing ‘C’ shaped magnetic microstructure. This is actuated using an external electromagnet or a rotating magnet. The artificial cilia can be actuated up to 20Hz. We demonstrate the active mixing these cilia can produce in the microchannel. Also, we demonstrate the capture of target species in a sample using these fast oscillating cilia. The surface of the cilia is functionalized by streptavidin which binds to biotin labelled fluorescent microspheres and mimic the capture of bacteria. We show very high capture efficiencies by using these methods. These simple to fabricate micro cilia can easily be incorporated into many microfluidic systems which require high mixing and capture efficiencies.

2:21PM R9.00008 3D flow focusing for microfluidic flow cytometry with ultrasonics
VASKAR GNYAWALI, Department of Mechanical and Industrial Engineering, Ryerson University, Toronto, Canada, ERIC M. STROHM, YASAMAN DAGHIGHI, MIA VAN DE VONDervoort, MICHAEL C. KOLIos, Department of Physics, Ryerson University, Toronto, Canada, SCOTT S.H. TSAI, Department of Mechanical and Industrial Engineering, Ryerson University, Toronto, Canada — We are developing a flow cytometer that detects unique acoustic signature waves generated from single cells due to interactions between the cells and ultrasound waves. The generated acoustic waves depend on the size and biomechanical properties of the cells and are sufficient for identifying cells in the medium. A microfluidic system capable of focusing cells through a 10 x 10 μm ultrasound beam cross section was developed to facilitate acoustic measurements of single cells. The cells are streamlined in a hydro-dynamically 3D focused flow in a 300 x 300 μm channel made using PDMS. 3D focusing is realized by lateral sheath flows and an inlet needle (inner diameter 100 μm). The accuracy of the 3D flow focusing is measured using a dye and detecting its localization using confocal microscopy. Each flowing cell would be probed by an ultrasound pulse, which has a center frequency of 375 MHz and bandwidth of 250 MHz. The same probe would also be used for recording the scattered waves from the cells, which would be processed to distinguish the physical and biomechanical characteristics of the cells, eventually identifying them. This technique has potential applications in detecting circulating tumor cells, blood cells and blood-related diseases.

2:34PM R9.00009 Viscoelastic focusing and separation of bioparticles in straight microchannels
GUOQING HU, CHAO LIU, LNM, Institute of Mechanics, Chinese Academy of Sciences — Viscoelasticity-induced particle migration has recently received increasing attention due to its ability to obtain high-quality focusing over a wide range of flow rates. However, its application is limited to low throughput regime since the particles can defocus as flow rate increases. Using an engineered carrier medium with constant and low viscosity and strong elasticity, the sample flow rates are improved to be one order of magnitude higher than those in existing studies. Utilizing differential focusing of particles of different sizes, here we present shearless particle/cell separation in simple straight microchannels that possess excellent parallelizability for further throughput enhancement. The present method can be implemented over a wide range of particle/cell sizes and flow rates. We successfully separate small particles from larger particles, MCF-7 cells from red blood cells (RBCs), and Escherichia coli (E. coli) bacteria from RBCs in different straight microchannels. We recommend further study on engineering rheological properties of carrier media for improving the separation performance of viscoelasticity-based microfluidic devices.

1We thank MOST2011CB707604 and NSFC11272321 for financial support.

2:47PM R9.00010 Paper-based flow fractionation system for preconcentration and field-flow fractionation
SEOKBIN HONG, Sogang University, RHOKYUN KWAK, Korea Institute of Science and Technology, WONJUNG KIM, Sogang University — We present a novel paper-based flow fractionation system for preconcentration and field-flow fractionation. The paper fluidic system consisting of a straight channel connected with expansion regions can generate a fluid flow with a constant flow rate for 10 min without any external pumping devices. The flow bifurcates with a fraction ratio of up to 30 depending on the control parameters of the channel geometry. Utilizing this simple paper-based bifurcation system, we developed a continuous-flow preconcentator and a field-flow fractionator on a paper platform. Our experimental results show that the continuous-flow preconcentrator can produce a 33-fold enrichment of the ion concentration and that the flow fractionation system successfully separates the charged dyes. Our study suggests simple, cheap ways to construct preconcentration and field-flow fractionation systems for paper-based microfluidic diagnostic devices.

1This work was supported by the National Research Foundation of Korea(NRF) grant funded by the Korea government(MSIP) (NRF-2015R1A2A2A04006181).
3:00PM R9.00011 Fabrication of thermo-responsive microfluidic membrane using photopolymerization patterning, HYEJEONG KIM, SANG JOON LEE, Center for Biofluid and Biomimic Research, Department of Mechanical Engineering, Pohang University of Science and Technology, Pohang, 790-784 — The programmed manipulation of responsive functional hydrogels is receiving large attention because of its unique functions and wide range of engineering applications. In this study, we developed an innovative stomata-inspired membrane (SIM) by fabricating a temperature-responsive hydrogel with a simple, cost-effective, and high-throughput photopolymerization patterning process. Polymerization-induced diffusion on the macro-scale surface gives rise to form a multi-partied polymer membrane with fine pores by simple UV irradiation. After heating the SIM, the less deformable thick frame supports the whole structure, and the highly deformable thin base regulates the size of pores. The morphological configuration of the SIM can be easily changed by varying the solution composition or selecting a suitable photomask with different pattern. The developed SIM has the special sensing-to-actuation functions of stimuli-responsive hydrogels. This membrane with temperature-responsive pores would be potentially utilized in numerous practical applications, such as filter membranes with self-adjustable pores, membrane-based sensors, membrane-based actuators, and multi-functional membranes etc.

1This study was supported by the National Research Foundation of Korea (NRF) and funded by the Korean government (MSIP) (Grant No. 2008-0061991).

3:13PM R9.00012 Roll-to-Roll Nanoimprint Lithography Simulations for Flexible Substrates, ANDREW SPANN, AKHILESH JAIN, ROGER BONNECAZE, University of Texas at Austin — UV roll-to-roll nanoimprint lithography enables the patterning of features onto a flexible substrate for bendable electronics in a continuous process. One of the most important design goals in this process is to make the residual layer thickness of the photosis in unpatterned regions as thin and uniform as possible. Another important goal is to minimize the imprint time to maximize throughput. We develop a multi-scale model to simulate the spreading of photosis drops as the template is pressed against the substrate. We include the effect of capillary pressure on the bending of the substrate and show how this distorts uniformity in the residual thickness layer. Our simulation code is parallelized and can simulate the flow and merging of thousands of drops. We investigate the effect of substrate tension and the initial arrangement of drops on the residual layer thickness and imprint time. We find that for a given volume of photosis, distributing that volume to more drops initially decreases the imprint time. We conclude with recommendations for scale-up and optimal operations of roll-to-roll nanoimprint lithography systems.

Tuesday, November 24, 2015 12:50PM - 2:21PM – Session R10 Convection and Buoyancy-Driven Flows: General 110 - Daria Frank, University of Cambridge

12:50PM R10.00001 Aerodynamical sealing by air curtains, DARIA FRANK, PAUL LINDEN, University of Cambridge — Air curtains are artificial high-velocity plane turbulent jets which are installed in a doorway in order to reduce the heat and the mass exchange between two environments. The performance of an air curtain is assessed in terms of the sealing effectiveness E, the fraction of the exchange flow prevented by the air curtain compared to the open-door situation. The main controlling parameter for air curtain dynamics is the deflection modulus Dm, representing the ratio of the momentum flux of the air curtain and the transverse forces acting on it due to the stack effect. In this talk, we examine the influence of two factors on the performance of an air curtain: the presence of an additional ventilation pathway in the room, such as a small top opening, and the effects of an opposing buoyancy force which for example arises if a downwards blowing air curtain is heated. Small-scale experiments were conducted to investigate the E(Dm)-curve of an air curtain in both situations. We present both experimental results and theoretical explanations for our observations. We also briefly illustrate how simplified models developed for air curtains can be used for more complex phenomena such as the effects of wind blowing around a model building on the ventilation rates through the openings.

1:03PM R10.00002 Thermal Convection on an Ablating Target, IGBAL MEHMEDAGIC, U. S. Army, ARDEC, Picatinny Arsenal, NJ, SIVA THANGAM, Stevens Institute of Technology, NJ — Modeling and analysis of thermal convection of a metallic targets subject to radiative flux is of relevance to various manufacturing processes as well as for the development of protective shields. The present work involves the computational modeling of metallic targets subject to high heat fluxes that are both steady and pulsed. Modeling of the ablation and associated fluid dynamics when metallic surfaces are exposed to high intensity pulsed laser fluence at normal atmospheric conditions is considered. The incident energy from the laser is partly absorbed and partly reflected by the surface during ablation and subsequent vaporization of the convecting melt also participates in the radiative exchange. The energy distribution during the process between the bulk and vapor phase strongly depends on optical and thermodynamic properties of the irradiated material, radiation wavelength, and laser pulse intensity and duration. Computational findings based on effective representation and prediction of the heat transfer, melting and vaporization of the targeting material as well as plume formation and expansion are presented and discussed in the context of various ablation mechanisms, variable thermo-physical and optical properties, plume expansion and surface geometry.

1:16PM R10.00003 Collective motion of multiple rafts on the Rayleigh-Benard convection, FAHRUDIN NUGROHO, DEWI LITA MARTANTI, RYAN PRATAMA, AGUNG BAMBANG S.U., PEKIK NURWANTORO, Department of Physics, Gadjah Mada University, Yogyakarta, Indonesia, DIAN ARTHA K, Faculty of Teaching and Educational Technology, Ahmad Dahlan University, Yogyakarta, Indonesia — We observe the dynamics of rafts on the Rayleigh-Benard convection. A single raft motion shows at least three types of motions i.e. the linear, oscillatory, and random motions. The velocity of single raft fits with the gaussian distribution function. While the multiple rafts case show more complex motion, including the possibility of collective motion. We show that there is an indication of collective motion of multiple rafts as one of the law of motion in the Rayleigh-Benard convection.
1:29PM R10.00004 Mixing and Displacement Buoyancy-Driven Exchange Flow Between Adjacent Zones, SALEH NABI, Mitsubishi Electric Research Laboratories, MORRIS FLYNN, University of Alberta — Buoyancy-driven flow between two finite zones containing fluid of slightly different density is investigated. The two zones are separated by either a single common doorway or top and bottom vents. In the former case, a two-layer exchange flow develops once the barrier is removed. A buoyant plume of light fluid mixes with the dense fluid leading, over time, to the development of non-trivial ambient density stratification. Meanwhile, a gravity current propagates into the light zone, which upon reflection and reaching the doorway in a form of an internal bore, alters the dynamics of the exchange flow. The exchange flow is also significantly altered if and when the first front in the dense zone falls below the top of the doorway, in which case an intermediate layer develops in the light zone. Conversely, when the two zones are separated by top and bottom vents, two oppositely directed exchange flows are generated. The transient evolution of the interface, stratification and buoyancy in the dynamics of the exchange flow. The exchange flow is also significantly altered if and when the first front in the dense zone falls below the top of the doorway, in which case an intermediate layer develops in the light zone. Conversely, when the two zones are separated by top and bottom vents, two oppositely directed exchange flows are generated. The transient evolution of the interface, stratification and buoyancy in each zone are estimated both for the case where the light zone does and does not contain a source of buoyancy. Simultile experiments help to identify the limitations of the analytical models for each scenario.

1:42PM R10.00005 Influence of mushy zone constant on the solid-liquid phase change process modeled by enthalpy-porosity technique, SAEED TIARI, MAHBOOBE MAHDAVI, SONGGANG QIU, Department of Mechanical Engineering, Temple University, Philadelphia, PA — In the present work, the effects of mushy zone constant on the melting and solidification processes simulation with enthalpy-porosity technique are investigated. The isothermal melting and solidification of gallium enclosed by a rectangular container is studied using a transient two-dimensional finite volume based model. A wide range of mushy zone constants are considered in the study of the thermal and fluid flow characteristics of the system. The results indicate that increasing the mushy zone constant value up to a limit leads to the acceleration of the solidification process, while it decreases the melting rate. However, the further increase of the constant does not affect the phase change process in melting and solidification. It is found that the mushy zone constant has a significant influence on the temperature distribution adjacent to the melt front and overall capability of the model. This is due to the effect of mushy zone constant on the flow in the mushy region and liquid layers nearby. The results also reveal that the increase of mushy zone constant results in the decrease of average wall heat flux in both melting and solidification processes.

1:55PM R10.00006 Coolant Design System for Liquid Propellant Aerospike Engines, MIRANDA MCCONNELL, Tennessee Technological University, RICHARD BRANAM, The University of Alabama — Liquid propellant rocket engines burn at incredibly high temperatures making it difficult to design an effective coolant system. These particular engines prove to be extremely useful by powering the rocket in a variable thrust that is ideal for space travel. When combined with aerospace engine nozzles, which provide maximum thrust efficiency, this class of rockets offers a promising future for rocketry. In order to troubleshoot the problems that high combustion chamber temperatures pose, this research took a computational approach to heat analysis. Chambers milled into the combustion chamber walls, lined by a copper cover, were tested for their efficiency in cooling the hot copper wall. Various aspect ratios and coolants were explored for the maximum wall temperature by developing our own MATLAB code. The code uses a nodal temperature analysis with conduction and convection equations and assumes no internal heat generation. This heat transfer research will show oxygen is a better coolant than water, and higher aspect ratios are less efficient at cooling.

2:08PM R10.00007 Bounds on heat transport in Rayleigh’s and related models of Bénard convection, CHARLES R. DOERING, ANDRE N. SOUZA, University of Michigan, BAOLE WEN, The University of Texas at Austin, GREGORY P. CHINI, University of New Hampshire, RICHARD R. KERSWELL, University of Bristol — We present new upper limits on convective heat transport in both the full and several low-dimensional Galerkin truncations of Rayleigh’s 1916 model of buoyancy-driven Bénard convection using both the so-called background method as well as optimal control variational techniques.

Tuesday, November 24, 2015 12:50PM - 2:34PM –
Session R11 Convection and Buoyancy-Driven Flows: Experimental Studies 111 - Yogesh Jaluria, Rutgers University

12:50PM R11.00001 Experimental Investigation of Transport Enhancement in Convective Air Flow by the Use of a Vortex Promoter, YOGESH JALURIA, Rutgers Univ, KEVIN GOMES, GE Company — This paper focuses on the effect of placing a passive vortex generator in a flow and the resulting increase in transport rates. The flow circumstance considered is that of a flat plate with protruding heat sources, placed in a uniform flow, with a vortex generator located upstream of the leading edge. The study consists of three parts. In the first part, the flow due to the vortex promoter by itself is considered. The periodic or chaotic behavior in the wake behind the promoter is investigated. By studying different sizes and shapes of vortex promoters, it is determined which configuration offers the largest disturbance in the flow and the frequency at which it occurs. In the second part of the study, the flow over a plate with isolated, finite-sized, protruding heat sources, without a vortex promoter, is considered. Again, the frequency of the disturbance downstream is investigated to determine the nature of the resulting flow and the disturbance frequency. The effect of varying the dimensions and locations of the heat sources on the flow downstream is investigated. It is found that a larger separation distance between two sources leads to higher transport rates. In the last part of the study, tests are done for the combination of vortex promoter and the plate, placing a vortex promoter in front of the plate. An effort is made to match the frequencies of the disturbances due to the vortex generator with those due to the plate in an attempt to achieve resonance. From these results, an optimal promoter is chosen that would lead to maximum heat transfer rate.

1:03PM R11.00002 Transient Convection from Forced to Natural with Flow Reversal on a Vertical Flat Plate, BLAKE W. LANCE, BARTON L. SMITH, Utah State University — Transient flow through the forced, mixed and natural convection regimes is studied experimentally on a vertical flat plate. Measurements are ensemble-averaged and include velocity from Particle Image Velocimetry and high fidelity thermal measurements in walls both for temperature and heat flux. The flow is a ramp-down flow transient encompassing all three convection regimes. The initial condition is forced convection downward with subsequent transition to mixed convection, ending with natural convection upward after a flow reversal. Velocity measurements provide time-mean and Reynolds stress profiles across the span of the test section. Near-wall data provide shear stress estimates by fitting a line to the data in the inner portion of the viscous sublayer. Wall heat flux is measured in the plate with thin film heat flux sensors. Flow reversal was observed near the heated plate and turbulence kinetic energy was redistributed from the heated boundary layer towards the freestream. Wall heat flux was decreased and shear stress decreased then reversed. These data are part of a CFD validation dataset meant to assess simulation accuracy and the final case made available for mixed convection.
1:16PM R11.00003 Physical modelling of LNG rollover in a depressurized container filled with water. PETR DENISSENKO, MAKSIM DADONAU, University of Warwick, UK, ANTOINE HUBERT, SIYAKA DEMBELE, Kingston University, UK, JENNIFER WEN, University of Warwick, UK — Stable density stratification of multi-component Liquefied Natural Gas causes it to form distinct layers, with upper layer having a higher fraction of the lighter components. Heat flux through the walls and base of the container results in buoyancy-driven convection accompanied by heat and mass transfer between the layers. The equilibration of densities of the top and bottom layers, normally caused by the preferential evaporation of Nitrogen, may induce an imbalance in the system and trigger a rapid mixing process, so-called rollover. Numerical simulation of the rollover is complicated and codes require validation. Physical modelling of the phenomenon has been performed in a water-filled depressurized vessel. Reducing gas pressure in the container to levels comparable to the hydrostatic pressure in the water column allows modelling of tens of meters industrial reservoirs using a 20 cm laboratory setup. Additionally, it allows to model superheating of the base fluid layer at temperatures close the room temperature. Flow visualizations and parametric studies are presented. Results are related to outcomes of numerical modelling.

1:29PM R11.00004 Exchange flow of two immiscible Newtonian fluids in a vertical tube. PRISCILLA VARGES, FERNANDA NASCENDES, BRUNO FONSECA, PAULO ROBERTO DE SOUZA MENDES, MONICA NACCACHE, PUC-Rio — Plugging cementing is an essential operation performed under a variety of well conditions. The cement plugs are rarely placed at the intended depth because the cement slurry usually is heavier than the well fluid. Failures are due primarily to migration of the denser fluid down the well at the top of which it is discharged. The aim of the research is to better understand the process of plugging operation in vertical wells. To this end, we performed an experimental and theoretical study of the buoyancy-driven flow of two immiscible Newtonian fluids in a vertical tube such that the heavier and more viscous fluid is placed on top. Since both fluids are Newtonian, the situation is always unstable, i.e. the fluid on top will always flow downward and displace the bottom fluid upwards, so that the relative positioning tends to invert. The influence of the governing parameters on the speed of inversion was investigated. Flow visualization was performed with a digital camera, and inversion velocities were obtained through image analysis. Preliminary results show that inversion speed decreases as the tube diameter is increased, increases as the viscosity ratio is increased, and also decreases as the density ratio is increased.

1:42PM R11.00005 A cryostat device for liquid nitrogen convection experiments. CHARLES DUBOIS, ALEXIS DUCHESNE, HERVE CAPS, GRASP - University of Liege — When a horizontal layer of expandable fluid heated from below is submitted to a large vertical temperature gradient, one can observe convective cells. This phenomenon is the so-called Rayleigh-Bénard instability. In the literature, this instability is mainly studied when the entire bottom surface of a container heats the liquid. Under these conditions, the development of regularly spaced convective cells in the liquid bulk is observed. Cooling applications led us to consider this instability in a different geometry, namely a resistor immersed in a bath of cold liquid. We present here experiments conducted with liquid nitrogen. For this purpose, we developed a cryostat in order to be able to perform Particle Image Velocimetry. We obtained 2D maps of the flow and observed, as expected, two Rayleigh-Bénard convective cells around the heater. We particularly investigated the vertical velocity in the central column between the two cells. We compared these data to results we obtained with silicone oil and water in the same geometry. We derived theoretical law from classical models applied to the proposed geometry and found a good agreement with our experimental data.

1:55PM R11.00006 The shape and behaviour of a horizontal buoyant jet adjacent to a surface. HENRY BURRIDGE, Department of Applied Mathematics and Theoretical Physics, University of Cambridge, GARY HUNT, Department of Engineering, University of Cambridge — We investigate the incompressible turbulent buoyant jet formed when fluid is steadily ejected horizontally from a circular source into a quiescent environment of uniform density. As our primary focus, we introduce a horizontal boundary. By definition, large source-boundary separations, the jet attaches and ‘clings’ to the boundary before, further downstream, pulling away from the boundary. Based on measurements of saline jets in freshwater we deduce the conditions required for a jet to cling. We present data for the variation in volume flux, flow envelope and centreline for both ‘clinging’ and ‘free’ jets. For source Froude numbers Fr0 ≥ 12 the data collapses when scaled, identifying universal behaviours for both clinging jets and for free jets.

2:08PM R11.00007 Dissolution patterns on caramel blocks. CAROLINE COHEN, JULIEN DERR, MICHAEL BERHANU, SYLVAIN COURRECH DU PONT, Laboratoire Matiere et Systeme Complexes, Universite Paris Diderot — We investigate erosion by dissolution processes. We perform laboratory experiments on hard caramel bodies, which dissolve on a short timescale, compared to geological material such as limestone. We put a block of caramel, tilted from the horizontal, in a water tank without flow. The dissolution syrup, which is denser than pure water, sinks and the flow detaching from the surface creates patterns underneath the caramel block. These patterns result from the coupled dynamics of the flow detaching and the eroding surface and are reminiscent of scallops observed in the walls of phreatic cave passages. We investigate the mechanisms of formation of these structures and their evolution depending on several parameters such as the fluid density or the flow velocity. We finally parallel the formation of patterns on melting iceberg.

2:21PM R11.00008 Spatio-temporal intermittency in stratified shear flow: effects of Prandtl number. ADRIEN LEFAUVE, PAUL LINDEN, DAMTP, University of Cambridge — We present laboratory experiments of a stratified shear flow in an inclined square duct, connecting two reservoirs of water at different densities. The exchange flow in which a layer of dense salt-water flows beneath a layer of lighter freshwater moving in the opposite direction is known to host a rich zoo of behaviors. As the driving density difference or inclination angle is increased, the initially sharp and flat density interface can support Holmboe waves, and then transition to a statistically steady turbulent intermediate mixed layer. Here we report on the significantly different dynamics observed when stratification is achieved by heat instead of salt. Even moderate values of density difference or inclination angle now allow the dramatic growth of interfacial waves causing the flow to transition to a fully-turbulent state, before it relaxes back to a laminar state and so forth. These novel laminar-turbulent cycles exhibit a remarkable periodicity and suggest that the Prandtl number (Pr = 700 for salt vs Pr = 7 for heat) affects the intermittency and mixing properties of stratified turbulence.

1Supported by EPSRC Programme Grant EP/K034529/1 entitled “Mathematical Underpinnings of Stratified Turbulence”

Tuesday, November 24, 2015 12:50PM - 3:26PM — Session R12 Granular Flows: Fluctuations and Instabilities 200 - Nathalie Friend, University of Cambridge
12:50PM R12.00001 Avalanches in a V-shape: inverted roll-waves and a curved free surface¹, NATHALIE FRIEND, University of Cambridge, DAMTP, JIM MCELWaine, Durham University, Department of Earth Sciences — In this work, we create avalanches in a V-shaped channel at different apertures and flowrates. For deep flows (at high apertures and flowrates), roll waves are triggered that have surprising features due to the chosen V-shape geometry. For shallower flows (low to medium apertures), the effects of roll waves are reduced and/or eliminated and the base flow appears. This background base flow is characterized by a curved free surface and recirculation cells whose structure and position is a strong function of the flowrate. An alternative rheology is proposed which accounts for this background base flow in terms of second-order stress differences.

¹NMV acknowledges support from the Royal Society (ref. no. DHI20121)

1:03PM R12.00002 Continuum modelling of piston driven shock waves through granular gases and ensuing pattern formations, NICK SIRMAS, MATEI RADULESCU, University of Ottawa — Two-dimensional event-driven Molecular Dynamics (MD) simulations were previously completed to investigate the stability of piston driven shock waves through dilute granular gases. By considering viscoelastic collisions, allowing for finite dissipation within the shock wave, instabilities were found in the form of distinctive high density non-uniformities and convective rolls within the shock structure. This work is now extended to the continuum level. Euler and Navier-Stokes equations for granular gases are modelled with a modified cooling rate to include an impact threshold necessary for inelastic collisions. The shock structure predicted by the continuum formulation is found in good agreement with the structure obtained by MD. Non-linear stability analyses of the travelling wave solution are performed, showing a neutrally stable structure and responding only to fluctuations in the upstream state. Introducing strong perturbations to the incoming density field, in accordance with the spacial fluctuations in upstream state seen in MD, yields similar instabilities as those previously observed. While the inviscid model predicts a highly turbulent structure from these perturbations, the inclusion of viscosity yields comparable wavelengths of pattern formations to those seen in MD.

1:16PM R12.00003 Plane shock waves and Haff’s law in a granular gas, LAKSHMINARAYANA REDDY, MEHEBOOB ALAM, Jawaharlal Nehru Centre for Advanced Scientific Research, Jakkur PO, Bangalore 560064 — The Riemann problem of planar shock waves is analyzed for a dilute granular gas by solving Euler- and Navier-Stokes-type equations numerically. The density and temperature profiles are found to be asymmetric, with the maxima of both density and temperature occurring within the shock-layer. The density-peak increases with increasing Mach number and inelasticity, and is found to propagate at a steady speed at late times. The granular temperature at the upstream end of the shock decay according to Haff’s law $\theta(t) \sim t^{-\frac{1}{2}}$, but the downstream temperature decays faster than its upstream counterpart. The Haff’s law seems to hold inside the shock up to a certain time for weak shocks, but deviations occur for strong shocks. The time at which the maximum temperature deviates from Haff’s law follows a power-law scaling with upstream Mach number and the restitution coefficient. The continual build-up of density inside the shock is discussed, the origin of which seems to be tied to a pressure instability in granular gases. It is shown that the granular energy equation must be ‘regularized’ to arrest the maximum density, and the regularized hydrodynamic equations should be used for shock calculations (Reddy & Alam, 2015, J. Fluid Mech., to be published).

1:29PM R12.00004 Rarefaction effects in dilute granular Poiseuille flow: Knudsen minimum and temperature bimodality, ACHAL MAHAJAN, MEHEBOOB ALAM, Jawaharlal Nehru Centre for Advanced Scientific Research, Jakkur PO, Bangalore 560064 — The gravity-driven flow of smooth inelastic hard-disks through a channel, analog of granular Poiseuille flow, is analyzed using event-driven simulations. We find that the variation of the mass-flow rate ($Q$) with Knudsen number ($Kn$) can be non-monotonic in the elastic limit (i.e. the restitution coefficient $\epsilon_n \rightarrow 1$) in channels with very smooth walls. The Knudsen minimum effect (i.e. the minimum flow rate occurring at $Kn \sim O(1)$ for the Poiseuille flow of a molecular gas) is found to be absent in a granular gas with $\epsilon_n \leq 0.99$, irrespective of wall roughness. Another rarefaction phenomenon, the bimodality of the temperature profile, with a local minimum at the channel centerline and two symmetric maxima $(T_{\text{max}})$ away from the centerline, is studied. We show that the inelastic dissipation is responsible for the onset of temperature bimodality [i.e. the excess temperature, $\Delta T = (T_{\text{max}}/T_{\text{min}} - 1) \neq 0$ near the continuum limit $(Kn \sim 0)$, but the rarefaction being its origin (as in molecular gas) holds beyond $Kn \sim O(0.1)$]. The competition between dissipation and rarefaction seems to be responsible for the observed dependence of both mass-flow rate and temperature bimodality on $Kn$ and $\epsilon_n$. [Alam et al. 2015, JFM (revied)].

1:42PM R12.00005 Transition in a granular chute flow due to periodic and aperiodic perturbations, BHARATHRAJ S, KUMARAN V, Department of Chemical Engineering, IISc Bangalore — Granular flow down an inclined plane exhibits a transition from a disordered, random state to an ordered state with layers of particles with in-layer hexagonal order, when there is a small change in the roughness of the base. In earlier studies, a rough base was created using a random arrangement of frozen particles at the base, and the roughness was varied by varying the ratio of the frozen and moving particle diameters. Here, the effect of a different form of base roughness, which is sinusoidal perturbations of varying amplitude and wavelength, is also examined. The transition from an ordered to disordered state is also observed when a sinusoidal base is used, when the amplitude of the sine wave increases beyond a critical value. The critical amplitude initially increases as the wavelength is increased, reaches a maximum and then decreases as the wavelength is further increased. The critical amplitude also increases as the height of the flow increases. The states induced by the sinusoidal base have peculiar transient features, where there is a tendency to order at intermediate times in disordered states, unlike the rough base where no such tendency is observed. We also formulate a boundary layer theory for the ordered state, which develops in two distinct stages of shear propagation.

1:55PM R12.00006 Nonlinear instability and convection in a vertically vibrated granular bed, PRIYANKA SHUKLA, Department of Mathematics, IIT Madras, Chennai, India, I.H. ANSARI, Jawaharlal Nehru Centre for Advanced Scientific Research, Jakkur PO, Bangalore 560064, India, D. VAN DER MEER, DETLEF LOHSE, POF Group, University of Twente, Enschede, The Netherlands, MEHEBOOB ALAM, Jawaharlal Nehru Centre for Advanced Scientific Research, Jakkur PO, Bangalore 560064, India The nonlinear instability of the density-inverted granular Leidenfrost state and the resulting convective motion in strongly shaken granular matter are analysed via a weakly nonlinear analysis. Under a quasi-steady ansatz, the base state temperature decreases with increasing height away from the vibrating plate, but the density profile consists of three distinct regions: (i) a collisional dilute layer at the bottom, (ii) a levitated dense layer at some intermediate height and (iii) a ballistic dilute layer at the top of the granular bed. For the nonlinear stability analysis, the nonlinearities up-to cubic order in perturbation amplitude are retained, leading to the Landau equation. The genesis of granular convection is to be tied to a subcritical pitchfork bifurcation from the Leidenfrost state. Near the bifurcation, the equilibrium amplitude is found to follow a square-root scaling law, $A_\text{c} \sim \sqrt{\Delta}$, with the distance $\Delta$ from bifurcation point. The strength of convection is maximal at some intermediate value of the shaking strength, with weaker convection both at weaker and stronger shaking. Our theory predicts a novel floating-convection state at very strong shaking [Shukla et al, JFM (2014), vol. 761, p. 123-167].
2:08PM R12.00007 Low-frequency oscillation in a narrow vibrated granular system. LORETO OYARTE GLVEZ, DEVARAJ VAN DER MEER, University of Twente — The analogy of the behaviour of granular materials with that of fluids has motivated much appealing research. An important example is a vertically shaken granular bed which exhibits fluid-like behavior, such as the Leidenfrost effect where a dense layer of grains floats on top of a gaseous layer, just like when a liquid droplet floats on its own vapour above a hot plate. When the shaking energy is increased the granular bed transits from the Leidenfrost to the convection state, for which a precursor is expected in the form of an oscillation of the bed as a whole. This precursor was observed numerically like an oscillation in the motion of the dense part, where the frequency of this oscillation is much lower than the frequency of the injected energy, and appears more relevant when the system is getting closer to the convective state. We built a setup that permits the observation of the granular Leidenfrost effect for a wide range of driving parameters. More specifically, a monodisperse granular material is contained in a transparent box and vertically shaken, and a fast camera is used to study its dynamics. The presence of a LFO is directly measured by images analysis and shows a good agreement with the previous numerical and experimental works.

2:21PM R12.00008 Motion of a Short Granular Polymer in Vibrations. P.C. HUANG, NTU, JACK WU, C.Y. TAO, Inst. of Physics. Academia Sinica, Y.Y. CHEN, NTU, J.C. TSAI, Inst. of Physics. Academia Sinica — Using both numerical simulations and laboratory experiments, we investigate the motions of a short granular polymer driven by vibrations. Surprisingly, our minimal models of constrained point masses with a simple assumption on the momentum transfer not only reproduce the rapid ratcheting motion in prior experiments [Phys. Rev. Lett. 112, 058001 (2014)], but also reveal the crucial role of random noises in triggering the spontaneous switching of bouncing modes in our previous report [http://meetings.aps.org/link/BAPS.2014.DFD.H24.1]. Subsequent experiments with a bead chain vibrated in an annular channel allow uninterrupted observations on the granular polymer. From the long-time statistics, we correlate the horizontal displacements to the different modes of response and identify the characteristic timescales for the transitions. Cross examinations of the numerical models and the statistical experiments suggest certain generic ratcheting and mode transitions that are insensitive to the mechanical details of such polymer.

2:34PM R12.00009 Transverse Diffusion in Bedload Transport. OLIVIER DEVAUCHELLE, ANAIS ABRAMIAN, GREGOIRE SEIZILLES, ERIC LAJEUNESSE, Institut de Physique du Globe de Paris — When a fluid flows over a granular bed, it entrains the grains as bedload. This interaction produces a beautiful variety of shapes and landscapes, such as dunes, ripples and meanders. In this context, Coulomb’s law of friction translates into a threshold shear stress, above which the grains are entrained. When the flow-induced stress is barely above this threshold, only a small proportion of the superficial grains move. Their trajectory is then strongly influenced by the layer of static grains below them. They mostly move in the flow direction, but the roughness of the underlying bed causes their velocity to fluctuate, and turns their trajectory into a random walk. As a consequence, bedload diffuses in the direction orthogonal to the flow. Laboratory experiments suggest that this diffusion opposes gravity to maintain the banks of a river. However, quantifying the terms of this balance remains an experimental challenge. We propose to use an instability generated by bedload diffusion to do so.

2:47PM R12.00010 On creating macroscopically identical granular systems with different numbers of particles. DEVARAJ VAN DER MEER, NICOLAS RIVAS, University of Twente, The Netherlands — One of the fundamental differences between granular mechanics and fluid hydrodynamics is the enormous difference in the total number of constituents. The small number of particles implies that the role of fluctuations in granular dynamics is of paramount importance. To obtain more insight in these fluctuations, we investigate to what extent it is possible to create identical granular hydrodynamic states with different number of particles. A definition is given of macroscopically equivalent systems, and the dependency of the conservation equations on the particle size is studied. We show that, in certain cases, and by appropriately scaling the microscopic variables, we are able to compare systems with significantly different number of particles that present the same macroscopic phenomenology. We apply these scalings in simulations of a vertically vibrated system, namely the density inverted granular Leidenfrost state and its transition to a buoyancy-driven convective state.

3:00PM R12.00011 Rare events in granular media: a volcanic-like explosion. EVGENY KHAIN, Oakland University, LEONARD SANDER, University of Michigan — Granular matter is ubiquitous in nature and exhibits a variety of nontrivial phenomena. Within the same system, different regions of granular media can be at a solid or a gas phase. Here we focus on a granular Leidenfrost effect: a solid-like cluster is levitating above the “hot” granular gas [1]. This state was observed experimentally, when granular matter was vertically vibrated in a two-dimensional container [2]. This solid-gas coexistence can be described by using granular hydrodynamics, taking into account the viscosity divergence in the solid cluster. The approach is similar to the one employed in investigating solid-fluid coexistence in dense shear granular flows [3]. We performed extensive molecular dynamics simulations of a simple model of inelastic hard spheres driven by a “thermal” bottom wall. Simulations showed that for low wall temperatures, the levitating cluster is stable, while for high wall temperatures, it breaks down, and a hot gas bursts out resembling a volcanic explosion. We found a hysteresis for a wide range of bottom wall temperatures, both the clustering state and the volcanic state are stable. However, even if the system is at the (stable) clustering state, a volcanic explosion is possible: it is a rare event driven by large fluctuations. We propose a simple simulation technique that allows investigating such rare events.

3:13PM R12.00012 Evolution of injected air stream in granular bed. RITWIK MAITI, ph.D. Student, GARGI DAS, PRASANTA DAS, Professor — An air stream injected through an orifice into a granular bed creates intriguing but aesthetically exotic patterns. The interaction of an air with an aggregate of cohesionless granules presents evolution of patterns from stationary bubble to meandering filament and finally to a floating canopy with the increase of air velocity.

Tuesday, November 24, 2015 12:50PM - 3:26PM – Session R13 Aerodynamics: Unsteady Aerodynamics II: Flapping and Flexible Wings
201 - Mingjun Wei, New Mexico State University
12:50PM R13.00001 Adjoint-based optimization for the understanding of the aerodynamics of a flapping plate

Supported by AFOSR

1:03PM R13.00002 Wing-Fixed PIV and force measurements of a large transverse gust encounter

1:16PM R13.00003 Proper Orthogonal Decomposition of Flow-Field in Non-Stationary Geometry

1:29PM R13.00004 On the correlation between force production and the flow field around a flapping flat-plate wing

1:42PM R13.00005 Effect of advanced and delayed rotation on the dominant flow pattern and its temporal evolution

1:55PM R13.00006 A Lagrangian approach to study flow topology around a flapping flat-plate wing

The incredible flight performance of insects can be attributed in part to the generation and maintenance of stable regions of vorticity, which is achieved by manipulating the wing kinematics. Along with the prolonged attachment of the leading edge vortex, the wing reversal mechanisms form the basis by which insects regulate the magnitude and direction of forces produced. The duration and starting point of these directional flips are studied in the current experimental investigation. Particle image velocimetry is conducted to evaluate the flow features inherent to changes in wing reversal during the stroke of a flat plate, which is modelled based on hoverfly characteristics. The duration of rotation is one-third of the total time period. A +10% phase shift is used for delayed rotation, a -10% phase shift for advanced rotation. Phase-averaged data is analysed to understand the influence of a delayed or advanced rotation on the formation and evolution of large and small scale structures, their interactions with the wing, and disintegration. Additionally, force data is used to quantify the effects of phase-shift in terms of lift and drag variation and is correlated with the vortex dynamics.

During a flapping cycle of an insect, complex time dependent flows are produced as the wing reciprocates, producing a maximum lift at the stroke reversals. By flipping the wing rapidly at the end of each stroke, the insect modulates the flow around the wing and hence the aerodynamic forces necessary to hover. The duration and starting point of the flip play an important role in determining the amount of lift produced. To understand and tailor the effect of wing kinematics on the aerodynamic performance we focussed on the vortex dynamics of the flow field. Phase-averaged data from particle image velocimetry was used to evaluate the flow features inherent to changes in rotation during a stroke of a flat plate, which is modelled based on hoverfly characteristics. The period of rotation is one-third of the total time period. A +10% phase shift is used for delayed rotation, a -10% phase shift for advanced rotation. Vortex detection methods like the $\Gamma_{\lambda}$ and $\Gamma_{\gamma}$ criteria are used to determine the effect of a delay or early rotation on the trajectories, size, shape and location of the prominent vortical structures. Proper orthogonal decomposition is used to study the influence of the phase-shifts on the dominant mode structure and the related time-scales.

The incredible flight performance of insects can be attributed in part to the generation and maintenance of stable regions of vorticity, which is achieved by manipulating the wing kinematics. Along with the prolonged attachment of the leading edge vortex during translation of the wing, the rotational motion at the end of the stroke is critical as it generates large amounts of lift required for the insect to remain air-borne while hovering. The wing reversal entails a change in the flow-field around the wing which is closely tied to variations in force production. Based on phase-averaged particle image velocimetry data we analyze the effect of a shift in the rotational phase of a flapping wing on the flow characteristics. A topological study is conducted using Lagrangian vortex detection techniques in order to characterize the shear layer formation, vortex interactions and flow separation. The Lagrangian analysis includes the calculation of Finite Time Lyapunov Exponents based on particle trajectories. An objective approach is employed to trace the location of separation or attachment points as an indication for changes in the strength, stability and shedding frequencies of vortices. These trajectories are correlated with fluctuations in aerodynamic force coefficients.

Forces and fluid velocities of this wing-gust interaction will be presented for two pre-gust conditions: attached flow on the wing and stalled flow over the wing. In both cases, the gust encounter results in a momentary spike in lift coefficient. The peak lift coefficient was measured between 3 and 6 and varies with angle of attack. At low angle of attack, the attached flow wing produces less lift before the gust and much more (non-circulatory) lift during the gust than the stalled wing. Although the flow over the wing at low angle of attack separates during the gust and reattaches afterwards, the recovery time is similar to that of the high angle case, on the order of 10 chord lengths travelled.

A topological study is conducted using Lagrangian vortex detection techniques in order to characterize the shear layer formation, vortex interactions and flow separation. The Lagrangian analysis includes the calculation of Finite Time Lyapunov Exponents based on particle trajectories. An objective approach is employed to trace the location of separation or attachment points as an indication for changes in the strength, stability and shedding frequencies of vortices. These trajectories are correlated with fluctuations in aerodynamic force coefficients.

Based on phase-averaged particle image velocimetry data we analyze the effect of a shift in the rotational phase of a flapping wing on the flow characteristics. A topological study is conducted using Lagrangian vortex detection techniques in order to characterize the shear layer formation, vortex interactions and flow separation. The Lagrangian analysis includes the calculation of Finite Time Lyapunov Exponents based on particle trajectories. An objective approach is employed to trace the location of separation or attachment points as an indication for changes in the strength, stability and shedding frequencies of vortices. These trajectories are correlated with fluctuations in aerodynamic force coefficients.
2:08PM R13.00007 Vortical Flow Structures in the Near-Wake of a Heaving Airfoil with Passively Actuated Leading and Trailing Flaps.\textsuperscript{1} FIRAS SIALA, ALEXANDER TOTPAL, JAMES LIBURDY, Oregon State University — The flow physics of flying animals has recently received significant attention, mostly in the context of developing bio-inspired micro air vehicles and oscillating flow energy harvesters. Of particular interest is the understanding of the impact of airfoil flexibility on the flow physics. Research efforts showed that some degree of surface flexibility enhanced the strength and size of the leading edge vortex. In this study, the influence of flexibility on the near-wake dynamics and flow structures is investigated using 2D PIV measurements. The experiments are conducted in a wind tunnel at a Reynolds number of 30,000 and a range of reduced frequencies from 0.09 to 0.2. The flexibility is attained using a torsion rod forming a hinge between the flap and the main wing. Vortex flow structures are visualized using large eddy scale decomposition technique and quantified using swirling strength analysis. It is found that trailing edge flexibility increases the vortex swirling strength compared to a rigid airfoil, whereas leading edge flexibility decreases the swirling strength. Furthermore, the integral length scale of the auto-correlated velocity fluctuations is found to be approximately equal to the actual vortex size. The vortex convective velocity is shown to be independent of flexibility and oscillation frequency, and it is represented by a trimodal distribution, with peak values at 0.8, 0.95 and 1 times the free stream velocity.

\textsuperscript{1}Oregon State University

2:21PM R13.00008 Chord-wise Tip Actuation on Flexible Flapping Plates\textsuperscript{1}. NATHAN MARTIN, MORTEZA GHARIB, Caltech — The aerodynamic characteristics of low aspect ratio flapping plates are strongly influenced by the interaction between tip and edge vortices. This has led to the development of tip actuation mechanisms which bend the tip towards the root of the plate in the span-wise direction during oscillation to investigate its impact. In our current work, a tip actuation mechanism to bend a flat plate’s two free corners towards one another in the chord-wise direction is developed using a shape memory alloy. The aerodynamic forces and resulting flow field are investigated from dynamically altering the tip chord-wise curvature while flapping. The frequency of oscillation, stroke angle, flexibility, and tip actuation timing are independently varied to determine their individual effects. These results will further the fundamental understanding of flapping wing aerodynamics.

\textsuperscript{1}This material is based upon work supported by the National Science Foundation Graduate Research Fellowship under Grant No. DGE 1144469.

2:34PM R13.00009 Efficient passive pitching motion caused by elastic deformation in flexible flapping wing MAVs. TRONG NGUYEN, TIEN TRUONG, KHOON SENG YEO, TEE TAI LIM, National University of Singapore — Computational and experimental models which mimic Hawkmoth wings were constructed to investigate the effects of wing flexibility. The wing actuation mechanism is minimized with only one degree of freedom in sweeping motion with neither active pitching nor elevation. Despite the simplicity of the imparted motion, the wing models in both computations and experiments delivered convincing deformation features such as wing twisting and camber which closely resembles the ones observed in real Hawkmoth wings. The generated aerodynamic forces are remarkable both in magnitude and efficiency. The study hence reveals that a complicated actuation mechanism might not be required to produce the sophisticated and efficient motion of insect wings, which in fact could be the result of collective elastic deformation thanks to their highly optimized structure mainly comprised of well-organized veins and membranes.

2:47PM R13.00010 On the thrust performance of a 2D flapping foil in a forward flight condition. SUNIL MANOHAR DASH, KIM BOON LIA, TEE TAI LIM, National University of Singapore — Past studies have shown that the thrust performance of a 2D airfoil undergoing simple harmonic motion in both pitch and heave in a forward flight condition is dependent on maximum effective angle of attack ($\alpha_{\text{o}}$) and Strouhal number ($S_{T}$). For a given $\alpha_{\text{o}}$, it is found that the thrust coefficient ($C_{T}$) increases with $S_{T}$ until it reaches a peak value at the critical Strouhal number ($S_{Tc}$); beyond which $C_{T}$ deteriorates considerably. In order to extend $S_{Tc}$ and therefore increase the max. $C_{T}$, the airfoil must oscillate at a higher $\alpha_{\text{o}}$. Further, it is found that, regardless of $\alpha_{\text{o}}$, thrust degradation is accompanied by cessation of the induced effective angle of attack profile ($\alpha(t)$) to exhibit simple harmonic function of time. As to why the dynamic function of $\alpha(t)$ is detrimental to thrust generation is not fully understood. In an attempt to better understand this phenomenon, both numerical simulations and comparative experiments are performed on a 2D flapping elliptic foil at Re of 5000. Our results show that the proximity of the leading edge vortex from the previous stroke to the oscillating foil plays a crucial role in the thrust generation. Detailed results will be discussed in the presentation.

3:00PM R13.00011 Locomotion of a flapping flexible plate in ground effect. XI-YUN LU, CHAO TANG, University of Science and Technology of China — Locomotion of a three-dimensional flapping flexible plate in ground effect is studied numerically by the coupled solution of the fluid flow and the plate motion. When the leading-edge of the flexible plate is forced to take a vertical oscillation near a ground, the plate moves freely due to the fluid-structure interaction. Mechanisms underlying the dynamics of the plate near the ground are elucidated. The ground effect can enhance propulsive speed and improve propulsive efficiency, especially in the medium bending stiffness regime. The analysis of unsteady dynamics and deformation of plate indicates that the ground effect becomes weaker for more flexible plate. Therefore it is found that a suitable degree of flexibility can improve the propulsive performance in ground effect. The vortical structure and pressure distribution around the plate and their connection with the dynamics of the plate are also investigated.

3:13PM R13.00012 Thrust and Lift generation of heaving and pitching oscillating foil propulsion in ground effect. AMIN MIVEHCHI, PhD Candidate, University of Rhode Island, JASON M. DAHL, STEPHEN LICHT, Assistant Professor, University of Rhode Island — Experimental results are presented for the thrust and lift generation on a NACA0012 airfoil undergoing heave and pitch oscillation near a solid boundary. For ground effect in the steady flow over a lifting surface, lift and drag forces are altered by an enhanced spanwise flow around the tip of the lifting surface, resulting in a strong low pressure region on the upper part of the wing and increased lift in the presence of a boundary. In the present study, this effect is investigated for an inherently unsteady flow, a propulsive flapping foil. It is found that ground effect has a significant effect on the instantaneous and average lift and thrust forces generated by the oscillating foil, and the lift and propulsive force. It is found that the forces on a flapping foil in the presence of the ground is not only dependent on the aspect ratio but shows high dependency on the kinematics of motion such as maximum angle of attack, frequency of flapping, and the distance from the ground. The relation between these parameters and their effect on the cycle averaged thrust, lift, propulsive efficiency, and instantaneous force over the airfoil is shown. It is hypothesized that ground effect may be used as a proxy sensor for identifying solid boundaries with biomimetic underwater vehicles. Keywords: Ground effect, Flapping foil propulsion, flow-structure interaction.
This study focused on comparing the load characteristics of symmetric, thin (NACA-0009) and thick (NACA-0021) airfoils at low Re numbers. This research has led to increased interest in understanding the characteristics of airfoils at Reynolds number regimes between observed flow structures and instantaneous load on the airfoils, as well as the aerodynamic load characteristics of thin and thick airfoils at low Re Numbers.

For the aligned interaction, the incident vortex induces a tip vortex of opposite in-plane vorticity; decrease the downwash; and increase the root-mean-square of both streamwise velocity and vorticity. The vortex, which leads to volume representations and thereby characterization of the streamwise evolution of the vortex structure as it approaches the trailer wing. The evolution of the incident vortex is affected by the upstream influence of the trailer wing, and is highly dependent on the location of vortex impingement. As the spanwise impingement location of the vortex moves from outboard of the wing tip to inboard, the upwash on the development of the vortex increases. For spanwise locations close to or intersecting the vortex core, the effects of upstream influence of the wing on the vortex are to: increase the streamwise velocity deficit; decrease the streamwise vorticity; increase the in-plane vorticity; decrease the downwash; and increase the root-mean-square of both streamwise velocity and vorticity.

The interaction of a perturbed trailing vortex with a wing is explored in the context of formation flight. The incident vortex is generated from a leader wing at a fixed angle-of-attack, which is subjected to controlled oscillation in the vertical direction over a range of frequencies and at amplitudes an order of magnitude smaller than the chord of the wing. Particle image velocimetry is used to determine the flow structure of the perturbed vortex along the stationary follower wing. Two spanwise locations of vortex impingement are characterized: aligned with and inboard of the tip of the wing. Images of streamlines, vorticity and turbulent kinetic energy reveal the temporal and spatial evolution of the vortex structure. For the aligned interaction, the incident vortex induces a tip vortex of opposite signed vorticity on the wing thereby forming a dipole. For the inboard interaction, the incident vortex induces a vortex of same sign vorticity at the wing-tip and a shear layer of opposite sign across the wing surface. The upwash of the incident vortex promotes flow separation at the leading edge for both of these interactions; this separation gives rise to substantial levels of turbulent kinetic energy that persist downstream through the region of reattachment.

The immersed boundary method is used to simulate the incompressible flow around two-dimensional airfoils at low Reynolds numbers in order to investigate the self-starting and self-rotating capability of a vertical axis wind turbine (VAWT) with NACA 0018 blades. By examining the torque generated by a three-bladed VAWT fixed at various orientations, a stable equilibrium and the optimal starting orientation that produces the largest torque have been observed. When Reynolds number is below a critical value, the VAWT oscillates around a stable equilibrium. However, the VAWT goes into continuous rotation from the optimal orientation when Reynolds number is above this critical value. It is also shown that VAWT with more blades is easier to self-start due to a wider range of positive starting torques. Moreover, with a proper choice of load model, a VAWT is able to self-rotate and generate a designed averaged power.

This project is supported by Caltech FLOWE center/Gordon and Betty Moore Foundation.

The present study investigates the flow over a golf ball and a smooth sphere around the critical Reynolds numbers under both stationary and self-spinning conditions by conducting Large-eddy simulations (LES) based on high resolution unstructured grids. For the stationary cases, the present calculation results validate the promotion of the drag crisis at a relatively lower Reynolds number due to the golf ball dimples. It also shows that the golf ball dimples have a limited effect on the time-dependent lateral force development in the subcritical regime, whereas the dimples are beneficial in suppressing the lateral force oscillations in the supercritical regime. For both spinning models, the inverse Magnus effect was reproduced in the critical regime, whereas in the supercritical regime the ordinary Magnus force was generated. Relatively weaker lift forces were also observed in the cases of the spinning golf balls when compared to the spinning smooth spheres.

At these low Re numbers, aerodynamics of an airfoil is influenced by laminar separation and its possible reattachment, which is in contrast to airfoil behavior at high Re numbers. This study focused on comparing the load characteristics of symmetric, thin (NACA-0009) and thick (NACA-0021) airfoils at low Re numbers ~2 – 4 × 10^4, and angles of attack between 2° to 12°, along with simultaneous flow visualization. The experiments were performed in a low speed flow visualization water tunnel facility, and two-component Laser Doppler Velocimetry was used to quantify the inflow conditions and turbulence intensity. A high precision force/torque transducer was used for the load measurements, while hydrogen bubble technique was used to visualize the airflow in the region of reattachment between observed flow structures and instantaneous load on the airfoils, as well as the aerodynamic load characteristics of thin and thick airfoils at low Re Numbers.
2:08PM R14.00007 On the lift increments with the occurrence of airfoil tones at low Reynolds numbers\(^1\), TOMOAKI IKEDA, Japan Aerospace Exploration Agency, DAI SUKE FUJIMOTO, AYUMU INASAWA, MASAHTO ASAI, Tokyo Metropolitan University — The aeroacoustic effects on the aerodynamics of an NACA 0006 airfoil are investigated experimentally at relatively low Reynolds numbers, \(Re = 30,000 - 70,000\). By employing two wind-testing airfoil models at different chord lengths, \(L = 40\) and \(100\) [mm], the aerodynamic dependence on Mach number is examined at a given Reynolds number. In a particular range of Reynolds number, tonal peaks of trailing-edge noise are obtained from a shorter-chord airfoil, while no apparent tones are observed with longer chord length at a lower Mach number. Suppression of the occurrence of a tonal noise leads to a greater lift slope in the present wind-tunnel experiment, evaluated via a PIV approach. The lift curves obtained experimentally at higher Mach numbers agree well with two-dimensional numerical simulations, performed at \(M = 0.2\). At the Mach number, the numerical results clearly indicate the occurrence of an acoustic feedback loop with discrete tones, within a range of angle of attack. A few three dimensional numerical results are also presented. In the simulation at \(Re = 50,000\), the suppression of tonal noise corresponds to the development of a turbulent wedge in the suction-side boundary layer at the angle of attack 4.0 [deg.], which agrees with the experiment.

\(^1\)This work was supported by Grant-in-Aid for Scientific Research from Japan Society for the Promotion of Science (Grant No. 25420139).

2:21PM R14.00008 Reynolds number effects on flow over twisted cylinder with drag reduction and vortex suppression\(^1\), JAE HWAN JUNG, HYUN Sik YOON, Pusan National University — We investigated the Reynolds number effects on the flow over a twisted cylinder in the range of \(310 \leq Re \leq 110^4\). To analyze the effect of the twisted cylinder, a large eddy simulation (LES) with a dynamic subgrid model was employed. A simulation of the smooth cylinder was also carried out to compare the results with those of the twisted cylinder. As Re increased, the mean drag and lift coefficient of the twisted cylinder increased with the same tendency as those of the smooth cylinder. However, the increases in the mean drag and lift coefficient of the twisted cylinder were much smaller than those of the smooth. Furthermore, elongated shear layer and suppressed vortex shedding from the twisted cylinder occurred compared to those of the smooth cylinder, resulting in a drag reduction and suppression of the vortex-induced vibration (VIV). In particular, the twisted cylinder achieved a significant reduction of over 96% in VIV compared with that of the smooth cylinder, regardless of increasing Re. As a result, we concluded that the twisted cylinder effectively controlled the flow structures with reductions in the drag and VIV compared with the smooth cylinder, irrespective of increasing Re.

\(^1\)This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP) through GCRC-SOP (No. 2011-0030015) and (NRF-2015R1D1A1A01020807)

2:34PM R14.00009 Reynolds Number Effects on Helicopter Rotor Hub Flow, DAVID REICH, STEVE WILLITS, SVEN SCHMITZ, The Pennsylvania State University — The 12 inch diameter water tunnel at the Pennsylvania State University Applied Research Laboratory was used with the objective of quantifying effects of Reynolds number scaling on drag and shed wake of model helicopter rotor hub flows. Hub diameter-based Reynolds numbers ranged from 1.06 million to 2.62 million. Measurements included steady and unsteady hub drag, as well as Particle Image Velocimetry. Results include time-averaged, phase-averaged, and spectral analysis of the drag and wake flow-field. A strong dependence of steady and unsteady drag on Reynolds number was noted, alluding to the importance of adequate Reynolds scaling for model helicopter rotor hubs that exhibit interaction between various bluff bodies.

2:47PM R14.00010 Numerical Investigations of an Optimized Airfoil with a Rotary Cylinder, KOMAL GADA\(^1\), HAMID RAHA\(^2\), CEERS/COE/CSULB — Numerical Investigations of an optimized thin airfoil with a rotary cylinder as a control device for reducing separation and improving lift to drag ratio have been performed. Our previous investigations have used geometrical optimization for development of an optimized airfoil with increased torque for applications in a vertical axis wind turbine. The improved performance was due to contributions of lift to torque at low angles of attack. The current investigations have been focused on using the optimized airfoil for micro-uav applications with an active control device, a rotary cylinder, to further control flow separation, especially during wind gust conditions. The airfoil has a chord length of 19.66 cm and a width of 25 cm with 0.254 cm thickness. Previous investigations have shown flow separation at approximately 85% chord length at moderate angles of attack. Thus the rotary cylinder with a 0.254 cm diameter was placed slightly downstream of the location of flow separation. The free stream mean velocity was 10 m/sec. and investigations have been performed at different cylinder’s rotations with corresponding tangential velocities higher than, equal to and less than the free stream velocity. Results have shown more than 10% improvement in lift to drag ratio when the tangential velocity is near the free stream mean velocity.

\(^1\)Graduate Assistant, Center for Energy and Environmental Research and Services (CEERS), College of Engineering, California State University, Long Beach

\(^2\)Professor and Director

3:00PM R14.00011 Comparative Study of Airfoil Flow Separation Criteria, NICK LAWS, WAAD KAHOUJI, BRENDEN EPPS, Thayer School of Engineering, Dartmouth College — Airfoil flow separation impacts a multitude of applications including turbomachinery, wind turbines, and bio-inspired micro-aerial vehicles. In order to achieve maximum performance, some devices are constructed to promote the edge of flow separation, and others use dynamic flow separation advantageous. Numerous criteria exist for predicting the onset of airfoil flow separation. This talk presents a comparative study of a number of such criteria, with emphasis paid to speed and accuracy of the calculations. We evaluate the criteria using a two-dimensional unsteady vortex lattice method, which allows for rapid analysis (on the order of seconds instead of days for a full Navier-Stokes solution) and design of optimal airfoil geometry and kinematics. Furthermore, dynamic analyses permit evaluation of dynamic stall conditions for enhanced lift via leading edge vortex shedding, commonly present in small flapping-wing flyers such as the bumblebee and hummingbird.

3:13PM R14.00012 A Computational Modeling Mystery Involving Airfoil Trailing Edge Treatments, YEUNUN CHOO, BRENDEN EPPS, Thayer School of Engineering, Dartmouth College — In a curious result, Fairman (2002) observed that steady RANS calculations predicted larger lift than the experimentally-measured data for six different airfoils with non-traditional trailing edge treatments, whereas the time average of unsteady RANS calculations matched the experiments almost exactly. Are these results reproducible? If so, is the difference between steady and unsteady RANS calculations a numerical artifact, or is there a physical explanation? The goals of this project are to solve this thirteen year old mystery and further to model viscous/load coupling for airfoils with non-traditional trailing edges. These include cupped, beveled, and blunt trailing edges, which are common anti-singing treatments for marine propeller sections. In this talk, we present steady and unsteady RANS calculations (ANSYS Flows) with careful attention paid to the possible effects of asymmetric unsteady vortex shedding and the modeling of turbulence anisotropy. The effects of non-traditional trailing edge treatments are visualized and explained.
12:50PM R15.00001 Metriplectic Simulated Annealing, P.J. Morrison, The University of Texas at Austin, G.R. Flierl, MIT — Metriplectic dynamics [1,2] is a general form for dynamical systems that represent the first and second laws of thermodynamics, energy conservation and entropy production. Entropy production provides asymptotic stability to equilibrium states, which because of constraints need not be trivial. The formalism will be used to perform quasigeostrophic computations, akin to those of [3], for obtaining a variety of vortex states.


1:03PM R15.00002 ABSTRACT WITHDRAWN

1:16PM R15.00003 A high order multi-resolution solver for the Poisson equation with application to vortex methods, Mads Mlholm Hejlesen, Henrik Juel Spitz, Technical University of Denmark, Jens Honoré Walthier, Technical University of Denmark and ETH Zurich — A high order method is presented for solving the Poisson equation subject to mixed free-space and periodic boundary conditions by using fast Fourier transforms (FFT). The high order convergence is achieved by deriving mollified Green’s functions from a high order regularization function which provides a correspondingly smooth solution to the Poisson equation. The high order regularization function may be obtained by deblurring algorithms used in image processing. At first we show that the regularized solution can be combined with a short range particle-particle correction for evaluating discrete particle interactions in the context of a particle-particle particle-mesh (P²PM) method. By a similar approach we extend the regularized solver to handle multi-resolution patches in continuum field simulations by super-positioning an inter-mesh correction. For sufficiently smooth vector fields this multi-resolution correction can be achieved without the loss of convergence rate. An implementation of the multi-resolution solver in a two-dimensional re-meshed particle-mesh based vortex method is presented and validated.

1:29PM R15.00004 Coupling of a compressible vortex particle-mesh method with a near-body compressible discontinuous Galerkin solver, Philippe Parmentier, Gégoire Winckelmans, Philippe Chatelain, Université catholique de Louvain (UCL) - Institute of Mechanics, Materials and Civil Engineering (iMMC), Koen Hillewaert, Cenaeo — A hybrid approach, coupling a compressible vortex particle-mesh method (CVPM, also with efficient Poisson solver) and a high order compressible discontinuous Galerkin Eulerian solver, is being developed in order to efficiently simulate flows past bodies; also in the transonic regime. The Eulerian solver is dedicated to capturing the anisotropic flow structures in the near-wall region whereas the CVPM solver is exploited away from the body and in the wake. An overlapping domain decomposition approach is used. The Eulerian solver, which captures the region away from the body and in the wake, also provides the outer boundary conditions to the Eulerian solver. Because of the coupling, a boundary element method is also required for consistency. The approach is assessed on typical 2D benchmark cases.

1Supported by the Fund for Research Training in Industry and Agriculture (F.R.I.A.)

1:42PM R15.00005 The Finite Time Lyapunov Exponent Field of N Interacting Vortices in the Zero Viscosity Limit, Richard Galvez, Syracuse University, Vanderbilt University, Melissa Green, Syracuse University — We present an analysis of the Finite Time Lyapunov Exponent (FTLE) field of interacting vortices in the potential flow limit. This work is based on an inviscid approximation, but develops a useful tool that will aid in the effort of understanding the interactions of vortices and turbulence in viscous fluids. The FTLE field of N interacting vortices is computed numerically in two dimensions in different physical scenarios: i) orbiting one another with no initial velocities, ii) approaching each other given an initial velocity and iii) as periodically produced behind a circular cylinder. For situation ii) we expand on the cases where the approach velocities of the vortices are less than or greater than a critical capture velocity, that is, the velocity necessary to escape a captured orbit between co-rotating vortices. We focus on the evolution and interaction of the Lagrangian coherent structures (LCS) in these scenarios to determine if there is a way to anticipate the character of vortex interaction by the initial structure of the LCS. Additional remarks will be made on the extrapolation of observations to a large number of interacting vortices (large N).

1This work was supported by the Air Force Office of Scientific Research under AFOSR Award No. FA9500-14-1-0210.

1:55PM R15.00006 New Vortex Shedding Criteria for Low Order Models of Unsteady Plate Motion, Field Manar, Anya Jones, Univ of Maryland-College Park — A complex potential flow model with a small number of point vortices of time-varying strength is developed to evaluate the flow around an infinitely thin flat plate undergoing arbitrary unsteady motion. Vortex strengths are determined using the Kutta condition, and vortex convection takes place according to an impulse-matching scheme. Previous work has had only limited success due to vortices not being properly shed from the plate and acquiring too much circulation. In this work, a new vortex shedding criterion based on the dynamics of the shear layer is investigated. This criterion seeks to approximate the occurrence of vortex pinch off by observing the tangential velocities in the shear layer. The effect of the new vortex-shedding criteria on the evolution of the flow are evaluated with respect to previous shedding criteria and experimental PIV results. One motivation for the development of this model is to predict the unsteady forces on a wing quickly, and at low computational cost. Given the velocity field computed via the complex potential model, the forces on the plate are computed by taking the time derivative of the total flow momentum, and are evaluated with respect to experimental measurements.
which clearly improves over the results obtained with the swirling strength criterion in a number of relevant two-dimensional case studies. The inner layer of turbulent boundary layer flows is, in particular, the region where the swirling strength criterion looses presence of intense vortical structures; (ii) vortex merging; (iii) spurious vortices created in many-vortex configurations and (iv) in the presence of background shear. The inner layer of turbulent boundary layer flows is, in particular, the region where the swirling strength criterion looses accuracy in a dramatic way. We propose an alternative vortex detection criterion, based on the curvature properties of the vorticity profile, which clearly improves over the swirling strength criterion in a number of relevant two-dimensional case studies.

A higher-order asymptotic formula for velocity of a viscous vortex pair . YASUHIDE FUKUMOTO, Institute of Mathematics for Industry, Kyushu University, UMMU HABIBAH, Graduate School of Mathematics, Kyushu University — We establish a general formula for the traveling speed of a counter-rotating vortex pair, being valid for thick cores, moving in an incompressible fluid with and without viscosity. Two-dimensional motion of vortices with finite cores, interacting with each other, has been extensively studied both analytically and numerically. Mathematical methods and numerical schemes have been highly developed for dealing particularly with vortices of uniform vorticity, called vortex patches. In contrast, this is not the case with vortices with distributed vorticity. We extend, to a higher order, the method of matched asymptotic expansions developed by Ting and Tung (1965 Phys. Fluids Vol. 8 pp. 1039-1051). The solution of the Navier-Stokes equations is constructed in the form of a power series in a small parameter, the ratio of the core radius to the distance between the core centers. A correction due to the effect of finite thickness of the vortices to the traveling speed makes its appearance at the 5th order. We manipulate a tidy formula of this correction term for a general vorticity distribution at the leading order. An alternative route to reach the same formula is also sought. We devise a two-dimensional counterpart of Helmholtz-Lamb’s formula which is applicable to vortex rings.

Classification and transitions of streamline topologies of structurally stable incompressible flows1 . TAKASHI SAKAJO, Kyoto University, TOMOO YOKOYAMA, Kyoto Univ. of Education — We consider Hamiltonian vector fields with a dipole singularity satisfying the slip boundary condition in two-dimensional multiply connected domains. An example of such Hamiltonian vector fields is an incompressible and inviscid flow in exterior multiply connected domains with a uniform flow, whose Hamiltonian is called the stream function. Here, we are concerned with streamline topologies of incompressible fluid flows, which are the level sets of the Hamiltonian. We first provide a classification procedure to assign a unique sequence of words, called the maximal words, to every structurally stable streamline pattern. Owing to this procedure, we can identify every streamline pattern with its representing sequence of words up to topological equivalence. In addition, based on the theory of word representations, we propose a combinatorial method to provide a list of possible transient structurally unstable streamline patterns between two different structurally stable patterns by simply comparing their maximal word representations without specifying any Hamiltonian. It reveals the existence of many non-trivial global transitions in a generic sense. We also demonstrate how the present theory is applied to fluid flow problems with vortex flows.

Construction of initial vortex-surface fields and Clebsch potentials for flows with high-symmetry using first integrals , PENGYU HE, YUE YANG, Peking University — We develop a systematic methodology to construct the explicit, general form of vortex-surface fields and Clebsch potentials based on first integrals of the characteristic equation of a given three-dimensional velocity-vorticity field. This methodology is successfully applied to the initial fields with the zero helicity density and high symmetry, e.g., initial fields with the Taylor-Green and the Kida-Pelz symmetries.

Contour surgery in multiply-connected domains , RHODRI NELSON, Kyoto University — In this talk we present a new method for computing the motion of vortex patches in multiply connected domains. The method works by first solving for the velocity field owing to an unbounded vortex at appropriate points on the boundaries (as if the boundaries were not present). Following this, a suitable modified Schwarz-problem is solved to give a ‘correction’ velocity such that the sum of this field and that due to the ‘unbounded’ vortex satisfy the no-normal flow boundary condition on all boundaries present. For flows in which complex distributions of vortex evolve, the algorithm performs contour surgery (allowing vortices to split or merge) to allow accurate, long time integration of such systems.

Added mass and critical mass in vortex induced vibration , EFS-TATHIOS KONSTANTINIDIS, Department of Mechanical Engineering, University of Western Macedonia, Kozani 50100 — The critical mass phenomenon is the observation that a circular cylinder suspended freely in a fluid stream without a mechanical restoring force exhibits significant vortex induced vibration if its mass is below some value whereas insignificant vibration occurs if the mass is above this value. While the phenomenon is known, its origin remains largely unknown. Furthermore, there are several outstanding questions regarding this phenomenon which cannot be explained on the basis of the existing theoretical framework. In this work, a new formulation of the added mass in the context of potential flow is presented. This leads to a new expression for the potential force, which is more complex than the classical one, that is subsequently employed in simplified form in order to analytically model the flow-structure interaction by decomposing the fluid force into potential and vortex components via the equation of cylinder motion. It is found that the model predicts a significant increase in the amplitude response of a freely suspended cylinder in sharp contrast to predictions using the classical formulation of the added mass. Finally, the model equations are employed to exemplify the phenomenology of the critical mass in real flows.

Tuesday, November 24, 2015 12:50PM - 3:13PM
Session R16 Flow Instability: Interfacial and Thin Films V 204 - Lou Kondic, New Jersey Institute of Technology
is time remaining to rupture. When the end walls as well as the surface temperature distributions.

realized. In order to understand flow patterns, we focus on the flow structures of the thermocapillary convection in a cross section normal to added a non-uniform temperature distribution to the film by placing a heated iron at one end of the ring, a net flow toward the heated iron was found that a unique flow pattern is induced. One of the present authors, DRP, carried out a series of experiments under microgravity condition fluid is driven to the colder to hotter regions by the non-uniform surface-tension distribution. In the case of thin free liquid film, however, it is stable, axisymmetric mode in the Newtonian limit. Our model predicts unstable modes at the strongly non-Newtonian limit, and show that the characteristic wavelength of the tongues increases with flux. Theoretically, we model the symmetry breaking as flow instability origin, where the viscous layer is thick, the flow is dominated by vertical shear. In the outer region where the viscous layer is thinner, it floats over the inviscid fluid. The viscous fluid is released axisymmetrically at constant flux, and is driven by gravity. Near the origin, where the viscous layer is thick, the flow is dominated by vertical shear. In the outer region where the viscous layer is thinner, it floats over the inviscid layer and the dominant stress is horizontal. The floating region of such flows remains axisymmetric when the viscous fluid is Newtonian. In contrast, when the viscous fluid is non-Newtonian, the floating region can be distributed in an array of extensional tongues. We use experimental and theoretical analysis to study the symmetry breaking of the extensional region. Experiments using polymeric fluids show that the characteristic wavelength of the tongues increases with flux. Theoretically, we model the symmetry breaking as flow instability of a power-law fluid that becomes Newtonian at low strain rates. He prepared a ring made of metal, and formed a thin film of water inside the ring. Once he prepared a ring made of metal, and formed a thin film of water inside the ring. Once he added a non-uniform temperature distribution to the film by placing a heated iron at one end of the ring, a net flow toward the heated iron was realized. In order to understand flow patterns, we focus on the flow structures of the thermocapillary convection in a cross section normal to the end walls as well as the surface temperature distributions.

1:03PM R16.00002 Features of the Interface Equation Coupling Thin and Thick Film Regimes in Conduction-Triggered Thermocapillary Flows, ZACHARY NICOLAOU, SANDRA TROIAN, California Institute of Technology, 1200 E California Blvd MC 128-95, Pasadena, CA 91125 — An attractive feature of moving boundary problems involving the coupling of adjacent thin film regimes is the simplification of the corresponding interface equation. For interfaces subject to conduction-triggered thermocapillary forces and damping by capillary forces, the evolution equation reduces to a 4th order nonlinear PDE. The dispersion equation for linear instability of a uniform state then reduces to Type II, characterized by a vanishing growth rate at $k=0$, a positive $k^2$ contribution from the driving force and a negative $k^4$ from capillary damping. Here we generalize to a moving interface coupling thin and thick film regimes. The resulting 4th order, nonlinear integro-differential equation contains the usual form of the capillary term but a nonlocal thermocapillary term due to far field contributions from the lateral transport of conserved quantities. The dispersion equation in no longer of Type II since the destabilizing term is no longer quadratic. Despite these differences, the generalized form retains certain pleasing features which can be exploited for further analysis.

1:16PM R16.00003 Self-similar rupture of thin free films of power law fluids, SUMEET THETHE, CHRISTOPHER ANTHONY, OSMAN BASARAN, School of Chemical Engineering, Purdue University, West Lafayette, IN 47906, PANKAJ DOSHI, Chemical Engineering and Process Development, National Chemical Laboratory, Pune, India — Rupture of a thin sheet (free film) of a power law fluid under the competing influences of destabilizing van der Waals pressure (vdWP) and stabilizing surface tension pressure (STP) is analyzed. In such a fluid, viscosity is not constant but decreases with the deformation rate raised to the $n-1$ power where $0<n<1$ is the power law exponent. For a Newtonian fluid, it is shown that when $n>6/7$, film rupture occurs under a balance between vdWP, inertial stress (IS), and viscous stress (VS), and the film thickness decreases as $r^{-n/3}$ and the lateral length scale as $r^{1-n/2}$ where $r$ is time remaining to rupture. When $n<6/7$, the dominant balance changes so that VS becomes negligible and the film ruptures under the competition between vdWP, IS, and STP. In this new regime, film thickness and lateral length vary as $r^{-2/7}$ and $r^{-4/7}$.

1:29PM R16.00004 Competing disturbance amplification mechanisms in two-fluid boundary layers, SANDEEP SAHA, IIT Kharagpur, JACOB PAGE, Imperial College London, TAMER ZAKI, John Hopkins University — The linear stability of boundary layers above a thin free film of lower viscosity is analyzed. Appropriate choice of the film thickness and viscosity excludes the possibility of interfacial instabilities. Transient amplification of disturbances is therefore the relevant destabilizing influence, and can take place via three different mechanisms in the two-fluid configuration. Each is examined in detail by solving an initial value problem whose initial condition comprises a pair of appropriately chosen eigenmodes from the discrete, continuous and interface modes. Two regimes are driven by the lift-up mechanism: (i) The response to a streamwise vortex and (ii) the normal vorticity generated by a stable Tollmien-Schlichting wave. Both are damped due to the film. The third regime is associated with the wall-normal vorticity that is generated by the interface displacement. It can lead to appreciable streamwise velocity disturbances in the near-wall region at relatively low viscosity ratios. The results demonstrate that a wall film can stabilize the early linear stages of boundary-layer transition, and explain the observations from the recent nonlinear direct numerical simulations of this configuration by Jung & Zaki (J. Fluid Mech., vol 772, 2015, 330-360).

1:42PM R16.00005 Instability of floating extensional flows, ROIY SAYAG, Ben-Gurion University, Dept. of Environmental Physics, GRAE WORSTER, University of Cambridge, DAMTP — We study the propagation of a viscous fluid over a thin layer of a denser and inviscid fluid. The viscous fluid is released axi-symmetrically at constant flux, and is driven by gravity. Near the origin, where the viscous layer is thick, the flow is dominated by vertical shear. In the outer region where the viscous layer is thinner, it floats over the inviscid layer and the dominant stress is extensional. The floating region of such flows remains axisymmetric when the viscous fluid is Newtonian. In contrast, when the viscous fluid is non-Newtonian, the floating region can be distributed in an array of extensional tongues. We use experimental and theoretical analysis to study the symmetry breaking of the extensional region. Experiments using polymeric fluids show that the characteristic wavelength of the tongues increases with flux. Theoretically, we model the symmetry breaking as flow instability of a power-law fluid that becomes Newtonian at low strain rates. Our model predicts unstable modes at the strongly non-Newtonian limit, and stable, axisymmetric mode in the Newtonian limit.

1:55PM R16.00006 Flow patterns in free liquid film caused by thermocapillary effect, ICHIRO UENO, LINHAO FEI, YOSUKE KOWATA, TOSHIIRO KANEKO, Tokyo University of Science, DONALD PETTIT, NASA — The basic flow patterns realized in a thin free liquid film driven by the thermocapillary effect are focused. Spatial attention is paid to the effect of the volume ratio of the liquid film to the hole sustaining the film on the flow patterns. We prepare a thin liquid film of less than 0.5 mm in thickness in order to stably realize the film under normal gravity. Liquid has in general negative temperature coefficient of its surface tension; that is, the fluid is driven to the colder to hotter regions by the non-uniform surface-tension distribution. In the case of thin free liquid film, however, it is found that a unique flow pattern is induced. One of the present authors, DRP, carried out a series of experiments under microgravity condition in the International Space Station (ISS) in 2003. He prepared a ring made of metal, and formed a thin film of water inside the ring. Once he added a non-uniform temperature distribution to the film by placing a heated iron at one end of the ring, a net flow toward the heated iron was realized. In order to understand flow patterns, we focus on the flow structures of the thermocapillary convection in a cross section normal to the end walls as well as the surface temperature distributions.
2:08PM R16.00007 A Combined Lagrangian-Thin Film Model for Investigating Film to Rivulet Transition on Surfaces with Various Wettabilities, MOUSSA TEMBELEY, IAN DIZON, ALI DOLATABADI, Concordia University — Understanding rivulet formation which arises from cloud droplets impact is of interest for the aerospace industry, where in icing conditions rivulets and droplets run back along the aerodynamics surfaces. In the present work, a numerical model based on a coupling between a Lagrangian method for spray generation and thin film approach is used to investigate the rivulet formation on various surfaces. The thin-film approximation which results from the simplification of the Navier-Stokes equations accounts for the surface wettability through a contact line force model which enables to describe film-to-rivulet transitions and film separation. After validating the thin film model with a Nusselt solution of a steady state laminar flow over a vertical plate, the transition from spray to rivulet is simulated on a cylinder with 3 wettabilities: hydrophilic, hydrophobic and superhydrophobic. The spray impingement on the cylinder is carried out in 2 configurations (i) vertical where a gravity-driven rivulet is formed on the cylinder side, and (ii) horizontal where the spray impacts on the cylinder under the effect of airflow. In addition to the simulations, these two configurations are investigated experimentally using a high speed camera and a small scale icing wind tunnel.

2:21PM R16.00008 Breakup of partially wetting nanoscale nematic liquid films1, MICHAEL LAM, New Jersey Institute of Technology, LINDA CUMMINGS COLLABORATION, LOU KONDIS COLLABORATION, TE-SHENG LIN COLLABORATION2 — The breakup of nematic liquid crystals (NLCs) films with thicknesses less than a micrometer is studied. Particular attention is paid to the interplay between the bulk elasticity and the anchoring (boundary) conditions at the substrate and free surface. Within the framework of the long wave approximation, a fourth order nonlinear partial differential equation (PDE) is derived for the free surface height. Numerical simulations of a perturbed flat film show that, depending on the initial average thickness of the film, satellite droplets form and persist on time scales much longer than dewetting. Formulating the model in terms of an effective disjoining pressure (elastic response and van der Waals interaction), simulations further suggest that satellite droplets form when the initial average film thickness corresponds to a positive effective disjoining pressure. Our results may shed light on the so-called “forbidden film thicknesses” seen in experiments.

1Supported by NSF grant DMS-1211713 2Affiliation: National Chiao Tung University, Taiwan

2:34PM R16.00009 Surface tension gradient enhanced thin film flow for particle deposition1, JAMES GILCHRIST, KEDAR JOSHI, TANYAKORN MUANGNAPOH, MICHAEL STEVER, Lehigh University — We investigate the effect of varying concentration in binary mixtures of water and ethanol as the suspending medium for micron-scale silica particles on convective deposition. By pulling a suspension along a substrate, a thin film is created that results in enhanced evaporation of the solvent and capillary forces that order particles trapped in the thin film. In pure water or pure ethanol, assembly and deposition is easily understood by a simply flux balance first developed by Dimitrov and Nagayama in 1996. In solvent mixtures having only a few percent of ethanol, Marangoni stresses from the concentration gradient set by unbalanced solvent evaporation dominates the thin film flow. The thin film profile is similar to that found in “tears of wine” where the particles are deposited in the thin film between the tears and the reservoir. A simple model describes the 10x increase of deposition speed found in forming well-ordered monolayers of particles. At higher ethanol concentrations, lateral instabilities also generated by Marangoni stresses cause nonuniform deposition in the form of complex streaks that mirror sediment deposits in larger scale flows.

1We acknowledge funding from the NSF Scalable Nanomanufacturing Program under grant No. 1120399

2:47PM R16.00010 Two layer flow between corrugated electrodes, ELIZAVETA DUBROVIN, RICHARD V. CRASTER, DEMETRIOS T. PAPAGEORGIOU, Imperial College London — In recent years there has been growing interest in the miniaturisation of electronic tools and much research has gone into finding appropriate techniques for patterning at small scales. One such technique exploits the electrohydrodynamic instabilities of a system to induce ordered structures. In this talk we present a model of the evolution of the interface between two perfect dielectric fluids flowing between two electrodes one of which is corrugated. With the help of a Floquet stability analysis and of full time dependant numerical simulations, we will show how the amplitude and the shape of the topography influence interfacial patterns.

3:00PM R16.00011 Laminar film flow over a three-dimensional thin film, THOMAS WARD, Department of Aerospace Engineering, Iowa State University — Two-dimensional laminar film flow over a three-dimensional thin liquid film will be investigated through mostly computational analysis. The boundary layer thin film evolution equations have been developed using the Blasius boundary layer solution for the external phase with lubrication analysis for the thin liquid film. The capillary length and the dynamic pressure are used as the length and pressure scale to non-dimensionalize the equations. The resulting dimensionless equations depend on the Reynolds, Re, and Weber, We, numbers and the two fluids viscosity ratio, λ. The dimensionless boundary layer thin film equation is then solved using a 4th order Runge-Kutta-Merson method. Sinusoidal perturbations of varying amplitude and wavelength in the initial thickness of the thin film are considered. Above certain values of the initial perturbation and parameters Re, We, and λ the thin film’s deformation is enhanced.

Tuesday, November 24, 2015 12:50PM - 3:00PM — Session R17 Flow Instability: Transition to Turbulence 205 - Nigel Goldenfeld, University of Illinois, Urbana-Champaign

12:50PM R17.00001 Predator-prey effective model for the laminar-turbulent transition in a pipe1, HONG-YAN SIH, TSUNG-LIN HSIEH, NIGEL GOLDENFELD, Loomis Laboratory of Physics, University of Illinois at Urbana-Champaign — The goal of our work is to understand the phenomenology of the laminar-turbulent transition in terms of standard phase transition concepts, and to calculate the universality class from first principles. Direct numerical simulations (DNS) of transitional pipe flow show that a collective mode — a zonal flow — is activated by Reynolds stress and suppresses turbulence subsequently, leading to stochastic predator-prey-like oscillations. Here we describe in detail the effective stochastic theory for such spatial-extended predator-prey modes. We present Monte Carlo simulations of the effective theory, showing that it reproduces the phenomenology of pipe flow experiments, including the phase diagram of puff decay and splitting. In particular, the theory predicts a super-exponential lifetime statistics for both puff decay and puff-splitting, in agreement with experimental data on pipe flow, and can be mapped exactly to the field theory of directed percolation. Our calculations strongly suggest that transitional turbulence in pipes is in the universality class of directed percolation.

1This work was partially supported by the National Science Foundation through grant NSF-DMR-1044901.
1:03PM R17.00002 Linear stability analysis of flows in a grooved channel. ALIREZA MOHAMMADI, Princeton University, JERZY MACIEJ FLORYAN, University of Western Ontario — It is known that longitudinal grooves which are parallel to the flow direction may either stabilize or destabilize the travelling wave instability in a pressure-gradient-driven channel flow depending on the groove wave number. These waves reduce to the classical Tollmien-Schlichting (TS) waves in the smooth channel limit. It is shown that another class of travelling wave instability exists if grooves with sufficiently high amplitude and proper wavelengths are used. It is demonstrated that the new instability is driven by inviscid mechanisms, with the disturbance motion having the form of a wave propagating in the streamwise direction with the phase speed approximately four times larger than the TS wave speed and with its streamwise wavelength being approximately twice the spanwise groove wavelength. The instability motion is concentrated mostly in the middle of the channel and has a primarily planar character, i.e. the dominant velocity components are parallel to the walls. A significant reduction of the corresponding critical Reynolds number can be achieved by increasing the groove amplitude. This mode reduces to the highly attenuated Squire mode in the smooth channel limit.

1This work has been carried out with support from the Natural Sciences and Engineering Research Council (NSERC) of Canada.

1:16PM R17.00003 Transition to turbulence in pipe flow as a phase transition. MUKUNDVASUDEVA, BJÖRN HOF, Institute of Science and Technology Austria — In pipe flow, turbulence first arises in the form of localized turbulent patches called puffs. The flow undergoes a transition to sustained turbulence via spatio-temporal intermittency, with puffs splitting, decaying and merging in the background laminar flow. However, the due to mean advection of the puffs and the long timescales involved (∼ 10⁵ advective time units), it is not possible to study the transition in typical laboratory set-ups. So far, it has only been possible to indirectly estimate the critical point for the transition. Here, we exploit the stochastic memoryless nature of the puff decay and splitting processes to construct a pipe flow set-up, that is periodic in a statistical sense. It then becomes possible to study the flow for sufficiently long times and characterize the transition in detail. We present measurements of the turbulent fraction as a function of Reynolds number which in turn allows a direct estimate of the critical point. We present evidence that the transition has features of a phase transition of second order.

1:29PM R17.00004 Turbulent structures in Kolmogorovian shear flows: DNS. LAURETTE S TUCKERMAN, MATTHEW CHANTRY, PMMH (UMR 7636 CNRS - ESPCI - UPMC Paris 6 - UPD Paris 7 - PSL), DWIGHT BARKLEY, University of Warwick — Patterns of turbulent and laminar flow form a vital step in the transition to turbulent in wall-bound shear flows. In flows with two unconstrained directions these patterns form oblique bands, whereas in pipe flow the structures are streamwise-localized puffs. To understand these structures we examine Waleffe flow, a sinusoidal shear flow, \( U \sin \left( \frac{\pi y}{y_*} \right) \), driven by a body force and stress-free boundary conditions at \( y = \pm 1 \). Introduced as a model for plane Couette flow we demonstrate the existence of turbulence bands which match those found in plane Couette flow, excluding the boundary layer regions of the latter flow. This agreement is reiterated in the studies of uniform flows. In flows with two unconstrained directions these patterns form oblique bands, whereas in pipe flow the structures are streamwise-localized puffs. To understand these structures we examine Waleffe flow, a sinusoidal shear flow, \( U \sin \left( \frac{\pi y}{y_*} \right) \), driven by a body force and stress-free boundary conditions at \( y = \pm 1 \). After establishing the ability of Waleffe flow to capture turbulent bands we study a series of models, capturing the shear dependent direction with a small number of Fourier modes. With only one nonzero Fourier wavenumber the fundamentals of bands are already observed. This minimal system offers the perfect testbed to study the emergence of bands. Considering small increases to the number of modes we find the rich behaviour associated with plane Couette flow. These models form a fascinating midpoint between the full Navier-Stokes equations and the minimal SSP model.

1:42PM R17.00005 Turbulent structures in Kolmogorovian shear flows: Models. MATTHEW CHANTRY, LAURETTE S TUCKERMAN, PMMH (UMR 7636 CNRS - ESPCI - UPMC Paris 6 - UPD Paris 7 - PSL), DWIGHT BARKLEY, University of Warwick — Oblique patterns of turbulence are observed immediately beyond transition in wall-bound shear flows with two unconstrained directions. Despite the ubiquitous nature of these structures, simple descriptions obtained directly from the Navier-Stokes equations are lacking. To this aim we examine Waleffe flow, a sinusoidal shear flow, \( U \sin \left( \frac{\pi y}{y_*} \right) \), driven by a body force and stress-free boundary conditions at \( y = \pm 1 \). After establishing the ability of Waleffe flow to capture turbulent bands we study a series of models, capturing the shear dependent direction with a small number of Fourier modes. With only one nonzero Fourier wavenumber the fundamentals of bands are already observed. This minimal system offers the perfect testbed to study the emergence of bands. Considering small increases to the number of modes we find the rich behaviour associated with plane Couette flow. These models form a fascinating midpoint between the full Navier-Stokes equations and the minimal SSP model.

1:55PM R17.00006 Determining the universality class of the transition to turbulence. GRÉGOIRE LEMOULT, BJÖRN HOF, IST Austria — In the counter-rotating regime of Taylor-Couette (TC) flow, turbulence appears abruptly through spatio-temporal intermittency (STI). STI is observed during the sub-critical transition to turbulence in many wall bounded shear flows, most notably pipe flow and Couette flows. K. Avila et al. (Science 333, 192-196 (2011)) recently characterized the onset of sustained turbulence in pipe flow and suggested that the transition could be a second order non-equilibrium phase transition. We explore this proposition in a TC experiment. Our TC set-up has an aspect ratio of 8 and an azimuthal length of more than 1300 gap-widths, minimizing the finite size effects and keeping the spatio-temporal dynamics 1D. We measured three independent critical exponents and they are in very good agreement with those of the directed percolation universality class in 1+1 dimensions. These experimental results strongly suggest that the transition to turbulence is a second order non-equilibrium phase transition.

2:08PM R17.00007 The behavior of droplet-laden pipe flow at the onset of turbulence. KYLE WINTERS, ELLEN LONGMIRE, University of Minnesota — The addition of either dispersed fluid droplets or solid particles to a pipe flow can modify the Reynolds number at which the flow transitions to turbulence, \( Re_c \). For dispersed solid particles, Matas et al. (2003) studied the behavior of \( Re_c \) as a function of volume fraction and particle size, and found that for certain regimes particles can promote transition, while for others, transition was delayed to higher values of \( Re_c \). To explore the phenomenon in droplet-laden flows, pressure and PIV measurements are taken in facility comprised of a pump-driven circuit with a 44mm diameter, D, and with an 8.8m (200D) development and test section. Static mixers are placed upstream to generate an even dispersion of silicone oil in a refractive index matched water-glycerin flow. Pressure signals were used to identify transitional structures and trigger a high repetition rate PIV system downstream. Information from the pressure drop traces is used to determine \( Re_c \) for various droplet sizes and volume fractions. Additionally, PIV data provide detailed information about velocity variations and the transitional structures in the flow. Pressure and PIV data from droplet laden flow are compared to similar data from single phase flows in our facility and in the literature.

1partially funded by Nano Dispersion Technologies
2:21PM R17.00008 New experiment in Plane Poiseuille flow with zero mean advection velocity: observation of stationary turbulent spots, LUKASZ KLOTZ, PMMH/ ESPCI-CNRS, GREGOIRE LEMOULT, IST Austria, JOSE EDUARDO WESFREID, PMMH/ ESPCI-CNRS — We describe a new experimental set-up which allows us to study the sub-critical transition to turbulence in a two dimensional shear flow (including plane Couette, plane Couette-Poiseuille and plane Poiseuille flows). Our facility is an extension of a classical plane Couette experiment, in which one uses a single closed loop of plastic belt to generate the opposite sign velocity at each wall of the test section. However, in our case, we use two independent closed loops of plastic belt, one at each wall of the test section. The speed of these belts may be controlled separately. That enables to set two different velocities (in value and direction) as a boundary conditions at each of two test section’s walls. In addition the pressure gradient in streamwise direction can be controlled. In particular, the plane Poiseuille flow with zero mean advection velocity can be created. We characterize by PIV the basic flow for different configurations. For a plane Poiseuille flows as base flow, we were able to observe for the first time the nearly stationary turbulent spots in this flow, with structures of characteristic wavelength ∼ the distance between the two plates.

2:21PM R17.00008 New experiment in Plane Poiseuille flow with zero mean advection velocity: observation of stationary turbulent spots, LUKASZ KLOTZ, PMMH/ ESPCI-CNRS, GREGOIRE LEMOULT, IST Austria, JOSE EDUARDO WESFREID, PMMH/ ESPCI-CNRS — We describe a new experimental set-up which allows us to study the sub-critical transition to turbulence in a two dimensional shear flow (including plane Couette, plane Couette-Poiseuille and plane Poiseuille flows). Our facility is an extension of a classical plane Couette experiment, in which one uses a single closed loop of plastic belt to generate the opposite sign velocity at each wall of the test section. However, in our case, we use two independent closed loops of plastic belt, one at each wall of the test section. The speed of these belts may be controlled separately. That enables to set two different velocities (in value and direction) as a boundary conditions at each of two test section’s walls. In addition the pressure gradient in streamwise direction can be controlled. In particular, the plane Poiseuille flow with zero mean advection velocity can be created. We characterize by PIV the basic flow for different configurations. For a plane Poiseuille flows as base flow, we were able to observe for the first time the nearly stationary turbulent spots in this flow, with structures of characteristic wavelength ∼ the distance between the two plates.

2:47PM R17.00010 Transition to turbulence in pulsatile pipe flow, BJORN HOF, IST Austria, DUO XU, University of Erlangen — Pulsating flows are common in nature and applications, the most prominent example being cardiovascular flow. Often such flows are at the verge of becoming turbulent yet the influence of pulsation on the transition process is unclear. We present detailed experiments carried out in a straight pipe of circular cross-section with a sinusoidally modulated flow rate. With decreasing frequencies (Wo < 10) the transition is delayed considerably to larger mean Reynolds numbers, however the qualitative transition scenario remains unchanged. Like for steady flows puffs are the first observable turbulent structures but their lifetimes are shorter and turbulence only becomes sustained at larger Re. For fixed frequency and increasing pulsation amplitude on the other hand a different transition process is found. Here turbulence only occurs during the decelerating flow phase. Like the steady flow transition also this transition appears to require finite amplitude perturbations. The structure of the resulting turbulent flow however differs considerably for the pulsatile case. We map out the stability threshold for both transition types co-exist and compete.

Tuesday, November 24, 2015 12:50PM - 3:26PM — Session R18 Boundary Layers: Superhydrophobic Surfaces 206 - Julie Crockett, Bringham Young University

12:50PM R18.00001 Traces of surfactants limit the drag reduction potential of superhydrophobic surfaces in realistic applications, FRANCOIS J. PEAUDECERF, JULIEN R. LANDEL, University of Cambridge, PAOLO LUZZATTO-FEGIZ, UC Santa Barbara — Large drag reductions have been measured for laminar flows over superhydrophobic surfaces (SHS), making them attractive for applications in pipelines, ships and submarines. However, experiments involving turbulent flows, typical of these applications, have often yielded limited drag reductions. A complete explanation for this issue has so far proved elusive. We propose that trace amounts of surfactants, unavoidable in the environment and in large-scale experiments, can yield poor performances of SHS, by producing Marangoni stresses when the edges of the SHS pattern are not aligned with the local flow velocity. To explore our hypothesis, we develop numerical simulations (inclusive of surfactants) for a flow over a textured SHS in a micro-channel, whose background shear is similar to a viscous sublayer. The texture consists of micro ridges perpendicular to the flow. We find that even small amounts of surfactants can prevent any drag reduction. As an experimental test, we flow de-ionized water with known surfactant concentrations through SHS micro-channels with texture similar to the simulations, while performing micro-PIV. At negligible surfactant concentrations, we find higher velocities between the ridges, as expected by classical models. However, as the concentration increases, we discover that the slip velocity drops to very small values even in the presence of a plastron. Our results show that the drag-reducing potential of superhydrophobic surfaces can be limited in realistic flow conditions.

1:03PM R18.00002 Effect of Interface Curvature on Super-Hydrophobic Drag Reduction, AMIRREZA RASTEGARI, RAYHANEH AKHAVAN, The University of Michigan, Ann Arbor — The effect of interface curvature on Super-Hydrophobic (SH) Drag Reduction (DR) has been investigated using DNS with lattice Boltzmann methods in laminar (Re_{bulk} = 50) and turbulent (Re_{bulk} = 3600, Re_{eq} ≈ 223) channel flows. SH surfaces with longitudinal arrays of micro-grooves (MG) of size 0.1 ≤ g/h ≤ 0.47 & g/w ≥ 1, 7 were investigated, where g and w denote the width of the MG and the separation in between them, respectively, and h denotes the channel half-height. The liquid/gas interfaces on the SH MG were modeled as ‘idealized’, stationary, curved, shear-free boundaries, with the interface curvatures determined from the Young-Laplace equation. The presence of interface curvature leads to enhancements of DR by up to 10% in laminar flow, and more modest enhancements or even decreases in DR in turbulent flow, compared to flat, shear-free interfaces. These enhancements or decreases in DR, relative to flat, shear-free interfaces, in both laminar and turbulent flow, are shown to arise primarily from the modified shape of the cross section of the channel in the presence of the curved interface.

1:16PM R18.00003 Skin-Friction Drag Reduction over Super-Hydrophobic Materials in Fully-Developed Turbulent Flow, JAMES W. GOSE, KEVIN GOLOVIN, STEVEN L. CECcio, MARC Perlin, ANISH TUTEJA, Univ of Michigan - Ann Arbor — As part an on-going research initiative to develop super-hydrophobic (SH) materials for high-speed naval applications, a team at the University of Michigan investigated SH materials for drag reduction in fully-developed turbulent flow. The SH materials were evaluated in a high-aspect ratio (width/height) channel flow facility capable of producing average flow speeds of 20 m/s, yielding a height (7 mm) based Reynolds number of 140,000. The SH materials examined were developed for large-scale application using various technologies including spraying, chemical etching, and mechanical abrasion. The materials were applied over a 100 mm (spanwise/width) area. The drag measurement methods were pressure drop along the test surface over length 150H (1050 mm) and by means of the velocity profile via particle image velocimetry. The SH materials were investigated further to determine the effects of various flow conditions including low (vacuum) and high pressures. The drag reduction measurements were coupled with extensive topological evaluation of the materials to illustrate the importance of each aspect of the individual SH features, as well as the collective structure of the surface, leading to insight regarding the relevant characteristics of an SH material’s ability to reduce skin-friction in fully-developed turbulent flow.

1The authors recognize the support of ONR.
1:29PM R18.00004 Effects of the pitch length of superhydrophobic surfaces on the effective slip length and skin-friction drag1, TAEYONG JUNG, HAECHEON CHOI, Seoul National University, JOHN KIM, University of California, Los Angeles — Many numerical studies have been conducted to investigate the effect of the grating parameters of superhydrophobic surfaces, such as the pitch length and gas fraction, on the slip velocity and its effect on skin-friction drag. However, the pitch lengths considered numerically so far are much larger, varying from $p^+ = O(10)$ to $O(10^2)$ in wall units, than those in experiments ($p^+ = O(1)$). In the present study, we perform a direct numerical simulation of turbulent channel flow over superhydrophobic surfaces with longitudinal microgrates having the actual grating parameters of $p^+ = 3.8$. The air layer inside the cavity ($d^+ = 18; d^+$ is the cavity depth) is also considered with the assumption of zero interface curvature. The minimal flow unit by Jimenez & Moin (1991) is adopted to resolve the small pitch length. Since the pitch length is composed of small cavity width, the growth of the slip velocity at the air-water interface is inhibited. As a result, the slip velocity ($u_{sl}$) is less than 2 for $p^+ = 3.8$, whereas $u_{sl}^+$ is greater than 15 for $p^+ = 540$. The effective slip length is an order of the viscous sublayer thickness, and the drag reduction is less than 20%. The detailed results for the cases of $p^+ \sim O(1)$ to $O(10^2)$ will be presented.

1Supported by NRF-2012M2A8A4055647

1:42PM R18.00005 Pool boiling thermal transport through micro-patterned metal superhydrophobic surfaces1, MATTHEW SEARLE, DANIEL MAYNES, JULIE CROCKETT, Brigham Young University - Provo — Pool boiling thermal transport through horizontal superhydrophobic surfaces decorated with rib and post micro-patterns was explored experimentally. The pool consisted of a water reservoir heated from below by electric heaters embedded in an aluminum block. A test surface was located at the bottom of the pool and fixed to the block. Instrumentation allowed simultaneous measurements of heat flux through the test surface, test surface temperature, and pool watre temperature. From these measurements, heat flux as a function of excess temperature (the difference between the test surface temperature and the water saturation temperature) was determined for each surface. Surface geometry was characterized by the cavity fraction (the ratio of projected cavity area to surface area on the test surface), distance between features, and microscale pattern geometry. The transition from nucleate to pool boiling was observed to occur at much lower excess temperatures for superhydrophobic surfaces, with greater deviation for larger cavity fraction. Heat flux versus excess temperature relationships are presented while exploring the influence of superhydrophobic surface microstructure on the thermal transport.

1 NSF CBET-1235881

1:55PM R18.00006 Direct Numerical Simulation of turbulent flows over superhydrophobic surfaces: capillary waves on gas-liquid interface1, JONGMIN SEO, Stanford University, RICARDO GARCIA-MAYORAL, University of Cambridge, ALI MANI, Stanford University — Superhydrophobic surfaces under liquid flow can produce significant slip, and thus drag reduction, when they entrap gas bubbles within their roughness elements. Our work aims to explore the onset mechanism to the failure of drag reduction by superhydrophobic surfaces when they are exposed to turbulent boundary layers. We focus on the effect of finite surface tension to the dynamic response of deformable interfaces between overlying water flow and the gas pockets. To this end, we conduct direct numerical simulations of turbulent flows over superhydrophobic surfaces allowing deformable gas-liquid interface. DNS results show that spanwise-coherent, upstream-traveling waves develop on the gas-liquid interface as a result of its interactions with turbulence. We study the nature and scaling of the upstream-traveling waves through semi-analytical modeling. We will show that the traveling waves are well described by a Weber number based on the slip velocity at the interface. In higher Weber number, the stability of gas pocket decreases as the amplitude of interface deformation and the magnitude of pressure fluctuations are augmented.

1Supported by Office of Naval Research and the Kwanjeong Educational Scholarship Foundation

2:08PM R18.00007 Velocity and Reynolds Stress Profiles in The Inner Part of a Turbulent Boundary Layer over Super-Hydrophobic Surfaces1, HANGJIAN LING, JOSEPH KATZ, Johns Hopkins University, SIDDARTH SRINIVASAN, GARETH MCKINLEY, Massachusetts Institute of Technology — Digital holographic microscopy is used to perform high-resolution velocity and Reynolds stress measurements in the inner parts of turbulent boundary layers over super-hydrophobic surfaces (SHS) and compare them to those of smooth walls. The SHSs are created by spray-coating perfluorodecyl polyhedral oligomeric silsesquioxane (POSS) dispersed in a poly (methyl methacrylate) binder onto a porous base which facilitates replenishment of air under a controlled pressure difference ($\Delta p$). The measurements are performed at friction Reynolds numbers of 1400-4300, surface roughness of $k_r = 10-20 \mu m$ ($k_r^+ =1-3$), and $\Delta p < 0$ or $> 0$. The wall stress $\tau_w$ is calculated from the velocity gradients in the viscous sublayer and total shear stress at the top of “roughness” elements. Results reveal that compared to a smooth wall, the SHS $\tau_w$ is reduced by $\sim 10\%$ for $k_r^+ < 1$, but increases for $k_r^+ > 2$ when roughness overcomes super-hydrophobicity. Accordingly, the log-layer shifts upward when $\tau_w$ is reduced, and downward when $\tau_w$ increases. For a SHS-dominated inner flow, the Reynolds stresses remain similar to that of the smooth wall. The measured relationship between slip length and reduction in wall viscous stress agrees with theoretical predictions involving both streamwise and spanwise slips.

1Sponsored by ONR

2:21PM R18.00008 Lift and Drag Measurements of Superhydrophobic Hydrofoils1, SAMRAT SUR, JEONG-HYUN KIM, JONATHAN ROTHSTEIN, University of Massachusetts - Amherst — For several years, superhydrophobic surfaces which are chemically hydrophobic with micron or nanometer scale surface features have been considered for their ability to reduce drag and produce slip in microfluidic devices. More recently it has been demonstrated that superhydrophobic surfaces reduce friction coefficient in turbulent flows as well. In this talk, we will consider that modifying a hydrofoil’s surface to make it superhydrophobic has on the resulting lift and drag measurements over a wide range of angles of attack. Experiments are conducted over the range of Reynolds numbers between $10,000<Re<50,000$. The effect of superhydrophobicity on separation point and vortex structure will be studied using Particle Image Velocimetry (PIV) and streak images. We will show that changes to the drag and lift coefficients along with changes to separation point at high angles of attack are observed when the hydrofoil is made superhydrophobic. The hydrofoils are coated Teflon that has been hot embossed with a 325 grit stainless steel woven mesh to produce a regular pattern of microposts. In addition to fully superhydrophobic hydrofoils, selectively coated symmetrical hydrofoils will also be examined to study the effect that asymmetries in the surface properties can have on lift and drag.

1Partially funded by NSF CBET-1334962
2:34PM R18.00009 Turbulent boundary layer over a convergent and divergent superhydrophobic surface. MUHAMMAD NADEEM, JINYUL HWANG, HYUNG JIN SUNG, KAIST — Direct numerical simulation (DNS) of spatially developing turbulent boundary layer (TBL) over a convergent and divergent superhydrophobic surface (SHS) was performed. The convergent and divergent SHS was aligned in the streamwise direction. The SHS was modeled as a pattern of slip and no-slip surfaces. For comparison, DNS of TBL over a straight SHS was also carried out. The momentum thickness Reynolds number was varied from 800 to 1400. The gas fraction of the convergent and divergent SHS was the same as that of the straight SHS, keeping the slip area constant. The slip velocity in the convergent SHS was higher than that of the straight SHS. An optimal streamwise length of the convergent and divergent SHS was obtained. The convergent and divergent SHS led to a modified effect of near-wall turbulent structures, resembling the narrowing and widening streaky structures near the wall. The convergent and divergent SHS had a relatively larger damping effect on near-wall turbulence than the straight SHS. These observations will be further analyzed statistically to demonstrate the effect of the convergent and divergent SHS on the interaction of inner and outer regions of TBL.

2:47PM R18.00010 Superhydrophobic surfaces in turbulent channel flow1. YIXUAN LI, KRISHNAN MAHESH, University of Minnesota — We discuss results from a direct numerical simulation which resolves the features of superhydrophobic surfaces in turbulent channel flow at $Re = 400$ to study the effect of feature geometry. The height of the grooves $h^+$ is 3.6, which is smaller than most previous numerical studies. A channel with only one groove on the bottom wall is first modeled to study the local effect of the groove geometry. Then an SHS with a groove coverage ratio $\phi = 87.5\%$ is created as the bottom wall of a turbulent channel flow of $Re = 400$. The effect of the grooves is quantified locally as well as over the entire channel wall. Results for slip velocity, turbulence intensity and spectra will be discussed. The influence of the grooves on the overall mean momentum budget will also be discussed.

1Office of Naval Research

3:00PM R18.00011 Direct Numerical Simulation of Superhydrophobic Surfaces1. KARIM ALAME, KRISHNAN MAHESH, University of Minnesota-Twin Cities — A volume of fluid methodology will be used to study the physics of superhydrophobic surfaces. The geometry of the surface will be resolved. The effect of pressure difference on the interface will be presented and contrasted to theory. Interface failure will be explored and simulations of microchannel flow will be compared to experiments. A turbulent channel with superhydrophobic grooves will be presented showing the interface behavior and implications on drag reduction. Extension to random textured surfaces will be discussed.

1This work is supported by the Office of Naval Research.

3:13PM R18.00012 Enhanced convective transport from an isothermal circular cylinder with hydrodynamic slip boundary condition. NIDHIL MOHAMED ABDUL REHMAN, RATNESH SHUKLA, Indian Institute of Science, Bangalore — Introduction of a slip in the tangential surface velocity suppresses vorticity production in a typical bluff body flow while simultaneously enhancing vorticity convection downstream and into the wake region. As a result the flow characteristics are altered significantly and the Reynolds number can be considerably increased. In this work we investigate the effect of the hydrodynamic slip on the convective heat transfer from the surface of a heated isothermal circular cylinder placed in the uniform cross flow of a viscous incompressible fluid through numerical simulations. We find that for fixed Reynolds and Prandtl numbers an increase in the Nusselt number or equivalently the hydrodynamic slip length results in a substantial augmentation of the heat transfer coefficient. We establish the dependence of the Nusselt number on the Knudsen, Reynolds and Prandtl numbers over a wide range of these parameters. We find that for given Reynolds and Prandtl numbers the Nusselt number undergoes a sharp transition between the low and high asymptotic limits that correspond to zero (no-slip) and infinite (shear-free perfect slip) Knudsen numbers. We establish that the high asymptotic limit corresponding to the shear-free perfect slip cylinder boundary scales as $Nu \sim Re^{0.5} Pr^{0.5}$.

Tuesday, November 24, 2015 12:50PM - 3:26PM – Session R19 Turbulence: Theory II

12:50PM R19.00001 A Generalized Brownian Motion Model for Turbulent Relative Particle Dispersion1, BHIMSEN SHIVAMOGGI, University of Central Florida — A generalized Brownian motion model has been applied to the turbulent relative particle dispersion problem (Shivamoggi [1]). The fluctuating pressure forces acting on a fluid particle are taken to follow an Uhlenbeck-Ornstein process while it appears plausible to take their correlation time to have a power-law dependence on the flow Reynolds number $Re$. This ansatz provides an insight into the result that the Batchelor-Townsend scaling holds only in the infinite-$Re$ limit and disappears otherwise. It provides a determination of the Richardson-Obukhov constant $g$ as a function of $Re$, with an asymptotic constant value in the infinite-$Re$ limit. This ansatz is further shown to be in quantitative agreement, in the small-$Re$ limit, with the Batchelor-Townsend ansatz for the rate of change of the mean square interparticle separation in 3D FDT. [1] B.K. Shivamoggi: arXiv: 1208.5786 (2014).

1My thanks to The Netherlands Organization for Scientific Research for Support.

1:03PM R19.00002 Turbulent Damping without Eddy Viscosity, SIMON THALABARD, Umass Amherst — The intrinsic Non-Gaussianity of turbulence may explain why the standard Quasi-Normal cumulant discard closures can fail dramatically, an example being the development of negative energy spectra in Millionschichkov’s 1941 Quasi-Normal (QN) theory. While Orszag’s 1977 EDQNM provides an ingenious patch to the issue, the reason why QN fails so badly is not so clear. Is it because of the Gaussian Ansatz itself? Or rather its inconsistent use? The purpose of the talk is to argue in favor of the latter option, using the lights of a new “optimal closure” recently exposed by [Turkington,2013], which allows Gaussians to be used consistently with an intrinsic damping. The key to this apparent paradox lies in a clear distinction between the ensemble averages and their proxies, most easily grasped provided one uses the Liouville equation rather than the cumulant hierarchy as a starting point. Schematically said, closure is achieved by minimizing a lack-of-fit residual, that retains the intrinsic features of the dynamics. For the sake of clarity, I will discuss the optimal closure on a problem where it can be entirely implemented and compared to DNS: the relaxation of an arbitrarily far from equilibrium energy shell towards the Gibbs equilibrium for truncated Euler dynamics.
1:16PM R19.00003 Two-point Spectral Modeling of Anisotropic Rapid Distortion

TIMOTHY CLARK, University of New Mexico, SUSAN KURIEN, Los Alamos National Laboratory, CHARLES ZEMACH, Retired — We perform simulations of a two-point spectral model for the evolution of the energy tensor as function of wave-vector, for arbitrarily anisotropic turbulence in the limit of rapid distortion. The resulting Reynolds stress tensor for such flow is analysed for the effects of anisotropy during evolution. According to the SO(3) rotation group decomposition of the energy tensor, the leading order isotropic contribution is labelled by rotational mode index \( j = 0 \), while higher order anisotropic contributions in statistically homogeneous flows contain a potentially very large array of rotational modes \( j = 2, 3, 4, \ldots \). We compare our results to those of the classical Launder, Reece and Rodi class of models in the rapid distortion limit. These models only retain anisotropy in a nominal manner up to \( j = 2 \), due to an a priori angle-averaging procedure on the energy tensor, reducing it to a function of wave-number alone. Although the Reynolds stress itself has maximum \( j = 2 \) in the SO(3) representation, the terms that contribute to its evolution generate higher order rotational modes. The contributions from the higher order modes are shown to be responsible for the deviation of the LRR solution from the true solution over time.

1:29PM R19.00004 Irreversibility and small-scale generation in 3-dimensional turbulent flows

ALAIN PUMIR, Ecole Normale Superieure de Lyon and CNRS, HAITAO XU, Tsinghua University, Beijing, China, RAINER GRAUER, Ruhr University Bochum, Germany, EBERHARD BODENSCHATZ'1, Max-Planck-Institute for Dynamics and Self-organisation, Goettingen, Germany. — In 3-dimensional turbulent flows, the irreversibility of turbulence manifests itself by an asymmetry of the probability distribution of the instantaneous power \( p \) of the forces acting on fluid elements: the third moment of \( p \) was found to be negative. Here, I will discuss the relation between this negative third moment and vortex stretching, which is traditionally related to the generation of small scales. The construction is based on a decomposition of the power \( p \) as a sum of a local contribution, due to the variation of velocity at a fixed point in space, plus a convective part, due to the displacement of particles in the flow. The third moment of the latter term, which dominates the statistics, is explicitly expressed in terms of vortex stretching.

1:42PM R19.00005 On the power law of passive scalars in turbulence

TOSHIYUKI GOTOH, TAKESHI WATANABE, Nagoya Institute of Technology — It has long been considered that the moments of the scalar increment with separation distance \( r \) obey power law with scaling exponents in the inertial convective range and the exponents are insensitive to variation of pumping of scalar fluctuations at large scales, thus the scaling exponents are universal. We examine the scaling behavior of the moments of increments of passive scalars 1 and 2 by using DNS up to the grid points of \( 10^9 \)\(^{13} \). They are simultaneously convected by the same isotropic steady turbulence at \( R_{\lambda} = 805 \), but excited by two different methods. Scalar 1 is excited by the random scalar injection which is isotropic, Gaussian and white in time at wave number band, while Scalar 2 is excited by the uniform mean scalar gradient. It is found that the local scaling exponents of the scalar 1 has a logarithmic correction, meaning that the moments of the scalar 1 do not obey simple power law. On the other hand, the moments of the scalar 2 is found to obey the well developed power law with exponents consistent with those in the literature. Physical reasons for the difference are explored.
respects the geometry of phase space, the entropy-driven cascade directionally is not precluded.

but for the strong turbulence limit, the random velocity field of the large-scale eddies is scale-dependent that leads to Kolmogorov’s energy bulence at distances to the wall larger than the position of the outer peak. This qualitative prediction is also supported by the aforementioned data.

and the finding of Dallas, Vassilicos & Hewitt (2009) that, in the intermediate layer, the eddy turnover time scales with skin friction velocity and extremely high Reynolds number data by Hultmark, Vallikivi, Bailey & Smits (2012, 2013). Townsend’s (1976) production-dissipation balance wavenumber range to the model at wavenumbers smaller than that spectrum. This necessary additional also accounts for the high Reynolds number’s variation to be more realistic while keeping with the Townsend-Perry attached eddy spectrum is to add a new eddy spectral model predicts that the integral length-scale varies very slowly with distance to the wall in the intermediate layer. The only way when reversed after being allowed to decay. During the first phase, a direct cascade transfers energy from large to small scales while, during the second, an inverse cascade does the opposite. Short-time Lyapunov (STL) analysis is used to study and compare the dynamics of both cascades. This allows us to identify a smallest length scale for the chaotic flow behavior, below which the system behaves as a unit dynamically enslaved to larger motions by the contracting effect of the model. Above it, the inertial forces become relevant and the system is fully chaotic.

When the inertial scales are isolated, the leading STL exponent is similar for both cascades, suggesting that the dynamics of the inertial range is conservative and time-symmetric, and that the direct and inverse energy cascades share similar energy transfer mechanisms. The cascade would thus be a bi-directional reversible process with similar up and down mechanisms, although, because the

cascades. When the inertial scales are isolated, the leading STL exponent is similar for both cascades, suggesting that the dynamics of the inertial range is conservative and time-symmetric, and that the direct and inverse energy cascades share similar energy transfer mechanisms. The cascade would thus be a bi-directional reversible process with similar up and down mechanisms, although, because the


cascades. When the inertial scales are isolated, the leading STL exponent is similar for both cascades, suggesting that the dynamics of the inertial range is conservative and time-symmetric, and that the direct and inverse energy cascades share similar energy transfer mechanisms. The cascade would thus be a bi-directional reversible process with similar up and down mechanisms, although, because the


1:03PM R20.00002 Modeling height-dependent spatio-temporal spectra in wallbounded turbulence, MICHAEL WILCZEK, Max Planck Institute for Dynamics and Self-Organization, RICHARD J.A.M. STEVENS, CHARLES MENEVEAU, Johns Hopkins University — Spatio-temporal spectra of wall-bounded turbulence show a non-trivial structure in the wavenumber-frequency domain. We here study spectra of streamwise velocity fluctuations by means of large-eddy simulations. Such spectra, for instance, indicate wavenumber-dependent frequency shifts induced by mean flow advection as well as frequency broadening related to large-scale velocity perturbations. In previous work, we introduced an advection model combining Taylor’s frozen eddy hypothesis and the Kraichnan–Tennekes random sweeping hypothesis to capture these observations. For the logarithmic layer of the flow, we furthermore introduced analytical model parameterizations for the height-dependent wavenumber part of the spectrum as well as the frequency shift and broadening. After summarizing these results, we will present further comparisons of the model with simulation data. In particular, we will validate the model for a range of heights across the logarithmic layer by focusing on comparisons of wavenumber-frequency spectra in the streamwise wavenumber-frequency plane as well as in the spanwise wavenumber-frequency plane.

1:16PM R20.00003 Scaling laws of turbulent Couette flow with wall-normal transpiration, STEPHANIE KRAHEBERGER, MARTIN OBERLACK, Tech Univ Darmstadt, SERGIO HOYAS, Universidad Politcnica de Valencia — An extensive DNS study of turbulent plane Couette flows with permeable boundary conditions, i.e. wall-normal transpiration, was conducted at Reτ = 250, 500, 1000 and varying transpiration velocities vT. The discretization employed is pseudo-spectral in wall-parallel and compact finite differences in wall-normal direction (see Hoyas et al., Phys. Fluids 2006). We derived a global stress relation for the flow, balancing total shear stresses, with very different friction velocities at lower and upper wall. This, in turn, was used to validate convergence of DNS statistics. Most important, we derived a viscous sublayer velocity scaling for the suction wall employing asymptotic methods. Moreover, using Lie group symmetry analysis applied to the multi-point correlation equation we derived scaling laws for the near-wall region on the blowing wall and the channel center, predicting mean velocity ⟨U⟩ and the Reynolds-stress components ⟨u_j u_i⟩, (see Oberlack et al., JSME Mech. Eng. Rew., 2015), which were nicely validated against DNS data.

1:29PM R20.00004 Properties of the total kinetic energy balance in wall-bounded turbulent flows, ANG ZHOU, University of New Hampshire, JOSEPH KLEWICKI, University of New Hampshire, University of Melbourne — The properties of the total kinetic energy balance in turbulent boundary layer and channel flows are explored empirically. The total kinetic energy transport equation, which is the combination of mean and turbulent kinetic energy transport equations, is appropriately simplified for fully developed turbulent channel flow and the two-dimensional flat plate boundary layer. Different from the turbulence kinetic energy equation, a suitable grouping of terms is found that cleanly segregates the leading balances in the total energy equation. Available high-quality data reveal a four-layer structure for the energetics that is qualitatively different from the four-layer description of the mean dynamics [Wei et al. 2005, J. Fluid Mech. 522, 303]. The wall-normal widths of the layers exhibit significant Reynolds number dependencies, and these are empirically quantified. Present findings indicate that each of the four layers is characterized by a predominance of some of the terms in the governing equations. Particular significance is attached to the ratio of the sum of viscous diffusion and dissipation terms to the production/turbulent diffusion term, since these groupings allow the characterization of the layer widths. The third layer exhibits a complex leading order balance exchange that is described in detail.

1:42PM R20.00005 High Reynolds number decay of turbulent Taylor-Couette flow, RUBEN A. VERSCHOOF, SANDER G. HUISMAN, ROELAND C.A. VAN DER VEEN, CHAO SUN, DETLEF LOHSE, Physics of Fluids - University of Twente — We study the decay of high-Reynolds number turbulence in a Taylor-Couette facility for pure inner cylinder rotation. The rotation of the inner cylinder (Re = 2 × 10^6) is suddenly decelerated as fast as possible, thus removing the energy input within seconds. Local velocity measurements show that the decay in this wall-bounded inhomogeneous flow is faster than observed for homogeneous isotropic turbulent flows, due to the strong viscous drag applied by the inner and outer cylinder surfaces. We found that the decay over time can be described with the differential equation Re'(t) = c_J(Re)Re^2, where the effects of the walls are included through the friction coefficient. A self-similar behavior of the azimuthal velocity is found: its normalized velocity profile as a function of the radius collapses over time during the decay process.

1:55PM R20.00006 Large eddy simulation study of spanwise spacing effects on secondary flows in turbulent channel flow, MOHAMMAD ALIYAKARIMIYANMAHALEH, WILLIAM ANDERSON, UT Dallas — The structure of turbulent flow over a complex topography composed of streamwise-aligned rows of cones with varying spanwise spacing, s, is studied with large-eddy simulation (LES). Similar to the experimental study of Vanderwel and Ganapathibusramani, 2015: J. Fluid Mech., we investigate the relationship between secondary flow and s, for 0.25 ≤ s/δ ≤ 5. For cases with s/δ > 2, domain-scale rollers freely exist. These had previously been called “turbulent secondary flows” (Willingham et al., 2014: Phys. Fluids; Barros and Christensen, 2014: J. Fluid Mech.; Anderson et al., 2015: J. Fluid Mech.), but closer inspection of the statistics indicates these are a turbulent tertiary flow: they only remain “anchored” to the conical roughness elements for s/δ > 2. For s/δ < 2, turbulent secondary flows are prevented from occupying the domain by virtue of proximity to adjacent, counter-rotating tertiary flows. Turbulent secondary flows are associated with the conical roughness elements. These turbulent secondary flows emanate from individual conical topographic elements and set the roughness sublayer depth. The turbulent secondary flows remain intact for large and small spacing. For s/δ < 1, a mean tertiary flow is not present.

1 Funding: DFG [WI 3544/1-1, WI 3544/3-1]; FOM Fellowships for Young Energy Scientists (YES!) and NSF (IAR 1243482, the WINDINSPIRE project)
Finite Reynolds number properties of a turbulent channel flow similarity solution. JOSEPH KLEWICKI, University of New Hampshire/University of Melbourne, MARTIN OBERLACK, Darmstadt Technical University — Finite Reynolds number behaviors of the asymptotically logarithmic mean velocity profile in fully developed turbulent channel flow are investigated. This is accomplished by exploiting invariance properties admitted by the appropriately simplified form of the mean momentum equation. These properties underlie the existence of a similarity solution over an interior inertial domain. This similarity solution, which was originally demonstrated by numerically integrating the relevant nonlinear equation, is consistent with the emergence of a logarithmic mean velocity profile as the Reynolds number becomes large. It is now shown that the governing nonlinear equation has an analytical solution that contains both linear and logarithmic terms, but with the coefficient on the linear term decaying to zero with Reynolds number. Existing DNS are used to elucidate Reynolds number dependent properties of this finite Reynolds number form of the similarity solution. Correspondences between these properties and those indicated by finite Reynolds number corrections to the classical overlap layer formulation for the mean velocity profile are described and discussed.

Support of the 2014 Mathematics of Turbulence program at the Institute for Pure and Applied Mathematics, UCLA, is gratefully acknowledged.

Transport of heat and momentum in oscillatory wall bounded flow. ALIREZA EBADI, DRUMMOND BILES, CHRISTOPHER WHITE, University of New Hampshire, IAN POND, YVES DUBIEF, University of Vermont, UNH TEAM, UVM TEAM — The balance of the leading order terms in the mean momentum and energy equations and their thrice integrated forms are investigated in oscillatory wall bounded flow using both DNS and experimental data. The integrated forms of the equations are used to investigate the dynamical contributions to the phase-averaged wall shear stress and wall heat flux. Preliminary results indicate that phases corresponding to flow acceleration are dynamically similar to oscillatory laminar flow and phases corresponding to flow deceleration are dynamically similar to fully developed turbulent flow. Moreover, the flow becomes more turbulent-like with increasing period of oscillation.

Isotropic boundary adapted wavelets for coherent vorticity extraction in turbulent channel flows. MARIE FARGE, LMD-IPSLS-CNRS Ecole Normale Supérieure, Paris, France, TELUO SAKURAI, KATSUNORI YOSHIMATSU, Nagoya University, Nagoya, Japan, KAI SCHNEIDER, M2P2-CNRS and CMI, Aix-Marseille University, Marseille, France, KOJI MORISHITA, Kobe University, Kobe, Japan, TAKASHI ISHIHARA, Nagoya University, Nagoya, Japan — We present a construction of isotropic boundary adapted wavelets, which are orthogonal and yield a multi-resolution analysis. We analyze DNS data of turbulent channel flow computed at a friction-velocity based Reynolds number of 395 and investigate the role of coherent vorticity. Thresholding of the wavelet coefficients allows to split the flow into two parts, coherent and incoherent vorticity. The statistics of the former, i.e., energy and enstrophy spectra, are close to the ones of the total flow, and moreover the nonlinear energy budgets are well preserved. The remaining incoherent part, represented by the large majority of the weak wavelet coefficients, corresponds to a structureless, i.e., noise-like, background flow and exhibits an almost equi-distribution of energy.

A Reduced Nonlinear Model of Wall-Bounded Shear Flow Turbulence. BRIAN FARRELL, Harvard University, PETROS IOANNOU, MARIOS NIKOLAIIDIS, National and Kapodistrian University of Athens, ALEXANDROS DINOPOULOS, DENNIS S. Ingber, National Institute of Standards and Technology, JOHNS HOPKINS UNIVERSITY, Johns Hopkins University — The roll/streak is the dominant structure in the dynamics of wall-bounded shear flow turbulence. It appears that this structure arises from a nonlinear instability, the various proposed mechanisms for which are referred to as self-sustaining processes. However, even once the nonlinear instability is identified there remains the problem of understanding how this instability is regulated to maintain the observed turbulent state. Here both of these questions will be addressed by adopting the perspective of statistical state dynamics (SSD), specifically its reduced nonlinear (RNL) implementation. RNL comprises the joint evolution of the streamwise constant mean flow (first cumulant) and second order perturbation statistics (second cumulant). This restriction greatly reduces the complexity of the dynamics while retaining a realistic SSP. The perturbations supporting the SSP in RNL arise from parametric instability of the time-dependence break of the statistical stability of these perturbations being enforced by a feedback mediated control process operating between the mean flow and the perturbations. In this talk it will be shown how the maintenance and regulation of RNL turbulence allows insight into the mechanism of turbulence in wall-bounded shear flow.

Non-unique frictional drag in turbulent plane Couette flows. DONGRONG ZHANG, GUSTAVO GIOIA, PINAKI CHAKRABORTY, Okinawa Institute of Science and Technology — There is a long standing mystery concerning frictional drag in fully developed turbulent plane Couette flows. In manifest defiance to the predictions from dimensional analysis, experiments have consistently shown that the frictional drag, f, is not a unique function of the Reynolds number, Re. In fact, the f vs. Re data fall on two distinct curves. The origin of these two curves dates back to the 1950s when Reichardt and Robertson independently performed their classical experiments. Subsequent works have found f vs. Re data to be in accord with the Reichardt curve or with the Robertson curve. Here we examine this problem from the perspective of the spectral link, the link between macroscopic properties (like f and the mean velocity profile, MVP) and the turbulent energy spectrum. We argue that since the flow is driven by moving boundaries, the boundaries affect the large length scales of the spectrum differently in the different setups. Using the spectral link we predict that the Reichardt and Robertson curves correspond to disparate features in the MVP: the presence or absence of an overshooting wake, respectively. While the different experiments and simulations did not report the spectrum, we verify our predictions by comparing the f vs. Re curves with the attendant MVPs.

Mean dynamics of a turbulent plane wall jet. FARAZ MEHDI, Hypertherm Inc., JOSEPH KLEWICKI, University of New Hampshire, University of Melbourne — Experimental and large-eddy simulation data are used to investigate the balances between viscous and inertial forces in plane turbulent wall jets. In recent years, analysis of the mean momentum balance in its unintegrated form has been shown to provide a mathematically and physically useful means for clarifying the leading order mean dynamics as a function of the transverse coordinate. Distinct from its laminar counterpart, each of the terms in the appropriately simplified form of the mean dynamical equation for the planar turbulent wall jet is leading order somewhere, but not everywhere, across the flow domain. Similar to what is observed in the canonical turbulent wall-flows, there is a wall region where the mean viscous force retains leading order. The wall jet, however, contains two peaks of opposite sign in its Reynolds stress profile. With distance from the wall, the first peak is associated with the loss of a leading order viscous force, while the outer peak is akin to the wholly inertial balance exchange that occurs in shear-wake flows. The physics of these balances exchanges are described, the scaling behaviors of the leading order balance layers are estimated, and the present findings are compared with previous models of planar wall jet structure.

Tuesday, November 24, 2015 12:50PM - 3:13PM –
Session R21 Turbulence: DNS 209 - David Goldstein, University of Texas at Austin
12:50PM R21.00001 Energy spectrum in high Reynolds number turbulence - high resolution DNS results. KOJI MORISHITA, Kobe University, TAKASHI ISHIHARA, Nagoya University, YUKIO KANEDA, Aichi Institute of Technology, MITSUO YOKOKAWA, Kobe University, ATSUYA UNO, RIKEN AICS — The energy spectrum and energy flux in high Reynolds number (Re) forced incompressible turbulence are investigated by using high-resolution DNS in a periodic box. We used negative viscosity (at a wavenumber range \( kL < 3 \)) to keep the total energy constant, and used well-developed turbulence fields as the initial conditions (\( L \) is the integral length scale). The DNS with up to \( 6144^3 \) grid points show that, after a transient period of the order of eddy turnover time, the standard deviation of the energy spectrum and that of the energy flux are largest at \( \text{Re} \eta \). As in previous studies, the energy spectra are insensitive to the values of \( \eta \) when \( k_{max} \eta \geq 1 \) (\( \eta \) is the Kolmogorov length scale). The time-averaged, normalized energy spectra of high \( \text{Re} \) turbulence at high \( k \) overlap well with each other when they are plotted against \( k \eta \). The normalized spectra have a slope steeper than \(-5/3\) (the Kolmogorov scaling law) by factor 0.1 at \( k \lambda \sim 1 \) (\( \lambda \) is Taylor micro-scale). The DNS suggest that there is another wavenumber range (\( kL < 1 \)) in which the spectrum has a slope close to \(-5/3\), and also that the latter range increases with \( \text{Re} \) and the Kolmogorov constant is \( 1.8 \pm 0.1 \).

1:03PM R21.00002 Direct numerical simulation of incompressible acceleration-driven variable-density turbulence\(^1\), ILANA GAT, CALIFORNIA INSTITUTE OF TECHNOLOGY, GEORGIOS MATHEOU, California Institute of Technology, NASA Jet Propulsion Laboratory, DANIEL CHUNG, University of Melbourne, PAUL DIMOTAKIS, California Institute of Technology — Fully developed turbulence in variable-density flow driven by an externally imposed acceleration field, e.g., gravity, is fundamental in many applications, such as inertial confinement fusion, geophysics, and astrophysics. Aspects of this turbulence regime are poorly understood and are of interest to fluid modeling. We investigate incompressible acceleration-driven variable-density turbulence by a series of direct numerical simulations of high-density fluid in-between slabs of low-density fluid, in a triply-periodic domain. A pseudo-spectral numerical method with a Helmholtz-Hodge decomposition of the pressure field, which ensures mass conservation, is employed, as documented in Chung & Pullin (2010). A uniform dynamic viscosity and local Schmidt number of unity are assumed. This configuration encapsulates a combination of flow phenomena in a temporally evolving variable-density shear flow. Density ratios up to 10 and Reynolds numbers in the fully developed turbulent regime are investigated. The temporal evolution of the vertical velocity difference across the shear layer, shear-layer growth, mean density, and Reynolds number are discussed. Statistics of Lagrangian accelerations of fluid elements and of vorticity as a function of the density ratio are also presented.

\(^1\)This material is based upon work supported by the AFOSR, the DOE, the NSF GRFP, and Caltech.

1:16PM R21.00003 A lattice-Boltzmann scheme of the Navier-Stokes equations on a 3D cuboid lattice. HAODA MIN, CHENG PENG, LIAN-PING WANG, University of Delaware — The standard lattice-Boltzmann method (LBM) for fluid flow simulation is based on a square (in 2D) or cubic (in 3D) lattice grids. Recently, two new lattice Boltzmann schemes have been developed on a 2D rectangular grid using the MRT (multiple-relaxation-time) collision model, by adding a free parameter in the definition of moments or by extending the equilibrium moments. Here we developed a lattice Boltzmann model on 3D cuboid lattice, namely, a lattice grid with different grid lengths in different spatial directions. We designed our MRT-LBM model by matching the moment equations from the Chapman-Enskog expansion with the Navier-Stokes equations. The model guarantees correct hydrodynamics. A second-order term is added to the equilibrium moments in order to restore the isotropy of viscosity on a cuboid lattice. The form and the coefficients of the extended equilibrium moments are determined through an inverse design process. An additional benefit of the model is that the viscosity can be adjusted independent of the stress-moment relaxation parameter, thus improving the numerical stability of the model. The resulting cuboid MRT-LBM model is then validated through benchmark simulations using laminar channel flow, turbulent channel flow, and the 3D Taylor-Green vortex flow.

1:29PM R21.00004 A numerical study of turbulence under time-dependent axisymmetric contraction and subsequent relaxation\(^1\), M.P. CLAY, P.K. YEUNG, Georgia Tech, Z. WARHAFT, Cornell Univ. — Turbulence subjected to axisymmetric strain is a fundamental problem which is common in engineering equipment with variable cross-section, but is not yet fully understood. We have performed direct numerical simulations on a deforming domain with grids up to \( 1024^3 \) and a time-dependent strain history designed to mimic spatial gradients in wind-tunnel experiments (Ayyalasomayajula & Warhaft J Fluid Mech 566, 273-307 (2006)). Isotropic turbulence with a specified energy spectrum is allowed to decay and then passed through a numerical conduit of 4:1 contraction ratio. The Reynolds stress tensor, velocity gradient variances, and longitudinal and transverse one-dimensional (1D) spectra are studied during both the contraction and subsequent relaxation. Contraction leads to amplification of energy in the compressed directions and departures from local isotropy. When the strain is removed local isotropy returns quickly while the energy decays with a power law exponent smaller than for decaying isotropic turbulence. The evolution of 1D spectra including changes in shape is consistent with experiments, but a large solution domain is important.

\(^1\)Supported by NSF Grant CBET-1510749 (Fluid Dynamics Program).

1:42PM R21.00005 Extreme events and small-scale structure in computational turbulence\(^1\), X.M. Zhai, P.K. Yeung, Georgia Tech, K.R. Sreenivasan, New York Univ — Detailed analyses have been made of data from a direct numerical simulation of turbulence on a periodic domain with \( 8192^3 \) grid points designed to improve our understanding of small-scale structure and intermittency. At the Reynolds number of this simulation (1300 based on the Taylor scale) extreme events of dissipation and enstrophy as large as \( 10^5 \) times the mean value are observed. These events are shown to possess a form that is different from similar events at low Reynolds numbers. Extreme vorticity appears to be “chunky” in character, in contrast to elongated vortex tubes at moderately large amplitudes commonly reported in the literature. We track the temporal evolution of these extreme events and find that they are generally short-lived, which suggests frequent sampling on-the-fly is useful. Extreme magnitudes of energy dissipation rate and enstrophy are essentially coincident in space and remain so during their evolution. Numerical tests show sensitivity to small-scale resolution and sampling but not machine precision. The connections expected between indicators of fine-scale intermittency such as acceleration statistics and the anomalous scaling of high-order velocity structure functions are also investigated.

\(^1\)Supported by NSF Grant ACI-1036170 (Track 1 Petascale Resource Allocations Program).
1:55PM R21.00006 Optimization of flow initialization and perturbation forcing for fast transition towards fully developed turbulent channel flow, XIN WEN, CHENG PENG, LIAN-PING WANG, University of Delaware — The turbulent channel flow has been used as the simplest setup to study the flow structure and dynamics in wall-bounded single-phase and multiphase turbulence. An aspect that has not been well studied in direct numerical simulation of such flow is how to initiate such flow simulation and force the flow so that a fully developed turbulence can be achieved relatively quickly. Often it may take ~ 50 eddy turnover times (defined in terms of the channel half width and wall friction velocity) for the flow to evolve to a fully developed stage, due to different time scales involved in the wall region and the center of the channel and coupling of the flows between the two. In this talk, we explore different ways to initialize the flow and to excite the flow at the early stage. The initialization typically consists of a mean flow and a disturbance flow. The excitations could be done by adding external perturbation forcing. The parameters in the initial flow and the forcing affect the speed of transition to realistic fully developed turbulence. We will discuss how to control these parameters so that realistic flow structures, Reynolds stress and rms velocity profiles can be generated quickly while maintaining a nearly constant mean flow speed.

2:08PM R21.00007 The effect of spatial discretization upon traveling wave body forcing of a turbulent wall-bounded flow, SOYOUNG YOU, DAVID GOLDSTEIN, University of Texas at Austin — DNS is employed to simulate turbulent channel flow subject to a traveling wave body force field near the wall. The regions in which forces are applied are made progressively more discrete in a sequence of simulations to explore the boundaries between the effects of discrete flow actuators and spatially continuum actuation. The continuum body force field is designed to correspond to the “optimal” resolvent mode of McKeon and Sharma (2010), which has the L2 norm of σ1. That is, the normalized harmonic forcing that gives the largest disturbance energy is the first singular mode with the gain of σ1. 2D and 3D resolvent modes are examined at a modest Reτ of 180. For code validation, nominal flow simulations without discretized forcing are compared to previous work by Sharma and Goldstein (2014) in which we find that as we increase the forcing amplitude there is a decrease in the mean velocity and an increase in turbulent kinetic energy. The same force field is then sampled into isolated sub-domains to emulate the effect of discrete physical actuators. Several cases will be presented to explore the dependencies between the level of discretization and the turbulent flow behavior.

2:21PM R21.00008 Turbulence structure subjected to “precession-like” rotation, KARTIK IYER, IRENE MAZZITELLI, LUCA BIFERALE, FABIO BONACCORSO, University of Rome Tor Vergata — We report results from a series of numerical experiments in which the orientation of the rotation axis of a turbulent flow simulated in a periodic domain is arbitrarily changed. It is well known that rotation weakens spectral transfer and renders the flow anisotropic across all scales. However, when the orientation of rotation is changed, the spectral transfer becomes stronger and the flow becomes more isotropic. The large scale vortical structures aligned with the rotation are destroyed by the change in rotation axis. Based on these findings we attempt to discuss the dynamics of rotating turbulence subjected to precession.

1Supported by the ERC AdG NewTURB num. 339032

2:34PM R21.00009 The intense vorticity structures in isotropic turbulence with a “Carreau-Yasuda” fluid, AFONSO GHIRA, CARLOS SILVA, Instituto Superior Técnico/University of Lisbon — Direct numerical simulations of isotropic turbulence are carried out to assess the flow topology and the dynamics of the intense vorticity structures in a shear-thinning fluid. Specifically, the Carreau-Yasuda fluid model is used to describe the shear-thinning viscosity while the intense vorticity structures are tracked using a numerical algorithm. The eddy characteristics are compared to the ones observed in Newtonian turbulence and the effects of the shear-thinning are assessed in relation to the small scale dynamics of the flow.

2:47PM R21.00010 Direct Numerical Simulations of Sound-Orifice-Boundary Layer Interaction, QI ZHANG, DANIEL BODONY, University of Illinois at Urbana Champaign — We report on a series of direct numerical simulations (DNS) of the interaction of a monochromatic incident acoustic field with a cavity-backed circular orifice in the presence of laminar and turbulent boundary layers of freestream Mach number 0.5 and momentum thickness Reynolds number 2,300, with application to acoustic liners. The simulations show that the addition of the orifice increases the drag and can induce laminar-to-turbulent transition at sufficiently high acoustic levels. Furthermore, the sound-orifice-boundary layer system support three distinct timescales whose spatial distributions change with the phase of the incident sound. Details of the near-orifice interaction are studied to create a model of the orifice discharge coefficient that is part of a time-domain, nonlinear reduced-order model (ROM) for the liner impedance. Comparisons between the ROM-predicted and DNS-measured near-orifice flow and acoustic impedance are given.

3:00PM R21.00011 A constant-energy physical-space forcing method for steadier statistics and faster convergence to homogeneous-isotropic turbulence, MAXIME BASSENNE, JAVIER URZAY, GEORGE I. PARK, PARVIZ MOIN, Center for Turbulence Research, Stanford University — We investigate a new constant-energy forcing method for homogeneous-isotropic turbulent flows forced linearly in physical space. The method bears no computational overhead and it consists of a proportional controller embedded in the forcing coefficient. Comparisons of this forcing method are made with other existing variable-energy approaches, using direct numerical simulations (DNS) and large-eddy simulations (LES). We find that the proposed forcing method shortens the transient period from an user-defined artificial flow field to forced turbulence while maintaining steadier statistics. For illustration, the proposed forcing method is applied to a dilute particle-laden homogeneous-isotropic turbulent flow to highlight some of the influences of the forcing strategies on the statistics of the disperse phase.

1Funded by PSAAP-II DoE/NNSA
12:50PM R22.00001 Characteristics of secondary flows in rough-wall turbulent boundary layers, CHRISTINA VANDERWIEL, BIHARTHAM GANAPATHISUBRAMANI, University of Southampton — Large-scale secondary motions consisting of counter-rotating vortices and low- and high-momentum pathways can form in boundary layers that develop over rough surfaces. We experimentally investigated the sensitivity of these secondary motions to spanwise arrangement of the roughness by studying the flow over streamwise-aligned rows of elevated roughness with systematically-varied spacing. The roughness is created with LEGO blocks mounted along the floor of the wind tunnel and Stereo-PIV is used to measure the velocity field in a cross-plane. Results show that the secondary flows are strongest when the spanwise spacing of the surface topology is comparable with the boundary layer thickness. We discuss how these results are relevant to flows over arbitrary topologies and how these secondary motions influence the Reynolds stress distribution in the boundary layer.

1:03PM R22.00002 The effect of transitionally-rough surfaces on near-wall turbulence, NABIL ABDERRAHAMAN-ELENA, RICARDO GARCA-MAYORAL, Univ of Cambridge — We present results of DNSs of channel flow with rough walls in the transitionally-rough regime, for \( k^+ \leq 15 \). Through flow visualization and statistical analysis, we show that the resulting fluctuations can be separated into two components: one due to the overlying near-wall turbulence, and one due to the presence of the roughness. The latter is essentially the phase-averaged fluctuation that is observed also for laminar flows, but intensely modulated in amplitude by the overlying turbulence. The above decomposition of the fluctuations can be used to develop predictive models for the onset of roughness effects.

1:16PM R22.00003 Direct numerical simulations of the dense regime of roughness, MICHAEL MACDONALD, LEON CHAN, DANIEL CHUNG, NICHOLAS HUTCHINS, ANDREW OOI, University of Melbourne — We investigate the sparse and dense regimes of roughness using Direct Numerical Simulations (DNS) of turbulent flow over three-dimensional sinusoidal surfaces in the transitionally rough regime. The sparse regime is known to lead to an increase in the Hama roughness function, \( \Delta U^+ \), as the roughness density increases, while the dense regime is associated with a decrease in \( \Delta U^+ \) as density increases. In this parametric study, the wavelength of the sinusoidal roughness elements is varied while the roughness height is fixed. The minimal-span channel is used, as the high cost of the grid would otherwise make the dense roughness simulations unattainable. It was found that the dense regime begins at solidity values (frontal area divided by wall-parallel projected area) greater than 0.15, in agreement with the literature. An analysis of the mean momentum balance above the roughness reveals that the decrease in \( \Delta U^+ \) in the dense regime is due to a reduction in the Reynolds shear stress. This reduction is located just above the roughness crest in the near-wall region, and the difference in the energy spectra of streamwise velocity between smooth and roughness clearly demonstrates that this is at long streamwise length scales.

1:29PM R22.00004 High Reynolds number rough-wall turbulent boundary layers, DOUGAL SQUIRE, CALEB MORRILL-WINTER, University of Melbourne, MICHAEL SCHULTZ, United States Naval Academy, NICHOLAS HUTCHINS, University of Melbourne, JOSEPH KLEWICKI, University of Melbourne, University of New Hampshire, IVAN MARUSIC, University of Melbourne — In his review of turbulent flows over rough-walls, Jimenez (2004) concludes that there are gaps in the current database of relevant experiments. The author calls for measurements in which \( \delta/k \) and \( k^+ \) are both large—low blockage, fully-rough flow—and where \( \delta/k \) is large and \( k^+ \) is small—low blockage, transitionally-rough flow—to help clarify ongoing questions regarding the physics of rough-wall-bounded flows. The present contribution details results from a large set of measurements carried out above sandpaper in the Melbourne Wind Tunnel. The campaign spans 45 rough-wall measurements using single and multiple-wire hot-wire anemometry sensors and particle image velocimetry. A floating element drag balance is employed to obtain the rough-wall skin friction force. The data span \( 20 < k_{\delta}^+ < 160 \) and \( 30 < \delta/k < 200 \) across a friction Reynolds number range of \( 2800 < R_{\infty} < 30000 \), targeting areas in the parameter space identified by Jimenez (2004) as being sparsely populated by pre-existing data. Smooth-wall data are also obtained across a similar Reynolds number range to enable comparison of smooth- and rough-wall structural features. Generally, the data indicate similarity in the outer-layer of smooth- and fully-rough-wall-bounded flows.

1:42PM R22.00005 Coupling between roughness and freestream acceleration in turbulent boundary layers, JUNLIN YUAN, UGO PIOMELLI, Queen’s University — To explain various rough-wall flow responses to different types of free-stream conditions previously observed, we carried out a direct numerical simulation of a spatially developing turbulent boundary layer with freestream acceleration. Unlike the equilibrium (self-similar) accelerating scenario, where a strong acceleration leads to complete laminarization and lower friction, in the present non-equilibrium case the friction coefficient increases with acceleration, due to the faster near-wall acceleration than that of the freestream. At the same time, roughness reduces the near-wall time scale of the turbulence, preventing the acceleration from linearly stretching the near-wall eddies and freezing the turbulence intensity as in the smooth case. In addition, acceleration leads to similar decrease of mean-velocity logarithmic slope on rough and smooth walls; this allows a clear definition of the roughness function in a local sense. Interestingly, this roughness function correlates with the roughness Reynolds number in the same way as in self-similar or non-accelerating flows. This study may also help develop benchmark cases for evaluating rough-wall treatments for industrial turbulence models.

1:55PM R22.00006 ABSTRACT WITHDRAWN —

2:08PM R22.00007 Turbulent boundary layer over 2D and 3D large-scale wavy walls, LEONARDO P. CHAMORRO, ALI M. HAMED, University of Illinois at Urbana-Champaign, LUCIANO CASTILLO, Texas Tech University — In this work, an experimental investigation of the developing and developed flow over two- and three-dimensional large-scale wavy walls was performed using high-resolution planar particle image velocimetry in a refractive-index-matching flume. The 2D wall is described by a sinusoidal wave in the streamwise direction with amplitude to wavelength ratio \( a/\lambda = 0.05 \). The 3D wall is defined with an additional wave superimposed on the 2D wall in the spanwise direction with \( a/\lambda = 0.1 \). The flow was characterized at Reynolds numbers of 4000 and 40000, based on the bulk velocity and the flume half height. Instantaneous velocity fields and time-averaged turbulence quantities reveal strong coupling between large-scale topography and the turbulence dynamics near the wall. Turbulence statistics show the presence of a well-structured shear layer that enhances the turbulence for the 2D wavy wall, whereas the 3D wall exhibits different flow dynamics and significantly lower turbulence levels, particularly for \( u'/v' \) which shows about 30% reduction. The likelihood of recirculation bubbles, levels and spatial distribution of turbulence, and the rate of the turbulent kinetic energy production are shown to be severely affected when a single spanwise mode is superimposed on the 2D wall. POD analysis was also performed to further understand distinctive features of the flow structures due to surface topography.
2:21PM R22.00008 Micro PIV measurements of turbulent flow over 2D structured roughness1, JOEL HARTENBERGER, MARC PERLIN, University of Michigan — We investigate the turbulent boundary layer over surfaces with 2D spanwise square and triangular protrusions having nominal heights of 100 - 300 microns for Reynolds numbers ranging from Reτ ≈ 1500 through Reτ ≈ 4500 using a high speed, high magnification imaging system. Micro PIV analysis gives finely resolved velocity fields of the flow (on the order of 10 microns between vectors) enabling a detailed look at the inner region as well as the flow in the immediate vicinity of the roughness elements. Additionally, planar PIV with lower resolution is performed to capture the remainder of the boundary layer to the freestream flow. Varying the streamwise distance between individual roughness elements from one to ten times the nominal heights allows investigation of k-type and d-type roughness in both the transitionally rough and fully rough regimes. Preliminary results show a shift in the mean velocity profile similar to the results of previous studies. Turbulent statistics will be presented also.

1The authors would like to acknowledge the support of NAVSEA which funded this project through the Naval Engineering Education Center (NEEC)

2:34PM R22.00009 Turbulent shear-flow over fractal arrays of surface-mounted cubes, DANIEL J. WISE, WERNHER BREVIS, Univ of Sheffield, SHEFFIELD FLUID MECHANICS GROUP TEAM — The turbulent shear-flow over a bottom-wall fully covered by periodic multi-scale arrangements of obstacles is examined via Particle Image Velocimetry (PIV), Volumetric 3D Velocimetry (V3V) and Acoustic Doppler Velocimetry (ADV) measurements. Three obstacle patterns are utilised, all based on different numbers of iterations of the Sierpinski carpet fractal. In each case 2D/3D velocity fields of the flow formed within the porous channels, namely the flow beneath the mean obstacle height, are presented and analysed with respect to standard statistics such as the mean, rms velocity profiles, and the Reynolds stresses. Point-wise measurements within the obstacle arrays reveal that the presence of the obstacles, and in particular their injection of energy at the associated wavelengths, has unexpected effects on the slope of the energy spectra within the turbulent porous flow. The region dominated by these spectral characteristics is defined. It is also shown that this behaviour is not observed in the outer flow.

2:47PM R22.00010 DNS study of amplitude modulation statistics of turbulent channel flows over rough walls, SICONG WU, University of Illinois at Urbana-Champaign, KENNETH CHRISTENSEN, University of Notre Dame, CARLOS PANTANO, University of Illinois at Urbana-Champaign — DNS of long turbulent channel flows over rough walls at friction Reynolds number up to 400 are considered. The walls are hexagonally packed with hemispheres with roughness height h/k=10 and 20 and average spacing between hemispheres from 2 to 4 times the roughness height. A conforming grid approach (unstructured) using spectral finite elements is used to fully resolve the flow with up to 1.6 billion grid points. Analysis of two-point correlation and Hilbert transform applied to the spectrally filtered fluctuating velocity signals is used to study the interaction between large-scale and small-scale turbulence structures. The talk focuses on the effect of roughness parameters on the amplitude modulation accuracy, both for the wall-parallel and wall-normal velocities.

3:00PM R22.00011 Amplitude modulation of streamwise velocity fluctuations in the roughness sublayer: evidence from large-eddy simulations1, ANKIT AWASTHI, WILLIAM ANDERSON, UT Dallas — Large-scale motions in the logarithmic region of turbulent boundary layers amplitude modulate the viscous sublayer (Marusic et al., 2010: Science; Mathis et al., 2009: J. Fluid Mech.). This finding has promising implications for large-eddy simulation of wall-bounded turbulence at high Reynolds number (wherein the turbulence integral length exhibits linear proportionality with wall-normal elevation). Existing amplitude modulation studies have addressed smooth wall flows, though high Reynolds number rough wall flows are ubiquitous. Under such conditions, roughness-scale vortices ablate the viscous sublayer and result in the roughness sublayer. The roughness sublayer depth scales with aggregate element height, k, and is typically 2k ~ 3k. Above this, Townsend’s Hypothesis dictates that the logarithmic layer is unaffected by the roughness sublayer. Here, we present large-eddy simulation results of turbulent channel flow over rough walls. We follow the decoupling procedure of Mathis et al., 2009: J. Fluid Mech., and present evidence that outer-layer dynamics amplitude modulate the roughness sublayer. Below the roughness element height, we report enormous sensitivity to element proximity. Above the elements, but within the roughness sublayer, topography dependence rapidly declines.

1This work was supported by the Air Force Office of Scientific Research, Turbulence and Transition Program (PM: Dr. R. Ponnoppan) under Grant # FA9550-14-1-0101. Computational resources were provided by the Texas Adv. Comp. Center at the Univ. of Texas.

3:13PM R22.00012 A phenomenological model for the roughness function in turbulent boundary layers with macro-scale roughness elements1, JASIM SADIQUE, XIANG YANG, CHARLES MENEVEAU, RAJAT MITTAL, The Johns Hopkins University — There has been extensive work done in the past to predict the roughness function associated with rough wall boundary layers and to connect it to the roughness topology. Correlations have been obtained from experiments for a variety of cases and attempts have also been made to use physics based models to obtain the roughness function. In this talk we present a way to derive an explicit formula that connects the rough wall boundary layer parameters to the roughness geometry and arrangement. We assume a two-layer model for the velocity: a log-law in the outer layer and an exponential profile in the canopy layer, and make use of the concept of ‘mutual sheltering’. The analysis focuses mainly on rectangular prism shaped roughness elements with different arrangements such as aligned, staggered, rotated, and inclined to the flow, and also with a distribution of heights. It is found that the derived formula, which is simple to apply, matches the results from a variety of large-eddy simulations and experiments. It is also shown that the disparate cases collapse onto a single curve using a parameter depending only on the geometry. This formula gives a quick and accurate way to predict the roughness function from the surface geometry, and can also be extended to other types of surfaces.

1This research is supported by The Office of Naval Research through ONR grant N00014-12-1-0582

Tuesday, November 24, 2015 12:50PM - 3:26PM –
Session R23 Biofluids: Red Blood Cell Dynamics and Clotting 300 - Xin Yong, State University of New York
The flow of red blood cells in stenosed microvessels and the influence of red blood cells on wall-bounded rolling motion of microparticles

12:50PM R23.00001 The flow of red blood cells in stenosed microvessels and the influence of red blood cells on wall-bounded rolling motion of microparticles, KOOHYAR VAHIDKHAI, PETER BALOGH, PROSENJIT BAGCHI, Rutgers University — In the first part of this work, we consider a 3D computational study of the flow of deformable red blood cells in stenosed microvessels. We observe that the apparent viscosity of blood increases by several folds, and the rate of increase with increasing vessel diameter is also higher than that in non-stenosed vessels, implying an enhancement of the wall-bounded rolling effect of the cells. Thus the RMS of the flow rate oscillations in the stenosed vessel is observed to be significantly higher that that in the non-stenosed vessel. Furthermore, several folds increase in the Eulerian velocity fluctuations and a transient flow reversal upstream the stenosed region are also observed, which would not occur in absence of the cells. In the second part, we consider the adhesive rolling motion of wall-bounded microparticles in presence of flowing red blood cells in microvessels. We observe two contrasting behaviors of the red blood cells: on one hand, the cells facilitate the establishment of the particle-wall contact, and, thereby, initiation of adhesion. On the other hand, they augment the rolling velocity of the particles. Implications of these results on the optimal design of drug carriers are discussed.

1:03PM R23.00002 Reduced-order models of the coagulation cascade1, KIRK B. HANSEN, SHAWN C. SHADDEN, University of California, Berkeley — Previous models of flow-mediated thrombogenesis have generally included the transport and reaction of dozens of biochemical species involved in the coagulation cascade. Researchers have shown, however, that thrombin generation curves can be accurately reproduced by a significantly smaller system of reactions. These reduced-order models are based on the system of ordinary differential equations representative of a well-mixed system, however, not the system of advection-diffusion-reaction equations required to model the flow-mediated case. Additionally, they focus solely on reproducing the thrombin generation curve, although accurate representation of certain intermediate species may be required to model additional aspects of clot formation, e.g., interactions with activated and non-activated platelets. In this work, we develop a method to reduce the order of a coagulation model through optimization techniques. The results of this reduced-order model are then compared to those of the full system in several representative cardiovascular flows.

1:16PM R23.00003 Mesoscopic Modeling of Blood Clotting: Coagulation Cascade and Platelets Adhesion, ALIREZA YAZDANI, ZHEN LI, GEORGE KARNIAKIS, Brown University — The process of clot formation and growth at a site on a blood vessel wall involve a number of multi-scale simultaneous processes including: multiple chemical reactions in the coagulation cascade, species transport and flow. To model these processes we have incorporated advection-diffusion-reaction (ADR) of multiple species into an extended version of Dissipative Particle Dynamics (DPD) method which is considered as a coarse-grained Molecular Dynamics method. At the continuum level this is equivalent to the Navier-Stokes equation plus one advection-diffusion equation for each species. The chemistry of clot formation is now understood to be determined by mechanisms involving reactions among many species in dilute solution, where reaction rate constants and species diffusion coefficients in plasma are known. The role of blood particulates, i.e. red cells and platelets, in the clotting process is studied by including them separately and together in the simulations. An agonist-induced platelet activation mechanism is presented, while platelets adhesive dynamics based on a stochastic bond formation/dissociation process is included in the model.

1:29PM R23.00004 Effect of Strain Rate on the Deformation of Red Blood Cells Entering a Constriction, JORDAN MANCUSO, WILLIAM RISTENPARK, Dept. Chemical Engineering, University of California Davis — Although much work has investigated the stretching behavior of RBCs in shear flows, relatively little work has examined the deformation that occurs in the physiologically important extensional flow at the entrance to a constriction. In particular, there is currently no analytical model to predict the extent of deformation as a function of the strain rate in the constriction entrance. Here we experimentally elucidate the relationship between strain rate and the dynamic stretching behavior of RBCs as they enter a microfluidic constriction. We systematically varied the strain rate and observed two contrasting behaviors of the red blood cells: on one hand, the cells facilitate the establishment of the particle-wall contact, and, thereby, initiation of adhesion. On the other hand, they augment the rolling velocity of the particles. Implications of these results on the optimal design of drug carriers are discussed.

1:42PM R23.00005 Chaotic dynamics of red blood cells in oscillating shear flow, PROSENJIT BAGCHI, DANIEL CORDASCO, Rutgers University — A 3D computational study of deformable red blood cells in dilute suspension and subject to sinusoidally oscillating shear flow is considered. It is observed that the cell exhibits either a periodic motion or a chaotic motion. In the periodic motion, the cell reverses its orientation either about the flow direction or about the flow gradient, depending on the initial conditions. In certain parameter range, the initial conditions are forgotten and the cells become entrained in the same sequence of horizontal reversals. The chaotic dynamics is characterized by a nonperiodic sequence of horizontal and vertical reversals, and swings. The study provides the first conclusive evidence of the chaotic dynamics of fully deformable cells in oscillating flow using a deterministic numerical model without the introduction of any stochastic noise. An analysis of the chaotic dynamics shows that chaos is only possible in certain frequency bands when the cell membrane can rotate by a certain amount allowing the cells to swing near the maximum shear rate. We make a novel observation that the occurrence of the vertical or horizontal reversal depends only on whether a critical angle, that is independent of the flow frequency, is exceeded at the instant of flow reversal.

1:55PM R23.00006 Mechanosensing Dynamics of Red blood Cells, JIANDI WAN, Rochester Institute of Technology — Mechanical stress-induced deformation of human red blood cells (RBCs) plays important physiological roles in oxygen delivery, blood rheology, transfusion, and malaria. Recent studies demonstrate that, in response to mechanical deformation, RBCs release adenosine-5’-triphosphate (ATP), suggesting the existence of mechanotransductive pathways in RBCs. Most importantly, the released ATP from RBCs regulates vascular tone and impaired release of ATP from RBCs has been linked to diseases such as type II diabetes and cystic fibrosis. To date, however, the mechanisms of mechanotransduction release of ATP from RBCs remain unclear. Given that RBCs experience shear stresses continuously during the circulation cycle and the released ATP plays a central role in vascular physiology, understanding the mechanotransductive release of ATP from RBCs will provide not only fundamental insights to the role of RBCs in vascular homeostasis but also novel therapeutic strategies for red cell dysfunction and vascular disease. This talk describes the main research in my group on integrating microfluidic-based approaches to study the mechanosensing dynamics of RBCs. Specifically, I will introduce a microfluidic approach that can probe the dynamics of shear-induced ATP release from RBCs with millisecond resolution and provide quantitative understandings of the mechanosensitive ATP release processes in RBCs. Furthermore, I will also describe our recent findings about the roles of the Piezo1 channel, a newly discovered mechanosensitive cation channel in the mechanotransductive ATP release in RBCs. Last, possible functions of RBCs in the regulation of cerebral blood flow will be discussed.
2:08PM R23.00007 Shape Recovery of Elastic Red Blood Cells from Shear Flow Induced Deformation in Three Dimensions1. YAN PENG, JOHN GOUNLEY, Old Dominion University — Red blood cells undergo substantial shape changes in vivo. Modeled as an elastic capsule, the shape recovery of a three-dimensional biconcave capsule from shear flow is studied for different preferred elastic and bending configuration. The fluid-structure interaction is modeled using the multiple-relaxation time lattice Boltzmann (LBM) and immersed boundary (IBM) methods. Based on the studies of the limited shape memory observed in three dimensions, the shape recovery is caused by the preferred elastic configuration, at least when paired with a constant spontaneous curvature. For these capsules, the incompleteness of the shape recovery observed precludes any conjecture about whether a single or multiple phase(s) are necessary to describe the recovery process. Longer simulations and a more stable methodology will be necessary.

1Y. Peng acknowledges support from Old Dominion University Research Foundation Grant #503921 and National Science Foundation Grant DMS-1319078.

2:21PM R23.00008 A simple model to understand the role of membrane shear elasticity and stress-free shape on the motion of red blood cells in shear flow1, ANNIE VIALLAT, AMU-CNRS, MANOUK ABKARIAN, Universit Montpellier CNRS, JULES DUPIRE, Microfactory company — The analytical model presented by Keller and Skalak on the dynamics of red blood cells in shear flow described the cell as a fluid ellipsoid of fixed shape. It was extended to introduce shear elasticity of the cell membrane. We further extend the model when the cell discoidal physiological shape is not a stress-free shape. We show that spheroid stress-free shapes enable fitting experimental data with values of shear elasticity typical to that found with micropipettes and optical tweezers. For moderate shear rates (when RBCs keep their discoid shape) this model enables to quantitatively determine an effective cell viscosity, that combines membrane and hemoglobin viscosities and an effective shear modulus of the membrane that combines shear modulus and stress-free shape. This model allows determining RBC mechanical parameters both in the tanktreading regime for cells suspended in a high viscosity medium, and in the tumbling regime for cells suspended in a low viscosity medium. In this regime, a transition is predicted between a rigid-like tumbling motion and a fluid-like tumbling motion above a critical shear rate, which is directly related to the mechanical parameters of the cell.

1A*MIDEX (n ANR-11-IDEX-0001-02) funded by the “Investissements d’Avenir”, Region Languedoc-Roussillon, Labex NUMEV (ANR-10-LABX-20), BPI France project DataDiag

2:34PM R23.00009 Quantification of hydrodynamic factors influencing cell lateral migration, STEPHANIE NIX, Akita Prefectural Univ, YOHISUKU IMAI, TAKUJI ISHIKAWA, Tohoku University — The study of the migration of blood cells perpendicular to the direction of blood flow, or lateral migration, is motivated by the differing behavior of the various types of blood cells. In vivo, red blood cells are observed to flow in the central region of the blood vessel, particularly in the microcirculation, while other types of cells in the blood, including white blood cells and platelets, are observed to flow disproportionately near the vessel wall. However, the specifics regarding the effect of hydrodynamic and biological factors are still unknown. Thus, in this study, we aim to quantify the effect of hydrodynamic factors on a cell model numerically using the boundary integral method. By using the boundary integral method, we can isolate the effect of a single hydrodynamic factor, such as a wall or given flow distribution, in an otherwise infinite flow. Then, we can use the obtained numerical results to develop a semi-analytical model describing the cell lateral migration dependent on only the flow geometry and the viscosity ratio between the cell and external fluid.

2:47PM R23.00010 Coarse-grained theory to predict red blood cell migration in pressure-driven flow at zero Reynolds number, QIN M. QI, Stanford University, VIVEK NARSMIHAN, Massachusetts Institute of Technology, ERIC S.G. SHAQFEH, Stanford University — The pressure-driven flow of blood in a rectangular channel is studied via the development of a modified Boltzmann collision theory. It is well known that the deformability of red blood cells (RBC) creates a hydrodynamic lift away from the channel walls and most importantly, forms a cell-free or Fahraeus-Lindqvist layer at the wall. A theory is presented to predict the uneven concentration distribution of RBCs in the cross-stream direction. We demonstrate that cell migration is mainly due to the balance between the hydrodynamic lift from the wall and cell-cell binary collisions. Each of these components is determined independently via boundary element simulations. The lift velocity shows a scaling with wall displacement that is similar to that from previous vehicular experiments. The collisional displacements vary nonlinearly with cross-stream positions a key input to the theory. Unlike the case of simple shear flow, a nonlocal shear rate correction is necessary to overcome the problem of zero lift and collision at the centerline. Finally a diffusion term is added to account for higher order collisions. The results indicate a decrease in cell-free layer thickness with increasing RBC volume fraction that is in good agreement with simulation of blood in 10-20% range of hematocrit.

3:00PM R23.00011 Experimental comparison of mammalian and avian blood flow in microchannels1, KATHRYN FINK, DORIAN LIEPFMANN, Univ of California - Berkeley — The non-Newtonian, shear rate dependent behavior of blood in microchannel fluid dynamics has been studied for nearly a century, with a significant focus on the characteristics of human blood. However, for over 200 years biologists have noted significant differences in red blood cell characteristics across vertebrate species, with particularly drastic differences in cell size and shape between mammals and non-mammalian classes. We present an experimental analysis of flow in long microchannels for several varieties of mammalian and avian blood, across a range of hematocrits, channel diameters, and flow rates. Correlation of shear rate and viscosity is compared to existing constitutive equations for human blood to further quantify the importance of red blood cell characteristics. Ongoing experimental results are made available in an online database for reference or collaboration.

1K.F. acknowledges funding from the ARCS Foundation and an NSF Graduate Research Fellowship through NSF Grant DGE 1106400.

3:13PM R23.00012 Patient-specific modeling and analysis of dynamic behavior of individual sickle red blood cells under hypoxic conditions1, XUEJIN LI, Brown University, E. DU, Florida Atlantic University, ZHEN LI, YU-HANG TANG, LU LU, Brown University, MING DAO, Massachusetts Institute of Technology, GEORGE KARNIAKIS, Brown University — Sickle cell anemia is an inherited blood disorder exhibiting heterogeneous morphology and abnormal dynamics under hypoxic conditions. We developed a time-dependent cell model that is able to simulate the dynamic processes of repeated sickling and unsickling of red blood cells (RBCs) under physiological conditions. By using the kinetic cell model with parameters derived from patient-specific data, we present a mesoscopic computational study of the dynamic behavior of individual sickle RBCs flowing in a microfluidic channel with multiple microgrotes. We investigate how individual sickle RBCs behave differently from healthy ones in channel flow, and analyze the alteration of cellular behavior and response to single-cell capillary obstruction induced by cell rheologic rigidification and morphological change due to cell sickling under hypoxic conditions. We also simulate the flow dynamics of sickle RBCs treated with hydroxyurea (HU) and quantify the relative enhancement of hemodynamic performance of HU.

1This work was supported by the National Institutes of Health (NIH) Grant U01HL114476.
Patients Undergoing Cardiac Resynchronization Therapy, LORENZO ROSSINI, UC San Diego, P.

The local unsteady flow characteristics are calculated, fully resolving the flow field throughout the entire cardiac cycle. The Quemada regions. The present paper examines these indices in three aortic geometries obtained from patients whose aortas are deformed due to a genetic shear stress, oscillatory shear index, relative residence time and temporal wall shear stress gradients have been shown to identify plaque prone between aortic valve and anastomosis, and the aortic pressure acting against aortic valve opening.

The angle of incidence of the outflow graft critically influences the volume of recirculating flow are virtually anastomosed along the ascending aorta or subclavian artery of the patient-specific model at different positions and angles that ostia of the upper branches. The flow induced by the combination of VAD output through the graft anastomosed to the aorta and the limited outflow graft in the aorta plays a key role in the hemodynamics of Left Ventricle Assist Devices (LVAD), a medical device with a growing importance in the treatment of end-stage heart failure. We use a patient-specific computational model of the VAD and the ascending aorta to investigate the impact of VAD outflow graft configuration on the residence time and wall shear stresses along the ascending aorta and the ostia of the upper branches. The flow induced by the combination of VAD output through the graft anastomosed to the aorta and the limited cardiac output through intermittent opening of the aortic valve is studied to determine the nature of thrombogenic flow patterns. Outflow grafts are virtually anastomosed along the ascending aorta or subclavian artery of the patient-specific model at different positions and angles that are surgically informed. Detailed markers of thrombosis, such as cell residence time, wall shear stress, and shear stress gradients are analyzed and compared for the different configurations. The angle of incidence of the outflow graft critically influences the volume of recirculating flow between aortic valve and anastomosis, and the aortic pressure acting against aortic valve opening.

Optimization of the Outflow Graft Position and Angle in a Left Ventricular Assist Device, PATRICK MCGAH, Univ of Washington, ANTHONY PRISCO, Medical College of Wisconsin, JENNIFER BECKMAN, NAHUSH MOKADAM, CLAUDIUS MAHR, ALBERTO ALISEDA, Univ of Washington — The placement of the outflow graft in the aorta plays a key role in the hemodynamics of Left Ventricle Assist Devices (LVAD), a medical device with a growing importance in the treatment of end-stage heart failure. We use a patient-specific computational model of the VAD and the ascending aorta to investigate the impact of VAD outflow graft configuration on the residence time and wall shear stresses along the ascending aorta and the ostia of the upper branches. The flow induced by the combination of VAD output through the graft anastomosed to the aorta and the limited cardiac output through intermittent opening of the aortic valve is studied to determine the nature of thrombogenic flow patterns. Outflow grafts are virtually anastomosed along the ascending aorta or subclavian artery of the patient-specific model at different positions and angles that are surgically informed. Detailed markers of thrombosis, such as cell residence time, wall shear stress, and shear stress gradients are analyzed and compared for the different configurations. The angle of incidence of the outflow graft critically influences the volume of recirculating flow between aortic valve and anastomosis, and the aortic pressure acting against aortic valve opening.

Wall shear stress indicators in abnormal aortic geometries, LISA PRAHL WITTBERG, STEVIN VAN WYK, LASZLO FUCHS, KTH Mechanics, EPHRAIM GUTMARK, University of Cincinnati, IRIS GUTMARK-LITTLE, Cincinnati Children’s Hospital — Cardiovascular disease, such as atherosclerosis, occurs at specific locations in the arterial tree. Characterizing flow and forces at these locations is crucial to understanding the genesis of disease. Measures such as time average wall shear stress, oscillatory shear index, relative residence time and temporal wall shear stress gradients have been shown to identify plaque prone regions. The present paper examines these indices in three aortic geometries obtained from patients whose aortas are deformed due to a genetic pathology and compared to one normal geometry. This patient group is known to be prone to aortic dissection and our study aims to identify early indicators that will enable timely intervention. Data obtained from cardiac magnetic resonance imaging is used to reconstruct the aortic arch. The local unsteady flow characteristics are calculated, fully resolving the flow field throughout the entire cardiac cycle. The Quemada model is applied to account for the non-Newtonian properties of blood, an empirical model valid for different red blood cell loading. The impact of the deformed aortic geometries is analyzed to identify flow patterns that could lead to arterial disease at certain locations.

The Generation and Propagation of Arterial Murmurs from a Stenosed Artery: A Computational Study, CHI ZHU, JUNG-HEE SEO, HANI BAKHSHAEE, RAJAT MITTAL, Johns Hopkins University — Cardiac auscultation - the procedure of diagnosing cardiovascular conditions using the stethoscope - has been used effectively for over a hundred years but still, the flow mechanism(s) responsible for the generation of these murmurs, as well as the effect of intervening tissue on the propagation of these murmurs, is not well understood. In this study, a one-way coupled, hybrid approach is used to investigate the propagation of murmurs generated from the flow in a stenosed artery. Specifically, the flow in the modeled artery is solved by an incompressible Navier-Stokes solver with the immersed-boundary method. The structural wave propagation in the tissue is resolved by a high-order, linear viscoelastic wave solver, and a mathematical decomposition is applied to separate the compressional and shear component of the acoustic wave propagating through the tissue. The simulations suggest, somewhat counterintuitively, that the shear wave contributes a significant component to the signal picked up by a stethoscope, and that this component carries much of the information that characterizes the source of the murmur. The implications of this for cardiac auscultation and further modeling of hemocoacoustics are discussed. The effect of the stenosis severity and the flow pulsatility will also be investigated.

A Computational Chemo-Fluidic Modeling for the Investigation of Patient-Specific Left Ventricle Thrombogenesis, RAJAT MITTAL, JUNG HEE SEO, THURA ABD, RICHARD T. GEORGE, Johns Hopkins University — Patients recovering from myocardial infarction (MI) are considered at high-risk for cardioembolic stroke due to the formation of left ventricle thrombus (LVT). The formation of LVT is the result of a complex interplay between the fluid dynamics inside the ventricle and the chemistry of coagulation, and the role of LV flow pattern on the thrombogenesis was not well understood. The previous computational study performed with the model ventricles suggested that the local flow residence time is the key variable governing the accumulation of coagulation factors. In the present study, a coupled, chemo-fluidic computational modeling is applied to the patient-specific cases of infracted ventricles to investigate the interaction between the LV hemodynamics and thrombogenesis. In collaboration with the Johns Hopkins hospital, patient-specific LV models are constructed using the multi-modality medical imaging data. Blood flow in the left ventricle is simulated by solving the incompressible Navier-Stokes equations and the biochemical reactions for the thrombus formation are modeled with convection-diffusion-reaction equations. The formation and deposition of key coagulation chemical factors are then correlated with the hemodynamic flow metrics to explore the biophysics underlying LVT risk.
1:55PM R24.00006 Ansys Fluent versus Sim Vascular for 4-D patient-specific computational hemodynamics in renal arteries1, AVINASH MUMBARADDI, HUIDAN (WHITNEY) YU, Indiana University-Purdue University Indianapolis, ALAN SAWCHUK, MICHAEL DALSING, School of Medicine, Indiana University — The objective of this clinical-need-driven research is to investigate the effect of renal artery stenosis (RAS) on the blood flow and wall shear stress in renal arteries through 4-D patient-specific computational hemodynamics (PSCH) and search for possible critical RASs that significantly alter the pressure gradient across the stenosis by manually varying the size of RAS from 50% to 95%. The identification of the critical RAS is important to understand the contribution of RAS to the overall renal resistance thus appropriate clinical therapy can be determined in order to reduce the hypertension. Clinical CT angiographic data together with Doppler Ultrasound images of an anonymous patient are used serving as the required inputs of the PSCH. To validate the PSCH, we use both Ansys Fluent and Sim Vascular and compare velocity, pressure, and wall-shear stress under identical conditions.

1Renal Imaging Technology Development Program (RITDP) Grant

2:08PM R24.00007 Numerical simulation of hemorrhage in human injury , KWITAE CHONG, CHENFANFU JIANG, ANAND SANTHANAM, PEYMAN BENHARASH, JOSEPH TERAN, JEFF ELDREDGE, University of California, Los Angeles — Smoothed Particle Hydrodynamics (SPH) is adapted to simulate hemorrhage in the injured human body. As a Lagrangian fluid simulation, SPH uses fluid particles as computational elements and thus mass conservation is trivially satisfied. In order to ensure anatomical fidelity, a three-dimensional reconstruction of a portion of the human body —here, demonstrated on the lower leg—is sampled as skin, bone and internal tissue particles from the CT scan image of an actual patient. The injured geometry is then generated by simulation of ballistic projectiles passing through the anatomical model with the Material Point Method (MPM) and injured vessel segments are identified. From each such injured segment, SPH is used to simulate bleeding, with inflow boundary condition obtained from a coupled 1-d vascular tree model. Blood particles interact with impermeable bone and skin particles through the Navier-Stokes equations and with permeable internal tissue particles through the Brinkman equations. The SPH results are rendered in post-processing for improved visual fidelity. The overall simulation strategy is demonstrated on several injury scenarios in the lower leg.

2:21PM R24.00008 A Computational Approach to Model Vascular Adaptation During Chronic Hemodialysis: Shape Optimization as a Substitute for Growth Modeling1, S. M. JAVID MAHMOUDADEH AKHERAT, MICHAEL BOGHOSSIAN, KEVIN CASSEL, Illinois Institute of Technology, MARY HAMMES, University of Chicago — End-stage-renal disease patients depend on successful long-term hemodialysis via vascular access, commonly facilitated via a Brachiocephalic Fistula (BCF). The primary cause of BCF failure is Cephalic Arch Stenosis (CAS). It is believed that low Wall Stress (WSS) regions, which occur because of the high flow rates through the natural bend in the cephalic vein, create hemodynamic circumstances that trigger the onset and development of Intimal Hyperplasia (IH) and subsequent CAS. IH is hypothesized to be a natural effort to reshape the vessel, aiming to bring the WSS values back to a physiologically acceptable range. We seek to explore the correlation between regions of low WSS and subsequent IH and CAS in patient-specific geometries. By utilizing a shape optimization framework, a method is proposed to predict cardiovascular adaptation that could potentially be an alternative to vascular growth and remodeling. Based on a functional objective that seeks after the vessel shape in such a way as to readjust the WSS towards normal physiological range, CFD and shape optimization are then coupled to investigate whether the optimal shape evolution is correlated with actual patient-specific geometries therefrom. 

1Supported by the National Institute of Diabetes and Kidney Diseases of the National Institutes of Health (R01 DK0769).

2:34PM R24.00009 Intraventricular filling under increasing left ventricular wall stiffness and heart rates1, MILAD SAMAEE, HONG KUAN LAI, JOSEPH SCHOVANEC, ARVIND SANTHANAKRISHNAN, Oklahoma State University, SHERIF NAGUEH, Houston Methodist Hospital — Heart failure with normal ejection fraction (HFNEF) is a clinical syndrome that is prevalent in over 50% of heart failure patients. HFNEF patients show increased left ventricle (LV) wall stiffness and clinical diagnosis is difficult using ejection fraction (EF) measurements. We hypothesized that filling vortex circulation strength would decrease with increasing LV stiffness irrespective of heart rate (HR). 2D PIV and hemodynamic measurements were acquired on LV physical models of varying wall stiffness under resting and exercise HRs. The LV models were comparatively tested in vivo in vitro flow circuit consisting of a two-element Windkessel model driven by a piston pump. The stiffer LV models were tested in comparison with the least stiff baseline model without changing pump amplitude, circuit compliance and resistance. Increasing stiffness at resting HR resulted in diminishing cardiac output without lowering EF below 50% as in HFNEF. Increasing HR to 110 bpm in addition to stiffness resulted in lowering EF to less than 50%. The circulation strength of the intraventricular filling vortex diminished with increasing stiffness and HR. The results suggest that filling vortex circulation strength could be potentially used as a surrogate measure of LV stiffness.

1This research was supported by the Oklahoma Center for Advancement of Science and Technology (HR14-02).

2:47PM R24.00010 Measurements of flow structure interaction in a plaqued artificial artery using an index matched flow facility, AKASH JAIN, LARRY BROCK, JIAN SHENG, Texas Tech Univ. — The aim of the experiment is to study the flow structure interaction in an arterial model with a simulated plaque inside a closed loop index matched pulsatile flow facility. The test section is 24.5 inches long 6 inches wide. The experimental models are compliant polymer (PDMS) tubes having an outer diameter of 9 mm and a wall thickness of 1 mm. The plaque on the models are simulated by means of a radially asymmetric bump. Both flow and polymeric structures are doped with different particles and imaged with Particle Image Velocimetry (PIV) method. To minimize the optical distortion near liquid solid interface, the facility is fully index matched with NaI at 40% by weight. A suite of analysis procedures quantifying complex interactions including solid-fluid phase separation, near wall flow analysis, and wall shear stress approximation as well as wall deformation quantification, have been developed and applied to study the healthy and plaqued artificial arteries in steady and pulsatile flow conditions. 3D ensemble velocity fields, wall shear stress distributions and corresponding strain deformations will be presented.

3:00PM R24.00011 Right Heart 4DMRI Flow Visualization in Normal and Hypertensive subjects, JEAN HERTZBERG, JAMES BROWNING, University of Colorado Boulder, BRETT FENSTER, National Jewish Health and University of Colorado Denver, JOYCE SCHROEDER, University of Colorado Denver — Recent advances in time-resolved 3D cardiac magnetic resonance imaging (4DMRI) have allowed for the 3-dimensional characterization of blood flow in the right ventricle (RV) and right atrium (RA). In this talk, an overview of a large, ongoing, multi-disciplinary investigation of 4D right heart hemodynamics in normal and pathologic patients is given, as well as lessons learned from 4DMRI cardiac research. Time-resolved visualization techniques for understanding and communicating complex right heart flow structures throughout the cardiac cycle are presented. Finally, a qualitative visual comparison of 3D flow structures in the vena cava, RA, and RV between healthy subjects and pulmonary hypertensive patients is presented.
Recent advances in cardiac magnetic resonance imaging (CMR) have allowed for the 3-dimensional characterization of blood flow in the right ventricle (RV) and right atrium (RA). In this study, we investigate and quantify differences in the characteristics of coherent rotating flow structures (vortices) in the RA and RV between subjects with right ventricular diastolic dysfunction (RVDD) and normal controls. Fifteen RVDD subjects and 10 age-matched controls underwent same day 3D time resolved CMR and echocardiography. Echocardiography was used to determine RVDD stage as well as pulmonary artery systolic pressure (PASP). CMR data was used for RA and RV vortex quantification and visualization during early ventricular diastole and the results are compared between healthy subjects and those with RVDD. The resulting trends are discussed and hypotheses are presented regarding differences in vortex characteristics between healthy and RVDD subjects cohorts.

Tuesday, November 24, 2015 12:50PM - 3:26PM –

12:50PM R25.00001 DNS and Modeling of Turbulent Gas-Liquid Channel Flows
GRETAR TRYGGVASON, MING MA, JIACAI LU, Univ of Notre Dame — DNS studies of gas-liquid flows in vertical turbulent channels are presented. Results from a simulation of a pressure driven turbulent channel flow with a friction Reynolds number of 500 where a large number of bubbles of different sizes are injected at time zero, shows that small bubbles quickly migrate to the wall, but the flow takes much longer to adjust to the new bubble distribution. The evolution of turbulent statistic and the void fraction distribution is examined, including area concentration and the components of the area tensor. Another series of simulations of bubbles injected into turbulent channel flow, where the bubbles are allowed to coalesce and break apart, is also presented. For high enough surface tension all the bubbles coalesce into one large slug, but as the surface tension is reduced, large enough bubbles break up and the flow eventually reaches an approximate equilibrium where coalescence is matched by breakup. The resulting state generally contains bubbles with a distribution of sizes. The various quantities characterizing the flow are followed over time and their dependency of the flow parameters examined. Preliminary attempts to model the flow using a set of averaged equations, using closure relations derived from the DNS data are discussed.

1Research supported by DOE (CASL)

1:03PM R25.00002 Nucleation of Super-Critical Carbon Dioxide in a Venturi Nozzle
DORRIN JARRAHBASHI, SANDEEP PIDAPARTI, DEVESH RANJAN, Georgia Institute of Technology — The supercritical carbon dioxide (S-CO2) Brayton cycle combines the primary advantages of the ideal Brayton and Rankine cycles by utilizing CO2 above its critical pressure. In addition to single phase and small back work ratios, supercritical fluids offer other advantages, e.g. heat transfer augmentation and low specific volume. Pressure reduction at the entrance of the compressor may cause homogeneous nucleation, vapor production, and collapse of bubbles due to operation near the saturation conditions. Transient behavior of the flow after nucleation may cause serious issues in operation of the cycle and affect the materials used in design. The flow of S-CO2 through a venturi nozzle near the critical point has been studied. A transient compressible 3D Navier-Stokes solver, coupled with continuity, and energy equation has been used. Developed FIT libraries based on a piecewise biquintic spline interpolation of Helmholtz energy have been integrated with OpenFOAM to model S-CO2 properties. The mass fraction of vapor created in the venturi has been calculated using homogeneous equilibrium model (HEM). The flow conditions that lead to nucleation have been investigated. The sensitivity of nucleation to the inlet pressure and temperature, flow rate, and venturi profile has been shown.

1:16PM R25.00003 Effect of different wall boundary conditions on the numerical simulation of bubbling fluidized beds
MOHAMMAD REZA HAGHGGOO, DONALD J. BERGSTROM, Department of Mechanical Engineering, University of Saskatchewan, RAYMOND J. SPITERI, Department of Computer Science, University of Saskatchewan — There are distinct wall boundary conditions proposed in the literature for the particulate phase in the context of a continuum description of gas-particle flows. It is not yet clear how these different wall boundary conditions affect the simulated flow behavior, nor is it clear which are the most realistic. To investigate this issue, an Eulerian–Eulerian two-fluid model was used to investigate the effect of different particle-phase wall boundary conditions on the numerical prediction of bubbling/slugging gas-particle fluidized beds. Because the bed dynamics are strongly influenced by the motion of the bubbles, the impact of wall boundary conditions on the bubble statistics was examined specifically. In addition, the averaged field variables, such as the particle velocity, were compared to published experimental measurements. The comparison shows good agreement between the numerical results, generated by the Mfix code, and their experimental counterparts. It is found that the particle wall boundary condition does play a significant role in predicting the flow behavior. However, it appears that the influence of the wall boundary conditions is more significant for the instantaneous flow variables and bubble statistics than for the averaged quantities.

1:29PM R25.00004 Investigation of Gas Holdup in a Vibrating Bubble Column
SHAHROUZ MOHAGHEGHIAN, BRIAN ELBING, Oklahoma State University — Synthetic fuels are part of the solution to the world’s energy crisis and climate change. Liquefaction of coal during the Fischer-Tropsch process in a bubble column reactor (BCR) is a key step in production of synthetic fuel. It is known from the 1960’s that vibration improves mass transfer in bubble column. The current study experimentally investigates the effect that vibration frequency and amplitude have on gas holdup and bubble size distribution within a bubble column. Air (disperse phase) was injected into water (continuous phase) through a needle shape injector near the bottom of the column, which was open to atmospheric pressure. The air volumetric flow rate was measured with a variable area flow meter. Vibrations were generated with a custom-made shaker table, which oscillated the entire column with independently specified amplitude and frequency (0-30 Hz). Geometric dependencies can be investigated with four cast acrylic columns with aspect ratios ranging from 4.36 to 2.5, and injector needle internal diameters between 0.32 and 1.59 mm. The gas holdup within the column was measured with a flow visualization system, and a PIV system was used to measure phase velocities. Preliminary results for the non-vibrating and vibrating cases will be presented.
1:42PM R25.00005 Turbulent hydraulic jumps: Effect of Weber number and Reynolds number on air entrainment and micro-bubble generation\(^1\), MILAD MORTAZAVI, ALI MANI, Center for Turbulence Research, Stanford University — Air entrainment in breaking waves is a ubiquitous and complex phenomenon. It is the main source of air transfer from atmosphere to the oceans. Furthermore, air entrainment due to ship-induced waves contributes to bubbly flows in ship wakes and also affect their performance. In this study, we consider a turbulent hydraulic jump as a canonical setting to investigate air entrainment due to turbulence-wave interactions. The flow has an inlet Froude number of 2.0, while three different Weber numbers (We = 1820, 729, 292), and two different Reynolds numbers (Re = 11000, 5500) based on the inlet height and inlet velocity are investigated. Air entrainment is shown to be very sensitive to the We number, while Re number has a minor effect. Wave breaking and interface collisions are significantly reduced in the low Weber number cases. As a result, micro-bubble generation is significantly reduced with decreasing Weber number. Vortex shedding events are observed to emerge at the toe of the jump in all of the cases. For high Weber number regimes, shedding of vortices is accompanied by engulfment of air pockets into the jump in a periodic manner, while for lower Weber number regimes such events are significantly suppressed. Reynolds number is shown to have a negligible effect on the air entrainment, wave breaking and micro-bubble generation, contrary to the previous assumptions in other studies.

\(^1\)Supported by ONR

1:55PM R25.00006 A Study of the Influence of Numerical Diffusion on Gas-Solid Flow Predictions in Fluidized Beds, RONA K CHANDRIZ, REZA SHEIKHI, Northeastern University — In this work, an investigation is made of the influence of numerical diffusion on the accuracy of gas-solid flow predictions in fluidized beds. This is an important issue particularly in bubbling fluidized beds since numerical error greatly affects the dynamics of bubbles and their associated mixing process. A bed of coal (classified as Geldart A) is considered which becomes fluidized as the velocity of nitrogen stream into the reactor is gradually increased. The fluidization process is simulated using various numerical schemes as well as grid resolutions. Simulations involve Eulerian-Eulerian two-phase flow modeling approach and results are compared with experimental data. It is shown that higher order schemes equipped with flux limiter give favorable prediction of bubble and particle dynamics and hence, the mixing process within the reactor. The excessive numerical diffusion associated with lower order schemes results in unrealistic prediction of bubble shapes and bed height. Comparison is also made of computational efficiency of various schemes. It is shown that the Monotonized Central scheme with down wind factor results in the shortest simulation time because of its efficient parallelization on distributed memory platforms.

2:08PM R25.00007 Effect of cavitation in high-pressure direct injection, BAHMAN ABOUL-HASANZADEH, ERIC JOHNSEN, University of Michigan — As we move toward higher pressures for Gasoline Direct Injection and Diesel Direct Injection, cavitation becomes an important issue. To better understand the effect of cavitation on the nozzle flow and primary atomization, we use a high-order accurate Discontinuous Galerkin approach using multi-GPU parallelism to simulate the compressible flow inside and outside the nozzle. Phase change is included using the six-equations model. We investigate the effect of nozzle geometry on cavitation inside the injector and on primary atomization outside the nozzle.

2:21PM R25.00008 Measurements of Jet Effect on a Ventilated Cavity\(^1\), IVAN KIRSCHNER, Applied Physical Sciences Corp, MICHAEL MOENY, MICHAEL KRANE, MICHAEL KINZEL, Applied Research Laboratory, Penn State University — An experimental study was performed to evaluate some of the claims of Paryshev (2006) regarding changes to ventilated cavity behavior caused by the interaction of a jet with the cavity closure region. The experiments, conducted in the 1.22m diameter Garfield Thomas Water Tunnel, were performed for a 0.0222 EDD to tunnel diameter ratio, Fr = 14.5 and 26.2. The model consisted of a converging-section nozzle mounted to the base of a 27.9mm 37\(^\circ\) cone cavitator placed on the tunnel centerline at the end of a 138.4mm long streamlined strut. A ventilated cavity was formed over the model. Then an air jet, issuing from a converging nozzle, was initiated. Changes to cavity behavior were quantified in terms of cavitation number, thrust-to-drag ratio, and stagnation pressure ratio at the jet nozzle. The results show that, while the overall trends predicted by Paryshev were observed, the data did not fully collapse, suggesting that many of the effects neglected by Paryshev’s model have measurable effect.

\(^1\)Acknowledgment support from the Office of Naval Research.

2:34PM R25.00009 LES of turbulent cavitation\(^1\), ASWIN GNANASKANDAN, KRISHNAN MAHESH, University of Minnesota — Large Eddy Simulation is employed to study two unsteady turbulent cavitating flows: cyclic cavitation over a cylinder and sheet to cloud cavitation over a wedge. A homogeneous mixture model is used to treat the mixture of water and water vapor as a compressible fluid. The governing equations are solved using a novel predictor-corrector method (Gnanaskandan and Mahesh, Int. Journal of Multiphase Flow, 2015, 70:22–34). Cavitating flow over a cylinder at Reynolds number (based on cylinder diameter and free stream velocity) Re = 3900 and cavitation number \(\sigma = 1.0\) is simulated and the wake characteristics are compared to the single-phase results at the same Reynolds number. It is observed that cavitation suppresses turbulence in the near wake and delays three-dimensional breakdown of the vortices. The role of cavitation-induced vorticity dilatation in suppressing vortex shedding frequency is discussed. Next, cavitating flow over a wedge at Re = 200,000 (based on wedge height and inlet velocity) and \(\sigma = 2.1\) is presented. The mean void fraction profiles obtained are compared to experiment and good agreement is obtained. Cavity auto–oscillation is observed, where the sheet cavity breaks up into a cloud cavity periodically. The Strouhal number corresponding to auto-oscillation also agrees well with the experiment. The process of transition from sheet to cloud cavitation will be discussed.

\(^1\)This work is supported by the Office of Naval Research

2:47PM R25.00010 Cavitation dynamics on a NACA0015 hydrofoil using time resolved X-ray densitometry\(^1\), HARISH GANESH, JULIANA WU, STEVEN CECcio, University of Michigan — Recent investigations of partial cavitation have shown that the transition from stable to shedding cavities over a range of attack angles and cavitation numbers. The role of attack angle is of particular interest, since it is related to the pressure gradient at cavity enclosure, and can lead to the formation of stronger reentrant flows. The relative importance of reentrant liquid flow and bubbly shock wave propagation will be discussed.

\(^1\)This work is supported by Office of Naval Research
of cavitating flow in the wake of a circular cylinder. The presence of developed cavitantion can alter the underlying vortical flow. In this study, cavitating dynamics in the wake of a circular cylinder is examined in order to determine the relationship between the void fraction in the cavity wake and the resulting modification to the flow compared to the non-cavitating flow. Cavitation in the wake of a cylinder is investigated using high-speed video cameras and cinematographic X-ray densitometry. Using synchronized top and side views from high-speed video cameras, the morphology and extent of the cavities forming on the wake of the circular cylinder is studied for a range of cavitation numbers, at a Reynolds number of $1 \times 10^5$, which lies at the transition region between sub-critical to critical regime of wake transitions. The time resolved and mean X-ray densitometry based void fraction of the spanwise and plan view averaged flow field will be related to the vortex dynamics in an attempt to understand the role of vapor production in the observed dynamics.

The formation process is driven by bubble coalescence, whereas its collapse is related to the pressure difference across the supercavity interface at its rear portion. Further, we examine the relationship between ventilation hysteresis, supercavity closures and air entrainment requirements for supercavity formation and sustenance under steady and unsteady flow conditions. These observations are directly related to the internal flows inside the supercavity.

We gratefully acknowledge support for this work provided by NSF CBET Grant 1134500.

1:16PM R26.00003 Actuation of interfacial waves in oil-water flows, KYEONG PARK, WEHE-LIYE WEHELIYE, MAXIME CHINAUD, PANAGIOTA ANGELI, Department of Chemical Engineering, University College London, Torrington Place, London, WC1E 7JE, JAMES PERCIVAL COLLABORATION. OMAR. K. MATAR COLLABORATION — Droplet detachment from interfacial waves in two-phase flows has pulled in noteworthy exploration interest. In order to examine this phenomenon experimentally and empower quantitative estimation, it is important to spatially confine the drop formation. In the present study, a cylinder, located close to the inlet of the test section and perpendicular to the direction of the flow, is placed in a two-phase stratified oil-water pipe flow. The introduction of this cylinder actuated interfacial waves and move from stratified to dispersed flow pattern. High speed visualisation and Particle Image Velocimetry (PIV) measurement are utilized to investigate the flow pattern maps of the two-phase flow and the velocity fields in the wake of the cylinder, respectively. These results will be compared with previous experimental studies.

1We gratefully acknowledge support for this work provided by NSF CBET Grant 1134500.

1Department of Chemical Engineering South Kensington Campus Imperial College London SW7 2AZ

2Department of Chemical Engineering South Kensington Campus Imperial College London SW7 2AZ

12:50PM R25.00001 Transport of temperature-velocity covariance in gas-solid flow and its relation to the axial dispersion coefficient, SHANKAR SUBRAMANIAM, BO SUN, Iowa State University — The presence of solid particles in a steady laminar flow generates velocity fluctuations with respect to the mean fluid velocity that are termed pseudo-turbulence. The level of these pseudo-turbulent velocity fluctuations has been characterized in statistically homogeneous fixed particle assemblies and freely evolving suspensions using particle-resolved direct numerical simulation (PR-DNS) by Mehrabadi et al. (JFM, 2015), and it is found to be a significant contribution to the total kinetic energy associated with the flow. The correlation of these velocity fluctuations with temperature (or a passive scalar) generates a flux term that appears in the transport equation for the average fluid temperature (or average scalar concentration). The magnitude of this transport of temperature-velocity covariance is quantified using PR-DNS of thermally fully developed flow past a statistically homogeneous fixed assembly of particles, and the budget of the average fluid temperature equation is presented. The relation of this transport term to the axial dispersion coefficient (Brenner, Phil. Trans. Roy. Soc. A, 1980) is established. The simulation results are then interpreted in the context of our understanding of axial dispersion in gas-solid flow.

1NSF CBET 1336941


12:50PM R25.00012 On the relationship between air entrainment, internal flows and closure mechanism in a ventilated supercavity, ASHISH KARN, ROGER ARNDT, JIARONG HONG, University of Minnesota — An understanding of underlying physics behind ventilation demand is critical for the operation of underwater vehicles based on ventilated supercavitation for a number of reasons viz. gas entrainment requirements for cavity formation and sustenance. The prior studies on the ventilation demand have reported that the gas entrainment requirement to form a supercavity is substantially larger than that needed to sustain it. This phenomenon, known as ventilation hysteresis, is particularly important from the viewpoint of reduction in gas requirements. However, little physical insights into this phenomenon has yet been provided. In this study, systematic investigations are conducted into ventilation hysteresis with respect to the formation and collapse behaviors of ventilated supercavities. It is suggested that the supercavity formation process is driven by bubble coalescence, whereas its collapse is related to the pressure difference across the supercavity interface at its rear portion. Further, we examine the relationship between ventilation hysteresis, supercavity closures and air entrainment requirements for supercavity formation and sustenance under steady and unsteady flow conditions. These observations are directly related to the internal flows inside the supercavity.

12:50PM R25.00011 X-ray densitometry based void fraction flow field measurements of cavitating flow in the wake of a circular cylinder, TIEZHI SUN, Harbin Institute of Technology, HARIKGANESH, STEVEN CECCIO, University of Michigan — At sufficiently low cavitation number, the wake vortices behind bluff objects will cavitate. The presence of developed cavitantion can alter the underlying vortical flow. In this study, cavitating dynamics in the wake of a circular cylinder is examined in order to determine the relationship between the void fraction in the cavity wake and the resulting modification to the flow compared to the non-cavitating flow. Cavitation in the wake of a cylinder is investigated using high-speed video cameras and cinematographic X-ray densitometry. Using synchronized top and side views from high-speed video cameras, the morphology and extent of the cavities forming on the wake of the circular cylinder is studied for a range of cavitation numbers, at a Reynolds number of $1 \times 10^5$, which lies at the transition region between sub-critical to critical regime of wake transitions. The time resolved and mean X-ray densitometry based void fraction of the spanwise and plan view averaged flow field will be related to the vortex dynamics in an attempt to understand the role of vapor production in the observed dynamics.

3:00PM R25.00011 X-ray densitometry based void fraction flow field measurements of cavitating flow in the wake of a circular cylinder, TIEZHI SUN, Harbin Institute of Technology, HARIKGANESH, STEVEN CECCIO, University of Michigan — At sufficiently low cavitation number, the wake vortices behind bluff objects will cavitate. The presence of developed cavitantion can alter the underlying vortical flow. In this study, cavitating dynamics in the wake of a circular cylinder is examined in order to determine the relationship between the void fraction in the cavity wake and the resulting modification to the flow compared to the non-cavitating flow. Cavitation in the wake of a cylinder is investigated using high-speed video cameras and cinematographic X-ray densitometry. Using synchronized top and side views from high-speed video cameras, the morphology and extent of the cavities forming on the wake of the circular cylinder is studied for a range of cavitation numbers, at a Reynolds number of $1 \times 10^5$, which lies at the transition region between sub-critical to critical regime of wake transitions. The time resolved and mean X-ray densitometry based void fraction of the spanwise and plan view averaged flow field will be related to the vortex dynamics in an attempt to understand the role of vapor production in the observed dynamics.
1:29PM R26.00004 Flow development investigation of concentrated unstable oil-water dispersions in turbulent pipe flows, VICTOR VOULGAROPOULOS, WEHELIYE WEHELIYE, MAXIME CHINALD, PAAAGIOTA ANGELI, Department of Chemical Engineering, University College London, Torrington Place, London, WC1E 7JE, KAROLINA IOANNOU COLLABORATION - This study explores the separation characteristics of unstable oil-water dispersed flows in pipes. The test section is a 7 m long acrylic pipe with a 37mm ID and the fluids used are tap water and an Exxsol oil (6.6cSt). An inlet system with more than a thousand capillary tubes of 1mm ID is implemented to actuate highly concentrated dispersions for a wider range of flow rates. High speed imaging combined with ring conductivity probes and pressure transducers are implemented in several axial positions along the pipe to study the flow development. Phase distribution and continuity are measured in the pipe cross-section and drop size information is acquired by high frequency dual impedance probes. The coalescence and sedimentation dynamics of the concentrated dispersions and the development of separate layers downstream the pipe are investigated. The experimental results are coupled with theoretical and semi-empirical models in an effort to predict the separation properties of the highly concentrated dispersed flows.

1Chevron Energy Technology, Houston, USA

1:42PM R26.00005 Droplet Size Distributions Resulting form Entrainment of Surface Oil Slick by Breaking Waves, CHENG LI, JOSEPH KATZ, Johns Hopkins University - A spectrum of droplet sizes, ranging from submicron to several millimeters, is generated by breaking waves impinging on an oil slick. Their size distribution is crucial for modeling the fate of oil spill, and understanding the underlying flow physics. Digital holography microscopy (DHM) is used for measuring the droplet size distributions at high resolution (1.1 µm/pixel), and at varying temporal scale, from the initial plunging phase (seconds) to long term (hours). The time-resolved DHM data is acquired simultaneously with high speed visualizations of the breakup and large scale features of the entrainment process. Experimental conditions include: (i) plunging and spilling breakers with wave heights of 28.8, 24.9, 22.28 cm; (ii) crude oil (MC252 surrogate), and oil premixed with dispersants (Corexit-9500A) giving two order of magnitude range of water-oil interfacial tension; (iii) Crude, fish, and motor oils with viscosity of 9.4, 63.1 and 306.5 cst, respectively. Shortly after entrainment of crude oil, the droplet radius distribution is bimodal, with a primary peak in the 0.25-10 µm range, and a secondary peak at 200-250 µm. Adding dispersants reduces the latter to 0.16 µm. The drastic reduction in interfacial tension upon introduction of dispersants increases the primary peak, and causes short term micro threading. The Secondary peaks dampen within seconds, as the larger droplets rise, whereas the primary peaks are sustained for longer periods.

1Supported by Gulf of Mexico Research Initiative (GoMRI)

1:55PM R26.00006 Modelling the Hydrodynamics and Transport in Multiphase Microreactors, LU YANG, Department of Chemical Engineering, MIT, YANXIANG SHI, BASF Corporation, MILAD ABOLHASANI, KLAVS JENSEN, Department of Chemical Engineering, MIT - Multiphase flow is prevalent in a variety of industrial applications, but the extent of these processes is often limited by the innate mass transfer resistance across phase boundaries. Microscale multiphase systems, owing to their reduced characteristic length scales, increase specific interfacial areas and unique hydrodynamic patterns, can significantly enhance the rate of mass transfer, thereby improving the efficiency of multiphase processes. However, many uncertainties still remain in the prediction of multiphase hydrodynamics and scalar transport on the microscale, primarily due to the complexity of the multiphase flow. In this work, we elucidate the mechanism of mass transfer enhancement in microscale multiphase flows, a computational fluid dynamic (CFD) model using the volume-of-fluid (VOF) method is developed, and the method is validated with experiments. By introducing a scalar transport equation with sink/source terms using the one-fluid formulation, we enable the simultaneous capturing of multi-phase hydrodynamics, mass transfer and reactions. In tandem with the numerical simulations, we also perform mass transfer analysis of multiphase flows based on the penetration theory and a two-stage theory, which further examines the mechanism of mixing enhancement in multiphase flow, and reveals a two-fold increase in mass transfer coefficients in the microreactors compared to conventional multiphase contactors.

2:08PM R26.00007 Linear stability analysis and direct numerical simulation of two layer channel flow, KIRTI SAHU, Indian Institute of Technology Hyderabad, India, RAMA GOVINDARAJAN, Tata Institute of Fundamental Research Narsingi, Hyderabad, India, MANOJ TRIPATHI, Indian Institute of Technology Hyderabad, India - We study the stability of two fluid flow through a plane channel at Reynolds numbers of a hundred to a thousand. The two fluids have the same density but different viscosities. The fluids, when miscible, are separated from each other by a mixed layer of small but finite thickness, across which viscosity changes from that of one fluid to that of the other. When immiscible, the interface is sharp. Our study spans a range of Schmidt numbers, viscosity ratios and location and plane thickness of the mixed layer. Our two-dimensional linear stability results predict well the behaviour displayed by our three-dimensional direct numerical simulations at early times. In both linear and non-linear regimes, the miscible flow is more unstable than the corresponding immiscible one, and the miscible flow breaks spanwise symmetry more readily to go into three-dimensionality. We show that the miscible flow over our range of parameters is always significantly more unstable than the corresponding immiscible case.

2:21PM R26.00008 Flow of two immiscible fluids in a periodically constricted tube: Transitions to stratified, segmented, churn, spray or segregated flow, JOHN TSAMOPOULOS, DIMITRIS FRAGGEDAKIS, YIANNIS DIMAKOPOULOS, Univ of Patras - We study the flow of two immiscible, Newtonian fluids in a periodically constricted tube driven by a constant pressure gradient. Our Volume-of-Fluid algorithm is used to solve the governing equations. First the code is validated by comparing its predictions to previously reported results for stratified and pulsing flow. Then it is used to capture accurately all the significant topological changes that take place. Initially, the fluids have a core-annular arrangement, which is found to either remain the same or change to a different arrangement, depending on the fluid properties, the pressure driving the flow or the flow geometry. The flow-patterns that appear are the core-annular, segmented, churn, spray and segregated flow. The predicted scalings near pinching of the core fluid concur with similarity predictions and earlier numerical results (Cohen et al. (1999)). Flow-pattern maps are constructed in terms of the Reynolds and Weber numbers. Our results provide deeper insights in the mechanism of the pattern transitions and are in agreement with previous studies on core-annular flow (Kouris & Tsamopoulos (2001 & 2002)), segmented flow (Luc & Sherwood (2009)) and churn flow (Bai et al. (1992)).

1GSRT of Greece through the program “Excellence” (Grant No. 1918, entitled “FilCoMicrA”)
2:34PM R26.00009 Reduced order modelling of counter-current two-layer flows\(^1\).\(^2\) GIANLUCA LAVALLE, MATHIEU LUCQUIAUD, PRASHANT VALLURI, The University of Edinburgh — The dynamics of two-layer flows has a great impact on absorption units of carbon-capture retrofits, since the wavy interface plays a crucial role on the transfer between the two fluids. Studying those flows by a direct numerical simulation (DNS) strategy results in a high computational cost requiring parallel computation. As an alternative approach, we present a reduced order model: the liquid film is computed with depth-integrated equations, and the coupling with the top phase is obtained by means of the Arbitrary Lagrangian-Eulerian (ALE) technique, according to which the grid follows the interface position. We study counter-current two-layer channel flows with a moderate density ratio, focusing on loading and flooding regimes, whose complete description is a central issue for many chemical applications. Also, we investigate the influence of flow rate and pressure gradient on the interface dynamics. Speed and growth rate of linear waves match with the Orr-Sommerfeld theory and our Level-Set DNS, and non-linear wave profiles agree with DNS. Finally, our model is tested with complex gas velocity profiles of cross-flow absorbers.

\(^1\)EPSRC grant No. EP/M001482/1

2:47PM R26.00010 Two-Phase Flow Hydrodynamics in Superhydrophobic Channels, KIMBERLY STEVENS, JULIE CROCKETT, DANIEL MAYNES, BRIAN IVERSON, Brigham Young University — Superhydrophobic surfaces promote drop-wise condensation and droplet removal leading to the potential for increased thermal transport. Accordingly, great interest exists in using superhydrophobic surfaces in flow condensing environments, such as power generation and desalination. Adiabatic air-water mixtures were used to gain insight into the effect of hydrophobicity on two-phase flows and the hydrodynamics present in flow condensation. Pressure drop and onset of various flow regimes in hydrophilic, hydrophobic, and superhydrophobic mini (0.5 \times 10 \text{mm}) channels were explored. Data for air-water mixtures with superficial Reynolds numbers from 20-200 and 250-1800, respectively, were obtained. Agreement between experimentally obtained pressure drops and correlations in literature for the conventional smooth control surfaces was better than 20 percent. Transitions between flow regimes for the hydrophobic and hydrophilic channels were similar to commonly recognized flow types. However, the superhydrophobic channel demonstrated significantly different flow regime behavior from conventional surfaces including a different shape of the air slugs, as discussed in the presentation.

3:00PM R26.00011 Experimental results and a self-consistent model of evaporation and high heat flux extraction by evaporating flow in a micro-grooved blade\(^1\), REZA MONAZAMI, MEHDI SAADAT, JIANZHONG ZHU, HOSSEIN HAJ-HARIRI, University of Virginia — The problem of evaporation from a vertical micro-grooved blade heated from above is investigated. The required superheat to handle the incoming flux is calculated using the results of the study by Monazami and Haj-Hariri (2012). The relation between the applied heat flux, dry-out length and the maximum equilibrium temperature for several geometries and working fluids are studied. Furthermore, a computational study of the evaporating meniscus is conducted to evaluate the evaporation rates and dissipated heat flux at the liquid-vapor interface. The computational study accounts for the flow and heat transfer in both liquid and vapor phases. The results of this study indicate that the micro-grooved structure can dissipate heat fluxes as high as 10MW/m\(^2\) for superheats as low as 5 degrees Kelvin. Experiments are conducted to verify the computational and analytical results. The findings of this work are applicable to the design of thermal management systems for high heat flux applications. Ref. Monazami, R. and Haj-Hariri, H. A mathematically-consistent formulation for evaporation of menisci in microchannels. American Physical Society, 65th Annual DFD Meeting, San Diego, CA, Nov 1820, 2012.

\(^1\)Partially supported by the MAXNET Energy Partnership (Max Planck Institute and UVA)

3:13PM R26.00012 The lifetime of evaporating dense sprays, ALOIS DE RIVAS, EMMANUEL VILLERMAUX, Aix-Marseille University — We study the processes by which a set of nearby liquid droplets (a spray) evaporates in a gas phase whose relative humidity (vapor concentration) is controlled at will. A dense spray of micron-sized water droplets is formed in air by a pneumatic atomizer and conveyed through a nozzle in a closed chamber whose vapor concentration has been pre-set to a controlled value. The resulting plume extension depends on the relative humidity of the diluting medium. When the spray plume is straight and laminar, droplets evaporate at its edge where the vapor is saturated, and diffuses through a boundary layer developing around the plume. We quantify the shape and length of the plume as a function of the injecting, vapor diffusion, thermodynamic and environment parameters. For higher injection Reynolds numbers, standard shear instabilities distort the plume into stretched lamellae, thus enhancing the diffusion of vapor from their boundary towards the diluting medium. These lamellae vanish in a finite time depending on the intensity of the stretching, and relative humidity of the environment, with a lifetime diverging close to the equilibrium limit, when the plume develops in an medium saturated in vapor. The dependences are described quantitatively.

Tuesday, November 24, 2015 12:50PM - 3:26PM –
Session R27 Experiments: Particles, Drops, and Bubbles 308 - Jordi Esteva deordal, North Dakota State University

12:50PM R27.00001 Multiple light scattering methods for multiphase flow diagnostics, JORDI ESTEVA DEORDAL, North Dakota State Univ — Multiphase flows of gases and liquids containing droplets, bubbles, or particulates present light scattering imaging challenges due to the interference from each phase, such as secondary reflections, extinctions, absorptions, and refractions. These factors often prevent the unambiguous detection of each phase and also produce undesired beam steering. The effects can be especially complex in presence of dense phases, multispecies flows, and high pressure environments. This investigation reports new methods for overcoming these effects for quantitative measurements of velocity, density, and temperature fields. The methods are based on light scattering techniques combining Mie and filtered Rayleigh scattering and light extinction analyses and measurements. The optical layout is designed to perform multiple property measurements with improved signal from each phase via laser spectral and polarization characterization, etalon decontamination, and use of multiple wavelengths and imaging detectors.
1:03PM R27.00002 Turbulent crude oil jets in crossflow: holographic measurements of droplet size distributions1 | XINZHI XUE, DAVID MURPHY, JOSEPH KATZ, Johns Hopkins University Department of Mechanical Engineering — Buoyant, immiscible jets and plumes are created by subsurface oil well blowouts. In this experimental study, high speed visualizations and digital holography follow vertical crude oil turbulent jets of varying Reynolds and Ohnesorge numbers, all falling in the atomization range, while being towed in a towing tank generating ‘crossflows’ at varying crossflow-to-exit speed velocity ratios. The droplet size distributions are measured using a submersed miniature holographic microscopy system, enabling comparison between the plume behavior and the droplet size distributions. Due to variations in rise-velocity with droplet size, the shape and dispersion rate of the plume depends on the interfacial tension. Hence, the crude oil plume rises faster than a ‘control’ miscible oil analog with the same density and viscosity. Premixing the oil with dispersant (Corexit 9500A) at dispersant to oil (DOR) ratios of 1:100 and 1:25 reduces the oil-seawater interfacial tension by up to two orders of magnitude, promoting formation of micro-droplets. Hence, the plume rises at a slower rate, with the large droplets rapidly escaping, leaving smaller ones behind. Furthermore, for the DOR 1:25 case, some of the microdroplets are entrained into the vortices prominent in the wake region under the plume.

1Funding provided by the Gulf of Mexico Research Initiative

1:16PM R27.00003 Confined nanoparticle measurement using Bessel Beam Microscopy | CHUMKI CHAKRABORTY, CRAIG SNOEYINK, Department of Mechanical Engineering, Texas Tech University — With the advent of Lab-on-chip technologies, study of near surface phenomenon has gained a lot of importance due to their huge impact on bulk fluid properties. Such studies demand imaging techniques with utmost precision to capture the intricate details of the interface. But, resolution for most of the optical imaging systems is limited due to the light spreading effects of diffraction. This diffraction limited resolution, can be improved by the use of Bessel Beam microscopy. Bessel beam imaging technique when combined with a TIRF (Total Internal Reflection Fluorescence) system can be used for high resolution particle tracking experiments, to reveal detailed information about near surface particle positions and motions with their velocity profile and distribution. With the experimental set up combining these two powerful tools, we plan to present our particle tracking velocimetry results in the interface regime of confined nanoparticles in a binary fluid mixture. Such a study can contribute towards a better understanding of near surface fluid-particle interfaces.

1:29PM R27.00004 Multi-camera PIV imaging in two-phase flow for improved dispersed-phase concentration and velocity calculation | CHANG LIU, KEN KIGER, Dept. of Mech. Engr., Univ. of Maryland — PIV/PTV has been widely used in making simultaneous measurements of velocity and concentration within multi-phase flows. A major problem confronted by researchers during data processing is to separate the image signals of the dispersed phase from carrier phase reliably and within the same measurement volume. For dilute concentrations, size and brightness criteria have been shown to provide satisfying results in identifying the dispersed phase. However this method is limited to fairly small concentrations due to effects of multiple-scattering and obscuration. To extend this technique, we introduce multi-camera imaging as a means to provide a more precise and reliable identification of the dispersed phase in the face of increased concentration. Specifically, the size-brightness criteria is used to nominally match corresponding dispersed-phase images of the same particle within the other views, and the subsequent out-of-plane position is used to get a more precise 3D location of the particle. In order to demonstrate this method, experiments using static test cell of solid glass sphere suspended in an aqueous gel have been conducted under various concentration and compared to corresponding single camera results.

1:42PM R27.00005 A Method to Improve the Accuracy of Particle Diameter Measurements from Shadowgraph Images1 | MARTIN A. ERININ, DAN WANG, XINAN LIU, JAMES H. DUNCAN, University of Maryland — A method to improve the accuracy of the measurement of the diameter of particles using shadowgraph images is discussed. To obtain data for analysis, a transparent glass calibration reticle, marked with black circular dots of known diameters, is imaged with a high-resolution digital camera using backlighting separately from both a collimated laser beam and diffuse white light. The diameter and intensity of each dot is measured by fitting an inverse hyperbolic tangent function to the particle image intensity map. Using these calibration measurements, a relationship between the apparent diameter and intensity of the dot and its actual diameter and position relative to the focal plane of the lens is determined. It is found that the intensity decreases and apparent diameter increases/decreases (for collimated/diffuse light) with increasing distance from the focal plane. Using the relationships between the measured properties of each dot and its actual size and position, an experimental calibration method has been developed to increase the particle-diameter-dependent range of distances from the focal plane for which accurate particle diameter measurements can be made.

1The support of the National Science Foundation under grant OCE0751853 from the Division of Ocean Sciences is gratefully acknowledged.

1:55PM R27.00006 Aqueous ammonium thiocyanate solutions as refractive index-matching fluids with low density and viscosity1 | BENJAMIN C. MORRISON, DANIEL BORREIRO-ECHEVERRY, Reed College — Index-matching fluids play an important role in many fluid dynamics experiments, particularly those involving particle tracking, as they can be used to minimize errors due to distortion from the refraction of light across interfaces of the apparatus. Common index-matching fluids, such as sodium iodide solutions or mineral oils, often have densities or viscosities very different from those of water. This can make them undesirable for use as a working fluid when using commercially available tracer particles or at high Reynolds numbers. A solution of ammonium thiocyanate (NH₄SCN) can be used for index-matching common materials such as borosilicate glass and acrylic, and has material properties similar to those of water (ρ ~ 1.1 g/cc and μ ~ 1.1 cSt) and p ~ 1.1 g/cc). We present an empirical model for predicting the refractive index of aqueous NH₄SCN solutions as a function of temperature and NH₄SCN concentration that allows experimenters to develop refractive index matching solutions for various common materials.

1This work was supported by the National Science Foundation (CBET-0853691) and by the James Borders Physics Student Fellowship at Reed College.
explore whether the bubbling regime, which is naturally established for certain values of the water-to-air velocity ratio, \( \frac{u_o}{u_a} \).

An RICM Study conducted with deformable droplets settling under gravity in a suspending liquid for Bond numbers of few molecular length scales, layering and adsorption of long chains of polymers can cause entropic repulsion due to a reduced configurational freedom. This repulsive force can prevent film rupture and lead to the formation of an equilibrium film. In the current work, experiments were conducted with deformable droplets settling under gravity in a suspending liquid for Bond numbers of \( O(10^{-4}) \). The film drainage was studied using a microinterferometric technique namely, Reflection Interference Contrast Microscopy (RICM) for two different systems: a) silicone oil drops in paraffin oil, b) glycerol drops in silicone oil. The RICM analysis for obtaining the film drainage profiles, was done using a combination of simple cosine theory and ray tracing algorithm. For the silicone oil-paraffin oil system, the film drainage behavior observed was as expected from simulations based on thin film drainage equations. On the other hand, glycerol drops of radii smaller than 130 \( \mu m \), resulted in the formation of an equilibrium film of silicone oil with an approximate thickness of 10 nm. The origin of this repulsive force is attributed to the presence of an immobilized layer of adsorbed polymer chains. Film drainage observed in glycerol drops of radii larger than 130 \( \mu m \), was found to destabilize in a non-axisymmetric mode. The rapid growth of this asymmetric instability can lead to stresses (\( O(100 \text{ Pa}) \)) higher than the yield stress of the adsorbed polymer layer.

2:21PM R27.00008 Manipulation of Nano-/Micro Particles Using Light-Actuated Marangoni Tweezers

2:34PM R27.00009 Experimental study on bi-phase flow Air-Oil in Water Emulsion

2:47PM R27.00010 A novel technique to control the bubble formation process in a co-flow configuration with planar geometry

3:00PM R27.00011 Light Attenuation Method for 3D data acquisition (LAM3D) of bottom particle deposits

---

1Supported by the Spanish MINECO, Junta de Andalucía and EU Funds under projects DPI2014-59292-C3-3-P, P11-TEP7495 and UJA2013/08/05
3:13PM R27.00012 Velocity and size distribution measurement of suspension droplets using PDPA technique, SHAHIN AMIRI, ALI AKBARNOZARI, CHRISTIAN MOREAU, ALI DOLATABADI, Concordia University — The creation of fine and uniform droplets from a bulk of liquid is a vital process in a variety of engineering applications, such as atomization in suspension plasma spray (SPS) in which the submicron coating materials are injected to the plasma gas through the suspension droplets. The size and velocity of these droplets has a great impact on the interaction of the suspension with the gas flow emanating from a plasma torch and can consequently affect the mechanical and chemical properties of the resultant coatings. In the current study, an aqueous suspension of small glass particles (2-8 µm) was atomized by utilizing an effervescent atomizer of 1 mm orifice diameter which involves bubbling gas (air) directly into the liquid stream. The gas to liquid ratio (GLR) was kept constant at 6% throughout this study. The mass concentration of glass particles varied in the range between 0.5 to 5% in order to investigate the effect of suspension viscosity and surface tension on the droplet characteristics, such as velocity and size distributions. These characteristics were simultaneously measured by using a non-intrusive optical technique, Phase Doppler Particle Anemometry (PDPA), which is based on the light signal scattered from the droplets moving in a measurement volume. The velocity and size distribution of suspension droplets were finally compared to those of distilled water under identical conditions. The results showed a different atomization behaviors due to the reduction in surface tension of the suspension spray.

Tuesday, November 24, 2015 12:50PM - 3:26PM —
Session R28 Geophysical Fluid Dynamics: General 309 - Glenn Flierl, MIT

12:50PM R28.00001 Eddy transport of reacting substances, GLENN FLIERL, MIT — We examine an exact formulation of eddy fluxes but extended to tracers which react with each other. The resulting formula is evaluated using the lattice model approach allowing not only control (including elimination) of sub-grid-scale diffusion and efficient enough computation to generate an adequate ensemble. The theory predicts that the flux is a non-local average of the mean gradients, even for passive scalars, and we can calculate the averaging kernel. The reaction terms alter the effective transport for a single scalar depending on decay time scale compared to that of the Lagrangian covariance. But, in addition, the eddies produce "cross-fluxes" whereby the transport of each tracer depends on the gradients of all other tracers. The results showed a different atomization behaviors due to the reduction in surface tension of the suspension spray.

1 Supported by the ERC AdG NewTURB num. 339032

1:03PM R28.00002 Eulerian and Lagrangian statistics in fully developed rotating turbulent flows, L. LUCA BIFERALE, FABIO BONACCORSO, IRENE MAZZITELLI, University of Rome ‘Tor Vergata’, ALESSANDRA LANOTTE, CNR ISAC and INFN, Lecce, Italy, PRASAD PERLEKAR, Tata Institute of Fundamental Research, India, STEFANO MUSACCHIO, Universitè de Nice Sophia Antipolis, CNRS, LJAD, Nice, France, MICHEL HINSBERG, FEDERICO TOSCHI, Fluid Dynamics Laboratory, Department of Physics and Eindhoven University of Technology, The Netherlands — We present results concerning both Eulerian and Lagrangian statistics for turbulent under rotation at small and large Rossby numbers. Concerning the Eulerian statistics we discuss the effects of the presence of strong coherent large-scale vortical structures on the small-scale statistics. Concerning Lagrangian properties, we discuss the effects of preferential sampling at changing the inertial properties of the particles also due to the centrifugal and Coriolis forces.

1MD acknowledges the support of the European Research Council (ERC) through the project MHetScale (617511)

1:16PM R28.00003 Lévy Dynamics of Stretching in 2-Dimensional Steady Random Flow Fields, MARCO DENTZ, IDAEA-CSIC, Barcelona, Spain, TANGUY LE BORGNE, Geosciences Rennes, UMR 6118, Université de Rennes 1, CNRS, Rennes, France, DANIEL LESTER, RMIT University, Melbourne, Australia, FELIPE P. J. DE BARROS, University of Southern California, 3620 S. Vermont Avenue, KAP 224B, Los Angeles, — Stretching and compression of material fluid elements is key for the understanding and quantification of the mixing dynamics. For 2-dimensional steady random flows the elongation of a material strip \( \rho(t) \) grows algebraically as \( \rho(t) \propto t^\gamma \). The stretching exponent \( \gamma \) depends on the heterogeneity strength. While the Poincaré-Bendixson theorem explains the absence of exponential stretching in steady 2d flows, the mechanisms of the algebraic stretching behavior and its relation to the flow statistics are not known. Here we formulate the deformation of a material fluid element in streamline coordinates, which unravels the dynamics of the stretching process as a Lévy walk. We provide an explicit relation between the stretching process and the flow heterogeneity and derive the scaling behavior of elongation with time. We find for the stretching exponent \( \gamma \) is bounded between 1/2 and 2, where \( \gamma = 1/2 \) corresponds to weak heterogeneity and \( \gamma = 2 \) to strongly heterogeneous flow fields.

1Supported by the European Research Council (ERC) through the project MHetScale (617511)

1:29PM R28.00004 A Dynamical System Approach to the Surface Search of Debris from MH370, ANA M MANCHO, V. J. GARCIA-GARRIDO, ICMAT-CSIC, S. WIGGINS, University of Bristol, C. MENDOZA, Universidad Politécnica de Madrid — The disappearance of Malaysia Airlines flight MH370 on the morning of the 8th of March 2014 is one of the great mysteries of our time. One relevant aspect of this mystery is that not a single piece of debris from the aircraft was found during the intensive surface search carried out in the months following the crash. Difficulties in the search efforts, due to the uncertainty in the plane’s final impact point and the time passed since the accident, brought the question on how the debris was scattered in an always moving ocean, for which there were multiple datasets that do not uniquely determined its state. Our approach to this problem is based on dynamical systems tools that identify dynamic barriers and coherent structures governing transport. By combining different ocean data with these mathematical techniques, we are able to assess the spatio-temporal state of the ocean in the priority search area at the time of impact and the following weeks. Using this information we propose a revised search strategy by showing why one might not have expected to find debris in some large search areas targeted by the search services and determining regions where one might have expected impact debris to be located and that have not been subjected to any exploration.

1This research has been supported by MINECO under grants MTM2014-56392-R and ICMAT Severo Ochoa project SEV-2011-0087 and ONR grant No. N00014-01-1-0769. Computational support from CESGA is acknowledged.
1:42PM R28.00005 Lagrangian coherent structures in the Gulf Stream. YI LIU, Department of Mechanical and Aerospace Engineering, Syracuse University, Syracuse, NY, USA, CHRIS WILSON, National Oceanography Centre, Liverpool, UK, MELISSA GREEN, Department of Mechanical and Aerospace Engineering, Syracuse University, Syracuse, NY, USA — Finite-time Lyapunov exponent (FTLE) is calculated to identify Lagrangian coherent structures in the Gulf Stream region. The velocity fields are determined using the geostrophic velocities derived from satellite altimetry data. The coherent structures in and around the Gulf Stream are delineated by the both positive and negative FTLE ridges, and represent boundaries between dynamically distinct regions that are important to investigate transport and mixing processes in the ocean. Alternating positive and negative FTLE ridge patterns are found to line the meandering jet, which indicate the regions of entrainment and detrainment along the jet. Results compare well with the Bower kinematic model of a meandering jet, although it is clear that the kinematic model is an over-simplification of the jet dynamics, and studying the dynamics of vortex interaction with the jet is important for understanding fluid transfer in the Gulf Stream region.

1:55PM R28.00006 Vortex Ring Induce Mixing: A Mixing Model. JASON OLSTHOORN, STUART B. DALZIEL, Department of Applied Mathematics and Theoretical Physics — The parameterization of stratified turbulent mixing is key to developing large scale simulations of geophysical flows. Stratified mixing if often quantified through its mixing efficiency, a quantity that has been reported to vary significantly depending on the mixing mechanism. In the work presented here, we will investigate periodically-forced, externally-mixed stratified flows where the mixing mechanism is produced external to the mixing location. The mixing induced by vortex rings is a frequently studied phenomena as it is often compared with the eddies of fully developed turbulence. We continue this work experimentally with the aid of modern measurement techniques. Additionally, we have developed a one-dimensional model of vortex-ring-induced mixing and compare this model with both laboratory and numerical experiments. The results of the density field evolution, and the mixing efficiency, demonstrate a quantitative agreement between the model and experiments.

2:08PM R28.00007 On accuracy of overturn-based estimates of turbulent dissipation in a Luzon Strait model simulation with realistic topography. MASOUD JALALI, VAMSI CHALAMALLA, SUTANU SARKAR, UC San Diego — Oceanic density overturns are commonly used to estimate the dissipation rate of turbulent kinetic energy using the Thorpe sorting method. However, the accuracy of the dissipation estimate under different conditions is unclear. To assess the accuracy of Thorpe estimates of turbulence dissipation, 3D LES are performed with scaled semiannual frequency in a scaled down model of Luzon Strait topography. The Thorpe-scale method is found to be able to qualitatively estimate the spatial distribution and phasing of dissipation rate but there are quantitative errors. It overestimates the magnitude of dissipation in locations with strong convectively driven turbulence. The extent of overestimation in the case of Luzon strait is up to more than one order of magnitude. However, the Thorpe estimate has reasonably good agreement with the dissipation in regions with shear-driven turbulence. An alternative model based on a modified estimate of the Thorpe scale from the vertical profile of density is introduced. This method is able to estimate convectively driven dissipation more accurately, although it is less accurate in regions with shear driven turbulence such as downslope jets. Both methods of inferring dissipation rate exhibit phase difference with respect to the value in the simulations.

2:21PM R28.00008 From convection rolls to finger convection in double-diffusive turbulence. YANTAO YANG, Physics of Fluids Group, University of Twente, ROBERTO VERZICCO, Dipartimento di Ingegneria Industriale, University of Rome “Tor Vergata”, DETLEF LOHSE, Physics of Fluids Group, University of Twente — The double diffusive convection (DDC), where the fluid density depends on two scalar components with very different molecular diffusivities, is frequently encountered in oceanography, astrophysics, and electrochemistry. In this talk we report a systematic study of vertically bounded DDC for various control parameters. The model is driven by an unstable density difference between two plates and stabilized by a temperature difference. As the convective strength of temperature difference becomes stronger, the flow transits from a state with large-scale convection rolls, which is similar to the Rayleigh-Bénard (RB) flow, to a state with well-organised salt fingers. When the temperature difference increases further, the flow breaks down to a purely convection state. During this transit the velocity decreases monotonically. Counterintuitively, the salinity transfer can be enhanced when a stabilising temperature field is applied to the system. This happens when convection rolls are replaced by salt fingers. In addition, we show that the Grossmann-Lohse theory originally developed for RB flow can be directly applied to the current problem and accurately predicts the salinity transfer rate for a wide range of control parameters.

2:34PM R28.00009 Mixing in the spiral roll state in heat convection between concentric double spherical boundaries. TOMOAKI ITANO, Kansai Univ., TAKAHIRO NINOMIYA, CKD Corporation, KOHEI IIDA, MASAKO SUGIHARA-SEKI, Kansai Univ. — Recent studies have indicated that the spherical Rayleigh-Bénard convection provides a variety of non-trivial (convective) flow states bifurcating from the (conductive) static state at the onset of instability. In the present study, focusing on one of them, “spiral roll state”, which was originally explored by Zhang et al.(2002), we elucidated the following three points of the state; (1) the state exists even in a fairly thicker gap than expected in the previous study, and (2) it is an exact autonomously rotating wave solution, and (3) the state bifurcates directly from the static state at the onset of instability. It is of great interest that this state involves globally mixing through the whole domain beyond a cell which would be formed by the other highly-symmetric nontrivial steady states bifurcating at the onset.

2:47PM R28.00010 The Modified Rayleigh-Benard Convection Problem and its Application to Permafrost Methane Emission Modeling. IVAN SUDAKOV, Physics Department, University of Dayton, SERGEY VAKULENKO, Institute for Problems in Mechanical Engineering, Russian Academy of Sciences — The original Rayleigh-Benard convection is a standard example of the system where the critical transitions occur with changing of a control parameter. We will discuss the modified Rayleigh-Benard convection problem which includes the radiative effects as well as the specific gas sources on a surface. Such formulation of this problem leads to identification of new kind of nonlinear phenomenon, besides the well-known Benard cells. Modeling of methane emissions from permafrost into the atmosphere drives to difficult problems, involving the Navier-Stokes equations. Taking into account the modified Rayleigh-Benard convection problem, we will discuss a new approach which makes the problem of a climate catastrophe in the result of a greenhouse effect more tractable and allows us to describe catastrophic transitions in the atmosphere induced by permafrost greenhouse gas sources.
The original experiment, the flow is slow to setup, but transition occurs quickly. The scaling and energetics will be discussed.

The simulations are initialized with a weak random flow field and allowed to evolve under the influence of constant shear and heat flux. Like strongly shear driven air-water interface, modeled as a flat interface with a specified Stokes drift and a constant heat flux cooling the interface. The extended box model recently reported in [Zgheib et al. 2014 Theor. Comput. Fluid Dyn. 28, 521-529] is used to propose a relation for the self-similar horizontal aspect ratio of the propagating front as a function of the initial horizontal aspect ratio. The experimental and numerical results are in good agreement with the proposed relation.

Tuesday, November 24, 2015 12:50PM - 3:26PM –
Session R29 Geophysical Fluid Dynamics: Sediment Transport

12:50PM R29.00001 Onset and cessation of grain motion in fluid-sheared beds1, ABE CLARK, JULIA SALEVAN, Yale University, MARK SHATTUCK, City College of New York, NICK OUELLETTE, Stanford University, COREY O’HERN, Yale University — We performed molecular dynamics simulations of granular beds driven by a model hydrodynamic shear flow to elucidate general grain-scale mechanisms that determine the onset and cessation of sediment transport. By varying the Shields number (the non-dimensional shear stress at the top of the bed) and particle Reynolds number (the ratio of particle inertia to viscous damping), we explore how variations of the flow velocity, particle inertia, and fluid viscosity affect the onset and cessation of bed motion. For low to moderate particle Reynolds numbers, a critical boundary separates mobile and static states. Transition times between these states diverge as this boundary is approached both from above and below. At high particle Reynolds number, inertial effects become dominant, and particle motion can be sustained well below flow rates at which mobilization of a static bed occurs. We also find that the onset of bed motion (for both low and high particle Reynolds numbers) is described by Weibullian weakest-link statistics, and thus is crucially dependent on the packing structure of the granular bed, even deep beneath the surface.

1This work was supported by the US Army Research Office under Grant No. W911NF-14-1-0005.

1:03PM R29.00002 Wave-Induced Pressure Under an Internal Solitary Wave and Its Impact at the Bed, GUSTAVO RIVERA, PETER DIAMESIS, JAMES JENKINS, Cornell University, DIEGO BERZI, Politecnico di Milano — The bottom boundary layer (BBL) under a mode-1 internal solitary wave (ISW) of depression propagating against an oncoming model barotropic current is examined using 2-D direct numerical simulation based on a spectral multidomain penalty method model. Particular emphasis is placed on the diffusion into the bed of the pressure field driven by the wake and any near-bed instabilities produced under specific conditions. To this end, a spectral nodal Galerkin approach is used for solving the diffusion equation for the wave-induced pressure. At sufficiently high ISW amplitude, the BBL undergoes a global instability which produces intermittent vortex shedding from within the separation bubble in the lee of the wave. The interplay between the bottom shear stress field and pressure perturbations during vortex ejection events and the subsequent evolution of the vortices is examined. The potential for bed failure upon the passage of the ISW trough and implications for resuspension of bottom particulate matter are both discussed in the context of specific sediment transport models.

1:16PM R29.00003 Highly-resolved numerical simulations of bed-load transport in a turbulent open-channel flow1, BERNHARD WOVINCKEL, Mechanical Engineering, UCSB, Santa Barbara, USA, TOBIAS KEMPE, Institute of Fluid Mechanics, TU Dresden, Germany, VLADIMIR NIKORA, School of Engineering, The University of Aberdeen, Schotland, RAMANDEEP JAIN, JOCHEN FROHLICH, Institute of Fluid Mechanics, TU Dresden, Germany — The study presents the analysis of phase-resolving Direct Numerical Simulations of a horizontal turbulent open-channel flow laden with a large number of spherical particles. These particles have a mobility close to their threshold of incipient motion and are transported in bed-load mode. The coupling of the fluid phase with the particless is realized by an Immersed Boundary Method. The Double-Averaging Methodology is applied for the first time convoluting the data into a handy set of quantities averaged in time and space to describe the most prominent flow features. In addition, a systematic study elucidates the impact of mobility and sediment supply on the pattern formation of particle clusters in a very large computational domain. A detailed description of fluid quantities links the developed particle patterns to the enhancement of turbulence and to a modified hydraulic resistance. Conditional averaging inapplied toero & end events provideinsight the processes involved incipient particle motion. Furthermore, the detection of moving particle clusters as well as their surrounding flow field is addressed by a moving frameanalysis.

1Funded by German Research Foundation (DFG), project FR 1593/5-2, computational time provided by ZIH Dresden, Germany, and JSC Juelich, Germany.

1:29PM R29.00004 Two-dimensional PIV measurements for studying the effect of bed permeability on incipient motion of synthetic sediment particles, HENG WU, CARLO C. ZUNIGA ZAMALLOA, JORGE E. S. BLANCO, BLAKE J. LANDRY, MARCELO H. GARCIA, University of Illinois at Urbana-Champaign — The experimental study of incipient motion, the regime where particles resting on a granular bed enter a process of sediment transport, can be approached using a single-particle pivoting model. Such pivoting model states that the fundamental mechanism of the incipient transport depends, among other factors, on the local fluid flow, bed-flow interface topology, the geometry, and specific density of the sediment particle; yet it does not specify the effect of the bed permeability. In this work the effect that bed permeability has on the incipient motion of a sediment particle is explored by conducting systematic Particle Image Velocimetry measurements of the flow around cylindrical and spherical particles at incipient motion conditions in a water flume. The permeable bed condition for the flume is achieved by placing a synthetic bed at the bottom which consists of cubically packed, uniformly sized spheres. The impermeable condition is obtained by placing at the bottom of the flume a sheet with rows of hemispheres glued to it, the hemispheres being of the same diameter as the ones in the permeable case. The mean velocity profiles are reported to illustrate the influence of the permeable or impermeable beds. The measured velocity data is also compared with the current pivoting model.
Experimental data fit nicely with the model results. The self-similar dune shape and power-law growth exponent are extracted by image processing for several flow velocities. A simple sand ripples appear all along the channel. We found that the first ripple near the channel inlet exhibit unreported long-term scale-invariant experimental results obtained in a linear, quasi-2D closed water channel. When a granular bed is submitted to a uniform shear flow, periodic experiments. On the other hand, the dynamical long term evolution of ripples and dunes formed over an erodible bed has been far less quantitatively predicts the orientation of dunes in Earth deserts. Finally, we explore the phase diagram and the stability of the fingering mode. Then, we derive a model for dunes orientation, which explains the coexistence of bedforms with different alignments and numerical simulations where a single wind regime can lead to two different dunes orientation depending on sediment availability. Sediment different length scales and orientations, which seem difficult to relate to a single wind cycle. We present results of underwater experiments and simulations, critical conditions for sediments to be mobilized have been reported. The abstraction of mobilized sediment to equivalent resuspension flux is that robust and truly based on local flow and bed conditions has not been formulated. Such a resuspension flux will be an improvement over the current models that are based on a correlation between average shear stress and the increase in the sediment load as a turbidity current propagates over a certain span of the bed. In this study we will present an improved model for resuspension flux which is a function of local bed shear stress and particle Reynolds number. This function incorporates the local kinematics of a particle lying on the bed along with turbulence characteristics of the flow such that a spatial average of shear stress and resuspension flux reconciles with the existing models proposed in the literature.

This work has been supported by ExxonMobil Upstream Research Company.

On the intermittency of sediment transport in conditions near the threshold of motion1, CHRISTIAN GONZALEZ, CRISTIAN ESCAURIAZA, Pontificia Universidad Catolica de Chile, DAVID RICHTER, DIOGO BOLSTER, University of Notre Dame, JOSEPH CALANTONI, Naval Research Laboratory — The dynamics of sediment particle transport in a water-filled cylinder is modeled by perturbing the turbulent boundary layer using a feedback control technique. As shown already by previous work, the bed remains flat close to the center of the cylinder, and radial ripples form at outer radii. The size of the inner flat region and the number or ripples depend on the frequency and amplitude of the cylinder’s oscillation. In the present work, we are interested in the dynamics and control of the bed forms when the primary sinusoidal signal of the oscillation is perturbed by adding a second sinusoidal signal with a relatively small amplitude, a different frequency, and a phase lag. Varying the parameters of the secondary signal results in a signal that can be asymmetric or modulated, for example. These properties translate into the bed producing simple behavior like the propagation of the ripples at a constant speed or more complex behavior like the time dependent coarsening and thinning of the ripples.

This research is funded by NWO (the Netherlands) through the VENI grant 863.13.022.

Two modes for dune orientation, SYLVAIN COURRECH DU PONT, Lab. Matière et Systèmes Complexes - Université Paris Diderot, CLMENT NARTEAU, XIN GAO, Institut Physique du Globe de Paris — Earth sand seas experience winds that blow with different strengths and from different directions in line with the seasons. In response, dune fields show a rich variety of shapes from small crescentic barchans to big star and linear dunes. Linear dunes often exhibit complex and compound patterns with different length scales and orientations, which seem difficult to relate to a single wind cycle. We present results of underwater experiments and numerical simulations where a single wind regime can lead to two different dunes orientation depending on sediment availability. Sediment availability selects the overriding mechanism for the formation of dunes: increasing in height from the destabilization of a sand bed or elongating in a finger on the non-erodible ground from a localized sand source. These mechanisms drive the dunes orientation. Therefore, dunes alignment maximizes dunes orthogonality to sand fluxes in the bed instability mode, while dunes are aligned with the sand transport direction in the fingering mode. Then, we derive a model for dunes orientation, which explains the coexistence of bedforms with different alignments and quantitatively predicts the orientation of dunes in Earth deserts. Finally, we explore the phase diagram and the stability of the fingering mode.

Self-similar evolution of 2D aquatic dunes over an erodible bed, DELPHINE DOPPLER, LMFA-Université Lyon 1, Université de Lyon, PIERRE YVES LAGRÈE, Institut Jean Le Rond d’Alembert - CNRS, PHILIPPE GONDRET, MARC RABAUD, FAST - Université Paris Sud, Orsay — Scale invariance of shape is a common feature of erosion patterns, such as barchan dunes, sand ripples under flowing waves or scour holes. Due to their universal and fascinating crescentic shape, barchans dunes have received much attention and scaling laws have been deduced from field observations, satellite images and laboratory experiments. On the other hand, the dynamical long term evolution of ripples and dunes formed over an erodible bed has been far less studied while the temporal behavior of erosion patterns contains substantial information on the physical processes involved. Here, we present experimental results obtained in a linear, quasi-2D closed water channel. When a granular bed is submitted to a uniform shear flow, periodic sand ripples appear all along the channel. We found that the first ripple near the channel inlet exhibit unreported long-term scale-invariant growth. The self-similar dune shape and power-law growth exponent are extracted by image processing for several flow velocity. A simple linear model is built using mass conservation and a granular flux law, so that the bed form is described by a self-similar order 2 linear system. Experimental data fit nicely with the model results.
Large-eddy simulation of sand dune morphodynamics\textsuperscript{1}, ALI KHOSRONEJAD, FOTIS SOTIROPoulos, St. Anthony Falls Lab. University of Minnesota, ST. ANTHONY FALLS LABORATORY, UNIVERSITY OF MINNESOTA TEAM — Sand dunes are natural features that form under complex interaction between turbulent flow and bed morphodynamics. We employ a fully-coupled 3D numerical model (Khosronejad and Sotiropoulos, 2014, Journal of Fluid Mechanics, 753:150-216) to perform high-resolution large-eddy simulations of turbulence and bed morphodynamics in a laboratory scale mobile-bed channel to investigate initiation, evolution and quasi-equilibrium of sand dunes (Venditti and Church, 2005, J. Geophysical Research, 110:F01009). We employ a currently available boundary layer approach along with convection-diffusion and bed-morphodynamics modules to simulate the suspended sediment and the bed-load transports respectively. The coupled simulation were carried out on a grid with more than 100 million grid nodes and simulated about 3 hours of physical time of dune evolution. The simulations provide the first complete description of sand dune formation and long-term evolution. The geometric characteristics of the simulated dunes are shown to be in excellent agreement with observed data obtained across a broad range of scales.

\textsuperscript{1}Acknowledgments This work was supported by NSF Grants EAR-0120914 (as part of the National Center for Earth-Surface Dynamics). Computational resources were provided by the University of Minnesota Supercomputing Institute.

Scour of Sand-Gravel Beaches in Front of Seawalls, REGIS XHARDE, JANNETTE FRANDSEN, OLIVIER GAUVIN-TREMBLAY, INRS-ETE — Large-scale physical experiments were conducted in the 5m-wide, 5m-deep and 120m-long wave flume at the Quebec Coastal Laboratory of the national scientific research institute (INRS) to evaluate wave-induced scour depth ($d_s$) at vertical seawalls and on natural beaches. In the initial part of the study, the equilibrium beach profile of a mixed sand-gravel beach with a mean grain size diameter of 12 mm was studied for various beach slopes using regular and irregular waves with intermediate water depths ($h_0 \in [2.3; 3.8]$ m) and different wave heights. In the second part of the study, a vertical seawall fronted by a 1:30 sloping mixed sand-gravel beach was tested for more than 50 wave trains using regular and irregular waves with various water depths at the seawall ($h_w$), wave heights and wave periods. The scour depth at the toe of the seawall is highly dependent on the form of wave breaking onto the structure. Sea states where plunging breakers occur directly onto the wall generate jets of water that may penetrate to the seabed and cause a local scour hole immediately adjacent to the seawall. Scour depth is maximum when $h_p/h_w > 1$ and $X_B/h_w < 1$, where $h_p$ is the breaker height and $X_B$ the distance from the seawall of the breaking wave. Comparison with existing semi-empirically derived scour prediction equations was performed.

Zombie Vortex Instability: Effects of Non-uniform Stratification & Thermal Cooling, JOSEPH BARRANCO, SFSU, SUYANG PEI, PHIL MARCUS, CHUNG-HSIANG JIANG, U.C. Berkeley — The Zombie Vortex Instability (ZVI) is a nonlinear instability in rotating, stratified, shear flows, such as in protoplanetary disks (PPD) of gas and dust orbiting new stars. The instability mechanism is the excitation of baroclinic critical layers, leading to vorticity amplification and nonlinear evolution into anticyclonic vortices and cyclonic sheets. ZVI is most robust when the Coriolis frequency, shear rate, and Brunt–Väisälä (BV) frequency are of the same order. Previously, we investigated ZVI with uniform stratification and without thermal cooling. Here, we explore the role of non-uniform stratification as would be found in PPDs in which the BV frequency is zero in the disk midplane, and increases away from the midplane. We find that ZVI is vigorous 1-3 pressure scale heights away from the midplane, but the non-isotropic turbulence generated by ZVI can penetrate into the midplane. We also explore the effect of thermal cooling and find that ZVI is still robust for cooling times as short as 5 orbital periods. ZVI may play important roles in transporting angular momentum in PPDs, and in trapping dust grains, which may trigger gravitational clumping into planetsimals.

Experimental Simulation of Buoyancy-Driven Vortical Flow in Jupiter Great Red Spot, HADY MAJKHALBAI, HADY MAJKHALBAI, PhD candidate/research assistant Dept of Mechanical and Aeronautical Engineering, TIANSHU LIU, Professor Department of Mechanical and Aerospace Engineering, PARVIZ MERATI, Professor and Chair Department of Mechanical and Aerospace Engineering — This new experimental study of Geophysical Buoyancy-Driven Vortical Flow presents a new approach to model the Great Red Spot (GRS) that explains some feature of this phenomena that other classic approaches such as shallow layer model and deep layer model do not. The low velocity region at the center and the counter rotating system at the core that recently were observed by high resolution imaging processing methods, have never been justified before. This setup generates flow structures similar to the GRS in the test zone and compares the results and suggests that a counter rotating flow structure at the lower altitude is the source of the GRS formation.

\textsuperscript{1}PhD candidate/research assistant Dept of Mechanical and Aeronautical Engineering Western Michigan University Kalamazoo MI 49008-5343 Room G-106 Fluids Lab T:(269)348-6229 F:(269)348-6231
\textsuperscript{2}Professor Department of Mechanical and Aerospace Engineering, PARVIZ MERATI, Professor and Chair Department of Mechanical and Aerospace Engineering — This new experimental study of Geophysical Buoyancy-Driven Vortical Flow presents a new approach to model the Great Red Spot (GRS) that explains some feature of this phenomena that other classic approaches such as shallow layer model and deep layer model do not. The low velocity region at the center and the counter rotating system at the core that recently were observed by high resolution image processing methods, have never been justified before. This setup generates flow structures similar to the GRS in the test zone and compares the results and suggests that a counter rotating flow structure at the lower altitude is the source of the GRS formation.

1:29PM R30.00004 ABSTRACT WITHDRAWN —
1:42PM R30.00005 Laboratory Observation of Instabilities in Stratified Taylor-Couette Flow \textsuperscript{1}, BRUCE RODENBORN, Centre College, RUY IBANEZ, HARRY L. SWINNEY, Center for Nonlinear Dynamics and Department of Physics, University of Texas at Austin — In 2001 Molemaker et al. (J. Fluid. Mech. 448, 1, 2001) predicted a new class of instabilities in a system of concentric rotating cylinders that contains a fluid with a vertically varying density. Dubrulle et al. (Astron. Astrophys. 429, 1, 2005) then showed that this phenomenon, which they named stratorotational instability (SRI), could be a source of instability and angular momentum transport in astrophysical accretion disks. Subsequent work by Shalybkov and Rüdiger (Astron. Astrophys. 438, 411, 2005) hypothesized that such stratified flow is stable when the ratio of outer and inner cylinder rotation rates \( \mu \) is less than the ratio of the inner and outer cylinder radii \( \nu \). Previous laboratory measurements by Le Bars and Le Gal (Phys. Rev. Lett. 99, 064502, 2007) confirmed this prediction for \( Re < 1200 \) with \( Re = \left( r_i - r_o \right) \Omega_i / \nu_i \). However, we find SRI exists for \( \mu > \nu \) when the density gradient is large. We also find that the onset of SRI is suppressed for Reynolds numbers \( Re > 4000 \), a region previously unexplored in experiments. For \( Re > 8000 \), we find that the fluid does not exhibit SRI but transitions to a previously unreported chaotic state that mixes the fluid.

1:55PM R30.00006 A Multiscale Dynamo Model Driven by Quasi-geostrophic Convection\textsuperscript{1}, KEITH JULIEN, MICHAEL CALKINS, University of Colorado at Boulder, STEVE TOBIAS, University of Leeds, JONATHAN AURNOU, University of California at Los Angeles — A convection-driven multiscale dynamo model is discussed for the plane layer geometry in the limit of low Rossby number. The small-scale fluctuating dynamics are described by a magnetically-modified quasi-geostrophic equation set, and the large-scale mean dynamics are governed by a diagnostic thermal wind balance. The model utilizes three timescales that respectively characterize the convective timescale, the large-scale magnetic diffusion timescale, and the large-scale thermal diffusion timescale. It is shown that in limit of low magnetic Prandtl number the model is characterized by a magnetic to kinetic energy ratio that is asymptotically large, with ohmic dissipation dominating viscous dissipation on the large-scales. For the order one magnetic Prandtl number model the magnetic and kinetic energies are equipartitioned and both ohmic and viscous dissipation are weak on the large-scales. For both cases the number is significantly smaller than that of fully nonlinear, generalized versions of the dynamo model originally developed by Childress and Soward. These models may be useful for understanding the dynamics of convection-driven dynamos in regimes that are only just becoming accessible to simulations of the full set of governing equations.

1:55PM R30.00007 Numerical and Statistical Simulations of an Idealized Model Tachocline\textsuperscript{1}, ABIGAIL PLUMMER, Harvard University, STEVE TOBIAS, University of Leeds, BRAD MARSTON, Brown University — Solar-type stars with outer convective envelopes and stable interiors are believed to have tachoclines. As in the Sun, the tachocline is a thin shear layer thought to play an important role in the magnetic activity of these stars. We use an idealized two-dimensional model tachocline to investigate a joint instability in which the differential rotation is only stable in the absence of a magnetic field. A set of parameters are identified using Direct Numerical Simulations (DNS) that produce a cycle in which energy is transferred abruptly between kinetic and magnetic potential energy reservoirs. Elements of this cyclic behavior are replicated using Direct Statistical Simulations (DSS). Insight is thus gained into the physics prompting these sharp transitions, suggesting that they are the result of eddies interacting to form new eddies.

1BM supported in part by NSF DMR-1306806 and NSF CCF-1048701.

2:08PM R30.00008 The Non-linear Saturation of the Goldreich-Schubert-Fricke Instability \textsuperscript{1}, JEFFREY OISHI, State Univ of NY - Farmingdale, KEATON BURNS, MIT, BEN BROWN, Brown University — Solar-type stars with outer convective envelopes and stable interiors are believed to have tachoclines. As in the Sun, the tachocline is a thin shear layer thought to play an important role in the magnetic activity of these stars. We use an idealized two-dimensional model tachocline to investigate a joint instability in which the differential rotation is only stable in the absence of a magnetic field. A set of parameters are identified using Direct Numerical Simulations (DNS) that produce a cycle in which energy is transferred abruptly between kinetic and magnetic potential energy reservoirs. Elements of this cyclic behavior are replicated using Direct Statistical Simulations (DSS). Insight is thus gained into the physics prompting these sharp transitions, suggesting that they are the result of eddies interacting to form new eddies.

2:08PM R30.00009 Hydromagnetic Dynamics and Magnetic Field Enhancement in a Turbulent Spherical Couette Experiment \textsuperscript{1}, DOUGLAS STONE, MATTHEW ADAMS, ONUR KARA, DANIEL LATHROP, University of Maryland, College Park — The University of Maryland Three Meter Geodynamo, a spherical Couette experiment filled with liquid sodium and geometrically similar to the earth’s core, is used to study hydromagnetic and hydromagnetic phenomena in rapidly rotating turbulence. An external coil applies a magnetic field in order to study hydromagnetic effects relevant to the earth’s outer core such as dynamo action, while an array of 31 external Hall sensors measures the Gauss coefficients of the resulting magnetic field. The flow state is strongly dependent on Rossby number, \( Ro = (\Omega_2 - \Omega_1) / \Omega_i \), where \( \Omega_2 \) and \( \Omega_1 \) are the inner and outer sphere rotation frequencies. The flow state is inferred from the torque required to drive the inner sphere. The generation of internal toroidal magnetic field through the Rayleigh-Seifert model the magnetic and kinetic energies are equipartitioned and both ohmic and viscous dissipation are weak on the large-scales. For both cases the number is significantly smaller than that of fully nonlinear, generalized versions of the dynamo model originally developed by Childress and Soward. These models may be useful for understanding the dynamics of convection-driven dynamos in regimes that are only just becoming accessible to simulations of the full set of governing equations.

2:21PM R30.00010 Elastorotational instability in Taylor-Couette flow with Keplerian ratio as analog of the Magnetorotational Instability\textsuperscript{1}, INNOCENT MUTABAZI, YANG BAI, OLIVIER CRUMÉYROLLE, LOMC, UMRS6294, CNRS-Université du Havre — The analogy between viscoelastic instability in the Taylor-Couette flow and the magnetorotational instability is confirmed in rotating viscoelastic flows of constant viscosity (Oldroyd-B model equations) and those of Magnetohydrodynamics (MHD). We have performed linear stability analysis of the Taylor-Couette flow with a polymer solution obeying the Oldroyd-B model. A diagram of critical states shows the existence of stationary and helicoidal modes depending on the elasticity of the polymer solution. A generalized Rayleigh criterion determines the potentially unstable zone to purely elasticity-driven perturbations. Experimental results yield four type of modes: one pure elasticity mode and three elastorotational modes that are the MRI-analog modes. Anti-Keplerian case has also been investigated. There is a good agreement between experimental and theoretical results [2].

1Work supported by the CPER and ANR-LABLEX EMC3.
linear problem exhibits potentially transient growth, yet features unconditional stability as amplitudes are determined analytically as a function of the radial stratification. Then, using as well a Kelvin modes decomposition, the viscous this set of equation is first discussed: perturbations are decomposed into Kelvin modes (also known as the shearing sheet approximation) which radial shear, (ii) a radial stratification, and (iii) a coupling between the flow and the background entropy gradient. The inviscid linear stability of disk with a radially stratified, non-magnetic zonal flow in a quasi-Keplerian balance (i.e. small pressure corrections are taken into account in the radial balance). The dynamics of the perturbations around this background flow obey a set of equations which main ingredients are: (i) a radial shear, (ii) a radial stratification, and (iii) a coupling between the flow and the background entropy gradient. The inviscid linear stability of this set of equation is first discussed: perturbations are decomposed into Kelvin modes (also known as the shearing sheet approximation) which amplitudes are determined analytically as a function of the radial stratification. Then, using as well a Kelvin modes decomposition, the viscous linear problem exhibits potentially transient growth, yet features unconditional stability as amplitudes are determined analytically as a function of the radial stratification. Then, using as well a Kelvin modes decomposition, the viscous linear problem exhibits potentially transient growth, yet features unconditional stability as $t \rightarrow \infty$. Finally, we demonstrate with 2D simulations of the viscous nonlinear problem that nonlinearity provides an energy transfer mechanism through modes that compensates the transfer induced by the linear shear. This mechanism allows for a sustained instability scenario despite the stability of the linear viscous problem.

3:13PM R30.00012 Microwaves from Extra Galactic Radio Pulsars are found to deflect at Impact Parameters corresponding to the Plasma Limbs of the Sun and Stars

Tuesday, November 24, 2015 12:50PM - 3:26PM
Session R31 Waves: Surface Waves

12:50PM R31.00001 The Wave Carpet: An Omnidirectional and Broadband Wave Energy Converter

M.-REZA ALAM, Univ of California - Berkeley — Inspired by the strong attenuation of ocean surface waves by muddy seafloors, we have designed, theoretically investigated, the performance, and experimentally tested the “Wave Carpet”: a mud-remailing synthetic seabed-mounted mat composed of vertically-acting linear springs and generators that can be used as an efficient wave energy absorption device. The Wave Carpet is completely under the water surface hence imposes minimal danger to boats and the sea life (i.e. no mammal entanglement). It is survivable against the high momentum of storm surges and in fact can perform even better under very energetic (e.g. stormy) sea conditions when most existing wave energy devices are needed to shelter themselves by going into an idle mode. In this talk I will present an overview of analytical results for the linear problem, direct simulation of highly nonlinear wave fields, and results of the experimental wave tank investigation.

1:03PM R31.00002 Cloaking water waves via an elastic buoyant carpet

AHMAD ZAREEI, MOHAMMAD-REZA ALAM, University of California, Berkeley — We propose a cylindrical cloak for gravity waves passing through an elastic floating carpet. This is achieved by a spatially variable flexural rigidity while mass density and water depth are kept constant. The cloak is deduced from transformation medium scheme and coordinate transformation of the coupled governing equation for the buoyant carpet and the fluid underneath. The major challenge is that the governing equation is not form-invariant; while transformation media scheme requires a form-invariant governing equation. We approximate the governing equation with a form-invariant equation which is exact for a homogeneous and isotropic floating carpet. We compare the results with the solution of the exact governing equation and show the scattering waves of the cylinder are significantly suppressed, hence cloaking is achieved.

1:16PM R31.00003 DNS of scalar transfer across an air-water interface during inception and growth of Langmuir circulation

AMINE HAFSI, ANDRES TEJADA-MARTINEZ, University of South Florida, FABRICE VERON, YI MA, University of Delaware, USF/UDEL COLLABORATION — Upon a blowing of a wind over an initially quiescent air-sea interface, first short capillary waves are generated which in time coexist with longer waves as part of a broad spectrum of waves. The interaction between the Stokes drift velocity induced by surface gravity waves and the mean current induced by surface wind stress leads to Langmuir turbulence (LT) characterized by Langmuir circulation (LC) consisting of parallel downwind-elongated, counter rotating vortices. Initial length scales from several centimeters when short capillary waves first appear up to tens of meters when the spectrum of waves broadens. Results are presented from direct numerical simulation (DNS) of an initially quiescent coupled air-water interface driven by an air flow with free stream speed of 5 m/s. Cases with a freely deforming interface (characterized by gravity-capillary waves giving rise to small-scale LC) and with the interface intentionally held fixed (i.e. without LC) will be compared to understand the mechanisms by which the LT enhances scalar transfer from the airstream to the waterside and bulk concentration throughout the water column. Time-permitting, we will compare our results with available laboratory physical experiments.

1:29PM R31.00004 Laboratory measurements of the inception and evolution of Langmuir Turbulence

YI MA, FABRICE VERON, University of Delaware, ANDRES TEJADA-MARTINEZ, AMINE HAFSI, University of South Florida — When wind starts to blow over a quiescent air-sea interface, both currents and surface waves are initially generated. The interaction between the wind-driven waves and currents leads to the generation of Langmuir turbulence (LC) consisting of counter rotating vortices aligned with the wind. Shortly thereafter, Langmuir turbulence (LT), that is multiple scales of LC, appear. In LT, length scales range from several centimeters when short capillary waves first appear up to tens of meters when the spectrum of waves broadens. We present results from a laboratory experiments where the evolution of the air-water interface starting from rest and the accompanying development of centimeter-scale Langmuir turbulence is investigated. We present surface infrared imagery and subsurface Particle Image Velocimetry. We show that from organized small scale LC to LT is very rapid leading to intense surface mixing whereby momentum initially transferred to the surface through viscosity efficiently mixes the near surface layers. Subsurface turbulence measurements are presented in the context of scalar (gas) flux through the air-water interface.
1:42PM R31.00005 Influence of wave age on the structure of the airflow above surface waves. MARC BUCKLEY, FABRICE VERON, University of Delaware — The role of the surface waves on the airflow dynamics is known to be significant but our physical understanding remains incomplete. In this talk, we present detailed airflow measurements taken in the laboratory for 17 different wind-wave conditions with wave ages $C_p/u^*$ ranging from 1.4 to 66.7. For these experiments, a combined Particle Image Velocimetry (PIV) and Laser Induced Fluorescence (LIF) technique was developed. Two-dimensional airflow velocity fields were obtained as low as 100 μm above the water interface. When the wind stress is too weak to generate surface waves, the mean velocity profile follows the law of the wall. When waves are present, turbulence and horizontal velocity are air is ejected away from the surface, and high velocity fluid is swept downward. Airflow separation is observed above young wind waves ($C_p/u^* < 3.7$) and the resulting spanwise vorticity layers detached from the surface, produce intense wave coherent turbulence. On average, the airflow is shielded downwound of wave crests, above the critical surface (defined by $U(z_c) = C_p$). Below $z_c$, the coupling of the airflow with the waves causes a reversed, upward shielding effect. Finally, we also show preliminary field measurements.

1:55PM R31.00006 Surface waves in a square container due to its resonant horizontal elliptic motion, MITSUAKI FUNAKOSHI, AI HIRAMITSU, Kyoto University — Surface waves in a square container due to its resonant horizontal elliptic or linear motion are investigated theoretically. The motion of the container is characterized by the ratio, expressed as $\tan \phi$, of the length of the minor axis to the length of the major axis of its elliptic orbit, and by the angle $\theta$ between the directions of the major axis and one of its sidewalls. Using the reductive perturbation method, nonlinear time evolution equations for the complex amplitudes of two degenerate modes excited by this motion are derived with the inclusion of linear damping. When $\tan \phi$ is small, for any $\theta$ these equations have two kinds of stable stationary solutions corresponding to regular co-rotating waves whose direction of rotation is the same as that of the container, and regular counter-rotating waves of the opposite direction of rotation. As $\tan \phi$ increases to one, the region of forcing frequency in which stable regular counter-rotating waves are observed shrinks and then disappears for $\theta$. Solutions with chaotic or periodic slow variations in amplitude and phase of excited surface waves are also obtained for forcing frequencies where no stable stationary solutions exist.

2:08PM R31.00007 PTV measurements of Lagrangian particle transport by surface gravity wave groups, TON VAN DEN BREMER, University of Edinburgh, COLIN WHITTAKER, ALISON RABY, University of Plymouth, PAUL TAYLOR, University of Oxford — We present detailed PTV (particle tracking velocimetry) measurements of the Lagrangian transport and trajectories of neutrally buoyant particles underneath two-dimensional surface gravity wave groups in a laboratory flume. By focussing our attention on wave groups of moderate steepness, we confirm the predictions of standard second-order multi-chromatic wave theory, in which the body of fluid satisfies the potential flow equations. Particles near the surface are transported forwards and their motion is dominated by Stokes drift. Particles at sufficient depth are transported backwards by the Eulerian return current that was first described by Longuet-Higgins & Stewart (1962) and forms an inseparable counterpart of Stokes drift for surface wave groups ensuring the (irrotational) mass balance holds. Finally, we provide experimental validation of a simple scaling relationship, derived based on the assumption of separation of scales, for the transition depth: the depth above which Lagrangian particles are transported forwards by the Stokes drift and below which such particles are transported backwards by the return current. We present results for a range of effective water depths.

2:21PM R31.00008 Measurements of wind-waves under transient wind conditions. LEV SHEMER, ANDREY ZAVADSKY, Tel-Aviv University — Wind forcing in nature is always unsteady, resulting in a complicated evolution pattern that involves numerous time and space scales. In the present work, wind waves in a laboratory wind-wave flume are studied under unsteady forcing. The variation of the surface elevation is measured by capacitance wave gauges, while the components of the instantaneous slope surface in across-wind and along-wind directions are determined by a regular or scanning laser slope gauge. The locations of the wave gauge and of the laser slope gauge are separated by few centimeters in across-wind direction. Instantaneous wind velocity was recorded simultaneously using Pitot tube. Measurements are performed at a number of fetches and for different patterns of wind velocity variation. For each case, at least 100 independent realizations were recorded for a given wind velocity variation pattern. The accumulated data sets allow calculating ensemble-averaged values of the measured parameters. Significant differences between the evolution patterns of the surface elevation and of the slope components were found. Wavelet analysis was applied to determine dominant wave frequency of the surface elevation and of the slope variation at each instant. Corresponding ensemble-averaged values acquired by different sensors were computed and compared. Analysis of the measured ensemble-averaged quantities at different fetches makes it possible to identify different stages in the wind-wave evolution and to estimate the appropriate time and length scales.

2:34PM R31.00009 Laneburg modified lens for surface water waves, HELENE PICHARD, Laboratoire de Physique et Mécanique des Milieux Hétérogènes, PMMH/ESPCI, Paris, AGNES MAUREL, Institut Langevin LOA, ESPCI, Paris, PHILLIPPE PETITJEANS, Laboratoire de Physique et Mécanique des Milieux Hétérogènes, PMMH/ESPCI, Paris, PAUL MARTIN, Department of Applied Mathematics and Statistics, Colorado School of Mines, USA, VINCENT PAGNEUX, Laboratoire d’Acoustique de l’Université du Maine, Le Mans — It is well known that when the waves pass across an elevated bathymetry, refraction often results in amplification of waves behind it. In this sense, focusing of liquid surface waves can be used to enhance the harvest efficiency of ocean power. An ocean wave focusing lens concentrates waves on a certain focal point by transforming straight crest lens of incident waves into circular ones just like an optical lens. These devices have attracted ocean engineers and are promising because they enable the effective utilization of wave energy, the remaining challenge being to increase the harvest efficiency of the lens. In this work, in order to improve well known focusing of surface liquid waves by lens, the propagation of liquid surface waves through a Laneburg modified lens is investigated. The traditional Laneburg lens is a rotationally symmetric lens with a spatially varying refractive-index profile that focuses an incident plane wave on the rim of the lens. The modified Laneburg lens allows to choose the position of the focal point, which can lie inside or outside the lens. This new degree of freedom leads to enhanced focusing and tunable focusing. The focusing of linear surface waves through this lens is investigated and is shown to be more efficient than classical profile lenses.

2:47PM R31.00010 Surfboard Dynamics, ELINE DEHANDSCHOEWERCKER, Laboratoire d’Hydroodynamique de l’Ecole Polytechnique (LadHyX), DAVID QUERE, Laboratoire de Physique et Mecanique des Milieux Heterogenes (PMMH), CHRISTOPHE CLANET, Laboratoire d’Hydrodynamique de l’Ecole Polytechnique (LadHyX) — There are two main phases in surfing: catching and riding the wave. Field observations reveal that the board shape and mass distribution play a major role in both phases. To understand and optimize wave transport, we have developed an experimental setup that allows us to control the different physical parameters. A wave-maker generates either propagating periodic waves (whose wavelength and amplitude are accurately controlled) or breaking waves (whose height and velocity can be changed). Balsa boards (with different aspect ratios, mass distributions and submerged volumes) are used as floating bodies with variable friction on the water waves. We thus study the motion of boards towed onto waves. We first focus on the conditions needed for the board to be captured by the wave. We also determine the surfboard dynamics by using a force sensor responsive to the propulsive force of the wave on the surfboard. Finally, we show that the main parameters that affect this force are the slope of the wave and the shape of the board.
3:00PM R31.00011 Resonant triad interactions of acoustic–gravity waves, USAMA KADRI, T.R. AKYLAS, MIT — Surface–acoustic wave disturbances in water of constant depth over a rigid bottom, due to the combined action of gravity and compressibility, are studied. In the linear theory, apart from free-surface (gravity) waves, there is also a countable infinity of acoustic (compression) modes. As the sound speed in water, typically, far exceeds the maximum gravity wave phase speed, these two types of modes feature vastly different spatial and/or temporal scales, and their linear coupling is weak. It is possible, however, to realize significant energy exchange between gravity and acoustic waves via nonlinear interactions. This scenario is analyzed for resonant wave triads that comprise two counter-propagating gravity waves and a long-crested acoustic mode. Owing to this disparity in length scales, the interaction time scale as well as the form of the amplitude evolution equations differ from those of a standard resonant triad. In the case of a perfectly tuned triad of uniform monochromatic wave trains, nearly all the energy initially in the gravity waves can be transferred to the acoustic wave. This mechanism, however, is less efficient when the interacting waves are modulated wavepackets.

3:13PM R31.00012 Quantifying wave-breaking dissipation using nonlinear phase-resolved wave-field simulations with a phenomenological-based wave breaking model, YUSHENG QI, DICK YUE, Department of Mechanical Engineering, MIT — We use direct nonlinear phase-resolved simulations based on a High-Order Spectral (HOS) method (Dommermuth & Yue 1987) to understand and quantify wave-breaking dissipation in the evolution of general irregular short-crested wave-fields. We achieve this by incorporating a robust phenomenological-based wave breaking model in HOS simulations to account for energy dissipation. This model can automatically simulate the onset of wave breaking, and the simulated wave-breaking dissipation strength differentiates corresponding to different wave breaking type (such as spilling or plunging breaking waves). The efficacy of this model is confirmed by direct comparisons against measurements for the energy loss in 2D and 3D breaking events. By comparing simulated wave-fields with and without the dissipation model in HOS, we obtain the dissipation field, which provides the times, locations and intensity of wave breaking events. From the dissipation field we further calculate the distribution of total length of breaking wave front per unit surface area per unit increment of breaking velocity (Phillips 1985), and obtain qualitative agreement with Phillips theoretical power-law.

Tuesday, November 24, 2015 12:50PM - 3:26PM — Session R32 General Fluid Dynamics: Rotating Flows and Multi-Physics Phenomena

12:50PM R32.00001 The hydrodynamic Phase Transitions and Creeping Flows in Cavities, MIRON KAUFMAN, PETRU S. FODOR, Cleveland State University — We discuss the analogy between the stream line function of creeping flows in rectangular cavities and the thermodynamic potential critical points and at phase transitions. Assuming no-slip boundary conditions, the corners of the rectangular cavity are stationary (fixed) points. We analyze two such points: 1. Corner where one wall is moving and the other is stationary; 2. Corner where both walls are stationary. The first one is analogous to a to a first-order transition (discontinuity) point while the second one is analogous to a thermodynamic critical point (second-order transition). Moffatt eddies, which impede mixing [P. S. Fodor, M. Kaufman, Proceedings of PPS-30, AIP Conf. Proc. 1664 (2015)], are present in the neighborhood of the second stationary point. The results discussed here are based on numerical solutions of the Navier-Stokes equations combined with analytical work valid in the vicinity of the stationary points.

1:03PM R32.00002 Heat transfer analysis in rotating spherical shells1, ARES CABELLO, RUBEN AVILA, Univ Nac Autonoma de Mexico — The study of flow patterns within rotating spherical annular geometries with natural convection is essential to understand the internal dynamics of the planets. We investigate the convective flows and the heat transfer rate in a spherical gap in which a temperature difference between the inner sphere and the outer sphere is present. A self gravity field which varies as a function of $1/r^n$ (where $r$ is the radial position and the integer exponent $n$ has the values 2,3,4,5) is assumed. The Boussinesq fluid equations are solved by using a spectral element method (SEM). To avoid the singularity at the poles, the cubed-sphere algorithm is used to generate the spherical grids at the surface. Finally, both spherical Ekman and Rayleigh numbers, that there exists a high correlation between the azimuthal motion of both the Busse cells and the zones where the maximum surface heat fluxes occur. The azimuthal position, as a function of time, of the maximum heat flux zones (which are located symmetrically with respect to the equator), allows to speculate on the nature of the phenomena occurring (in geological times) on the surface of the terrestrial planets.

1Thanks to DGAPA-PAPIIT project: IN117314-3

1:16PM R32.00003 Convective flow patterns in inclined rectangular cavities with rotation1, RUBEN AVILA, DIANA PEREZ-ESPEJEL, Univ Nac Autonoma de Mexico — The natural convection in inclined three dimensional rectangular cavities with rotation is numerically analyzed using a spectral element method. When the rate of rotation ($Ta$ number) is equal to zero, the critical Rayleigh number $Ra_c$ for the onset of transverse or longitudinal rolls is obtained by solving (using the Tau-Chebyshev spectral method) the equations of the linear stability theory. In the numerical approach, the rotation is imposed once the steady state of the longitudinal or transverse rolls is attained. The cavity rotates around an axis that is orthogonal to its cold and hot surfaces, and passes through the center of these surfaces. In all the analyzed cases, the tilted angle $\delta$, from the horizontal, varies in the interval $0^\circ < \delta < 90^\circ$ (the cavity is heated from its lower surface, then an unstable condition prevails) and $90^\circ < \delta \leq 180^\circ$ (the cavity is heated from its upper surface, then a stable condition prevails). We report the influence of the $Ta$ number on the critical $Ra$ number, the average Nusselt number (evaluated at the hot surface), and the flow patterns in the tilted cavity.

1DGAPA-PAPIIT Project: IN117314-3

1:29PM R32.00004 Interpreting global behavior of quasi-Keplerian flows as a response to boundary forcing, E. M. EDLUND, Massachusetts Institute of Technology, H. Ji, Princeton Plasma Physics Laboratory — A series of experiments conducted in the Hydrodynamic Turbulence Experiment (HTX), a modified Taylor-Couette device, have explored the response of the azimuthal flows to the forcing imposed by the boundaries. The HTX device has rings on the axial end-caps that can take speeds different from that of the inner and outer cylinders. This extra degree of freedom allows us to tune the mean flow profiles, with the possibility of achieving flows remarkably close to the ideal Couette profile that are expected in the absence of axial boundaries. These “optimized” cases have the interesting property that the azimuthal velocity profiles are effectively independent of Reynolds number. In contrast, non-optimized cases show progressive departure from ideal Couette as the Reynolds number is increased. We present a model that captures this Reynolds number dependence and interpret this from the perspective of angular momentum flux across the boundaries. By varying the boundary components, we also show that optimized flows can only be achieved when there exists pressure balance between the boundary and the bulk. These observations have important implications for the design of Taylor-Couette experiments that attempt to make connections to astrophysics at large Reynolds numbers.
1:42PM R32.00005 Transient growth and its consequences in rotating channel flow\textsuperscript{1}

\textit{Sharath Jose, Tata Institute of Fundamental Research, Centre for Interdisciplinary Sciences, Vishnu Prasad, Shell Technology Center Bangalore, Benoit Pier, LMFA (CNRS-Université de Lyon), France, Rama Govindarajan, Tata Institute of Fundamental Research, Centre for Interdisciplinary Sciences — We know that pressure-driven flow through a channel, which is being rotated about its spanwise coordinate, is unstable for a range of rotation numbers Ro. The critical Reynolds number is very sensitive to the rotation rate. We present here nonmodal stability characteristics of the system in the parameter regime where exponential instabilities do not exist. We show that transient growth is markedly different at low and high Ro, with high growth rates and asymmetric streamwise structures at low Ro and spanwise structures created by the far weaker Orr mechanism in operation at high Ro. The latter is a demonstration of the Taylor-Proudman theorem. In the nonlinear regime, except when the rotation rate is very high, chaotic flows of varying strength are obtained. At high Ro weak structures confined to one side of the channel give way to a completely laminar profile.}

\textsuperscript{1}Funding from CEFIPRA is gratefully acknowledged

1:55PM R32.00006 Numerical and Experimental study of secondary flows in a rotating two-phase flow: the tea leaf paradox\textsuperscript{1}

\textit{Antoni Calderer, St. Anthony Falls Lab., University of Minnesota, Douglas Neal, Richard Prevost, LaVision Inc., Arno Mayrhofer, Institute for Water Management, Hydrology and Hydraulic Engineering, Alan Lawrenz, John Foess, Department of Mechanical Engineering, Michigan State University, Fotis Sotiropoulos, St. Anthony Falls Lab., University of Minnesota — Secondary flows in a rotating flow in a cylinder, resulting in the so called tea leaf paradox, are fundamental for understanding atmospheric pressure systems, developing techniques for separating red blood cells from the plasma, and even separating coagulated trub in the beer brewing process. We seek to gain deeper insights in this phenomenon by integrating numerical simulations and experiments. We employ the Curvilinear Immersed boundary method (CURVIB) of Calderer et al. (J. Comp. Physics 2014), which is a two-phase flow solver based on the level set method, to simulate rotating free-surface flow in a cylinder partially filled with water as in the tea leaf paradox flow. We first demonstrate the validity of the numerical model by simulating a cylinder with a rotating base filled with a single fluid, obtaining results in excellent agreement with available experimental data. Then, we present results for the cylinder case with free surface, investigate the complex formation of secondary flow patterns, and show comparisons with new experimental data for this flow obtained by Lavision.}

\textsuperscript{1}Computational resources were provided by the Minnesota Supercomputing Institute

2:08PM R32.00007 Design optimization of a vaneless “fish-friendly” swirl injector for small water turbines

\textit{Ajith Aiody, Sean D. Peterson, University of Waterloo: Department of Mechanical and Mechatronics Engineering — Small-scale hydro-electric plants are attractive options for powering remote sites, as they draw energy from local bodies of water. However, the environmental impact on the aquatic life drawn into the water turbine is a concern. To mitigate adverse consequences on the local fauna, small-scale water turbine design efforts have focused on developing “fish-friendly” facilities. The components of these turbines tend to have wider passages between the blades when compared to traditional turbines, and the rotors are designed to spin at much lower angular velocities, thus allowing fish to pass through safely. Galt Green Energy has proposed a vaneless casing that provides the swirl component to the flow approaching the rotor, eliminating the need for inlet guide vanes. We numerically model the flow through the casing using ANSYS CFX to assess the evolution of the axial and circumferential velocity symmetry and uniformity in various cross-sections within and downstream of the injector. The velocity distributions, as well as the pressure loss through the injector, are functions of the pitch angle and number of revolutions of the casing. Optimization of the casing design is discussed via an objective function consisting of the velocity and pressure performance measures.}

2:21PM R32.00008 Numerical investigation of power consumption and mixing time in a stirred vessel with regular and multiscale impellers

\textit{Salur Basbug, George Papadakis, Christos Vassilicos, Imperial College London — The flow field inside a stirred tank is obtained by means of direct numerical simulation based on finite volume method at Re=500. Two different types of four-bladed radial impellers are considered: the first one is a regular type with rectangular blades and the second one is a modified version of the former with irregular blade edges, having the same thickness and the surface area. The shaft power is averaged over more than sixty revolutions and the comparison between the two cases shows that the impeller with irregular blades has lower energy consumption. Moreover, a passive scalar is injected into the vessel for a quarter period of revolution and the scalar transport equation is solved to investigate the mixing times. The coefficient of variation of the passive scalar is averaged over the whole volume in order to obtain a quantitative indicator of the mixing progress. The homogenization curves depend on the instantaneous flow conditions due to the transient nature of the mixing process, therefore multiple curves are averaged to obtain a representative result. There are indications that irregular blades can decrease mixing time with respect to regular ones.}

2:34PM R32.00009 Experimental investigation of the flow field and power consumption characteristics of regular and fractal blade impellers in a dynamic mixer\textsuperscript{1}

\textit{K. Steiros, P.J.K. Bruce, O.R.H. Buxton, J.C. Vassilicos, Imperial College London — Experiments have been performed in an octagonal un-baffled water tank, stirred by three radial turbines with different geometry impellers: (1) regular rectangular blades; (2) single-iteration fractal blades; (3) two-iteration fractal blades. Shaft torque was monitored and the power number calculated for each case. Both impellers with fractal geometry blades exhibited a decrease of turbine power number compared to the regular one (15% decrease for single-iteration and 19% for two iterations). Phase locked PIV in the discharge region of the blades revealed that the vortices emanating from the regular blades are more coherent, have higher kinetic energy, and advect faster towards the tank’s walls where they are dissipated, compared to their fractal counterparts. This suggests a strong link between vortex production and behaviour and the energy input for the different impellers. Planar PIV measurements in the bulk of the tank showed an increase of turbulence intensity of over 20% for the fractal geometry blades, suggesting higher mixing efficiency. Experiments with pressure measurements on the different geometry blade surfaces are ongoing to investigate the distribution of forces, and calculate hydrodynamic centres of pressure.}

\textsuperscript{1}The authors would like to acknowledge the financial support given by European Union FP7 Marie Curie MULTISOLVE project (Grant Agreement No. 317260)
Leading-edge vortex trajectories under the influence of Coriolis acceleration

**2:47PM R32.00010** Leading-edge vortex trajectories under the influence of Coriolis acceleration, ERIC LIMACHER, CHRIS MORTON, DAVID WOOD, Univ of Calgary — Leading-edge vortices (LEVs) can form and remain attached to a rotating wing indefinitely, but the mechanisms of stable attachment are not well understood. Taking for granted that such stable structures do form, a practical question arises of where an LEV core persists in the body-fixed frame of reference. Noting that span-wise flow exists within the LEV core, it is apparent that a mean streamline aligned with the axis of the LEV must exist. The present work uses the Navier-Stokes equations along this steady, axial streamline in order to consider the accelerations that act in the streamline-normal direction to affect its local curvature. With some simplifying assumptions, an ordinary differential equation is derived that describes the trajectory of the axial streamline through the vortex core. Using empirical values of axial velocity in the vortex core from previous studies, it can be shown that Coriolis and centrifugal forces alone can account for the tilting of the stable LEV into the wake within several chord lengths from the root of rotation in the span-wise direction. This result supports an hypothesis that LEVs are observed at inboard locations because Coriolis force must act over a finite distance to tilt the stable LEV away from the leading edge.

**3:00PM R32.00011** A Multiscale simulation method for ice crystallization and frost growth, MIAD YAZDANI, United Tech Res Ctr — Formation of ice crystals and frost is associated with physical mechanisms at immensely separated scales. The primary focus of this work is on crystallization and frost growth on a cold plate exposed to the humid air. The nucleation is addressed through Gibbs energy barrier method based on the interfacial energy of crystal and condensate as well as the ambient and surface conditions. The supercooled crystallization of ice crystals is simulated through a phase-field based method where the variation of degree of surface tension anisotropy and its mode in the fluid medium is represented statistically. In addition, the mesoscale width of the interface is quantified asymptotically which serves as a length-scale criterion into a so-called Adaptive AMR (AAMR) algorithm to tie the grid resolution at the interface to local physical properties. Moreover, due to the exposure of crystal to humid air, a secondary non-equilibrium growth process contributes to the formation of frost at the tip of the crystal. A Monte-Carlo implementation of Diffusion Limited Aggregation method addresses the formation of frost during the crystallization. Finally, a virtual boundary based Immersed Boundary Method (IBM) is adapted to address the interaction of ice crystal with convective air during its growth.

**3:13PM R32.00012** Geometrical Scaling of an Ablative Bluff Body under Different Outer Flow Velocity and Temperature Configurations, MICHAEL ALLARD, CHRISTOPHER M. WHITE, University of New Hampshire, YVES DUBIEF, University of Vermont — Experimental results investigating the geometrical scaling and local properties of an eroding low temperature ablator (para-dichlorobenzene) are presented. The bluff body is placed in a heated open-circuit wind tunnel and the effects of incoming outer flow velocity (uniform and spatially varying) and temperature on the ablation process are investigated. Image sequencing of the projected area in the streamwise-spanwise and streamwise-wall normal flow direction are used to quantify the time evolution of the geometrical shape and compute local recession rates and curvature. The geometrical self-similarity and local recession rates are evaluated and compared to Moore et al. (Phys. Fluids (2013) 25:116602) and Huang et al. (J. Fluid Mech. (2015) 765:R3) who investigated erosion under the action of fluid shear force and dissolution, respectively.

---

**Tuesday, November 24, 2015 12:50PM - 3:26PM**

---

**Session R33 Drops: Wetting and Spreading III**

**12:50PM R33.00001** Finite time singularity in a glass, FRANCOIS GALLAIRE, FRANCESCO VIOLA, Laboratory of Fluid Mechanics and Instabilities, STI, EPFL, Lausanne, Switzerland, BENJAMIN DOLLET, Institut de Physique de Rennes, UMR 6251 CNRS/Universite de Rennes 1, Rennes, France, PIERRE-THOMAS BRUN, Department of Mathematics, Massachusetts Institute of Technology, Cambridge, Massachusetts 02139, USA — Using a simple liquid-foam sloshing system as prototype, we demonstrate that nonlinear friction effects, resulting from the multiscale nature of moving contact lines, become predominant at low amplitude and result in a finite-time arrest of the oscillations. This result is in strong contrast with the classical exponential relaxation induced by linear damping. We proceed to derive a model for the oscillation of foam in a cylinder accounting for capillary effects near the container walls, which we solve using multiple scales analysis. These results help rationalize our experimental observations and reveal the importance of sublinear effects in perturbation theory.

**1:03PM R33.00002** Spreading of viscoelastic droplets, YULI WANG, DO-QUANG MINH, GUSTAV AMBERG, KTH Mechanics — We intend to gain new insights into the spreading dynamics of viscoelastic droplets from a numerical perspective. Focusing on the Giesekus droplet and the Oldroyd-B droplet, we simulated the viscous spreading and the spreading after impacting on a horizontal surface. The results qualitatively agree with some experimental observations on Boger fluids and shear-thinning fluids. We discuss how shear-thinning and elasticity influence the contact line motion, given detailed information on the flow field, the stress distribution and the contact line morphology in the near vicinity of the contact line. The results suggest that viscoelastic droplets can spread faster than their Newtonian counterparts. The spreading speed of the Oldroyd-B fluid shows dependence on elasticity while the one of the Giesekus droplet does not.

**1:16PM R33.00003** The inner region of the moving contact line - diffusive and nanoscale models, ANDREAS NOLD, Imperial College London, DAVID N. SIBLEY, Loughborough University, BEN D. GODDARD, The University of Edinburgh, SERAFIM KALLIADASIS, Imperial College London — Much of the work within the Complex Multiphase Systems group [1] at Imperial College London for the last number of years has been to understand the moving contact line problem. In [2], it was shown that contrary to the classical asymptotic theory at the moving contact line, the intermediate region is in fact an overlap region between the inner and the outer regions. Here, we investigate the inner region independently for the Navier-Stokes/ Cahn-Hillard (NS/CH) model for binary fluids, as well as dynamic density functional theory (DDFT) for a simple fluid. We show that in the NS/CH model, the overlap region is recovered in the sharp-interface limit, and we link the slip length to the mobility of the system. In contrast, DDFT, which is based on statistical mechanics of fluids, allows to incorporate nanoscale details. Results are presented for advancing and receding contact lines for a wide range of contact angles. The numerical method employs spectral methods in an unbounded domain along the surface. Advantages are discussed, both for differential and integral DDFT equations. [1] http://www3.imperial.ac.uk/complexmultiphasesystems. [2] Sibley, D.N., Nold, A. and Kalliadasis, S. J. Fluid Mech. 764, 445 (2015).

---

1We acknowledge financial support from ERC Advanced Grant No. 247031 and Imperial College through a DTG International Studentship.
1:29PM R33.00004 Numerical study of liquid-gas flow on complex boundaries

SHENG WANG, OLIVIER DESJARDINS, Sibley School of Mechanical and Aerospace Engineering, Cornell University — Simulation techniques for liquid-gas flows near solid boundaries tend to fall into two categories, either focusing on accurate treatment of the phase interface away from wall, or focusing on detailed modeling of contact line dynamics. In order to fill the gap between these two categories and to simulate liquid-gas flows in large scale engineering devices with complex boundaries, we develop a conservative, robust, and efficient framework for handling moving contact lines. This approach combines a conservative level set method to capture the interface, an immersed boundary method to represent the curved boundary, and a macroscopic moving contact line model. The performance of the proposed approach is assessed through several simulations. A drop spreading on a flat plate and a circular cylinder validate the equilibrium contact angle. The migration of a drop on an inclined plane is employed to validate the contact line dynamics. The framework is then applied to perform a 3D simulation of the migration of a drop through porous media, which consists of irregular placed cylinders. The conservation error is shown to remain small for all the simulations.

1:42PM R33.00005 Bifurcation analysis of the behavior of partially wetting liquids on a rotating cylinder

DIMITRIS MAMALIS, KHELLIL SEFIANE, The University of Edinburgh, KIRTI CHANDRA SAHU, Indian Institute of Technology Hyderabad, THIELE, University of Muenster — We analyze the behavior of a partially wetting liquid on a rotating cylinder using the model of Thiele [1] that takes into account the effects of gravity, viscosity, rotation, surface tension and liquids wettability. Such a system can be considered as a prototype for many other systems with a spatial heterogeneity and a lateral driving force in the proximity of a first- or second-order phase transition. Thiele [1] found that a partially wetting drop on a rotating cylinder undergoes a depinning transition as the rotation speed is increased, whereas for ideally wetting liquids the behavior changes monotonically. We analyze in detail the transition in the bifurcation behavior for partially wetting liquids as the wettability of the liquid decreases, and, in particular, how the global bifurcation related to depinning of drops is created when increasing the contact angle. We employ various numerical continuation techniques that allow us to track stable and unstable steady and time-periodic states. We support our findings by time-dependent numerical simulations and asymptotic analysis of steady-state and time-periodic solutions for large rotation numbers. [1] U. Thiele, “On the depinning of a droplet of partially wetting liquid on a rotating cylinder,” J. Fluid Mech. 671, 121-136 (2011)

1:55PM R33.00006 Drop spreading on under-liquid substrates: Inertial to Viscous Regimes

NAGA SIVA KUMAR GUNDA, SURJYASISH MITRA, SUSHANTA MITRA, Micro & Nano-scale Transport Laboratory, Department of Mechanical Engineering, Lassonde School of Engineering, York University, Toronto, ON, Canada — Spreading of liquid drops on a substrate placed in air medium is a well understood phenomenon from the theory of minimization of surface energy. This process has been studied rigorously over the past few decades due to its wide array of applications like printing, coating, microfluidic devices as well as it presents the challenging problem of contact line dynamics. However, many applications involve spreading of oil recovery, emulsions, liquid-liquid displacement in porous media, etc. warrants the need to study this phenomenon in the presence of a surrounding liquid medium. In the present study, an experimental investigation of the spreading process of a laser-oil drop on an ITO-coated glass substrate submerged inside water has been conducted. The experimental investigation reveals two different regimes of under-liquid drop spreading, one which is dominated by inertia and a later regime, where viscous effects, with contributions from both the drop and surrounding liquid, takes over. In doing so, we have identified the characteristic time scales for each regime and also the transition point from one regime to another.

2:08PM R33.00007 Contact line moving over a sinking sphere

SEONG JIN KIM, JIM AN, Virginia Tech, KAMEL FEZZAA, TAO SUN, Argonne National Laboratory, SUNGHWAN JUNG, Virginia Tech — Spreading dynamics of a contact line over a sinking sphere with a constant speed into a liquid reservoir are studied both experimentally and theoretically. A high-speed camera system with X-ray illumination is employed to accurately characterize contact line motions. Over a range of Reynolds number from 30 to 1000, the spreading speed of the contact line is linearly dependent on the sinking speed of the sphere. A simple scaling equation from the force balance between the drag-induced pressure and inertia agrees with the experimental results. In addition, the numerical solution of Navier-Stokes equation, showing pressure distribution and streamline near the sinking sphere, validates our scaling equation.

2:21PM R33.00008 ABSTRACT WITHDRAWN

2:34PM R33.00009 Numerical simulations of the moving contact line problem using a diffuse-interface model

MUHAMMAD AFZAAL, Imperial College London, UK, DAVID SIBLEY, Loughborough University, UK, PEW EW DUNCAN, PETR YATSYSHIN, MIGUEL A. DURÁN-OLIVENCIA, ANDREAS NOLD, Imperial College London, UK, NIKOS SAVVA, Cardiff University, UK, MARKUS SCHMUCK, Heriot-Watt University, UK, SERAFIM KALLIADASIS, Imperial College London, UK — Moving contact lines are ubiquitous phenomenon both in nature and in many modern technologies. One prevalent way of numerically tackling the problem is with diffuse-interface (phase-field) models, where the classical sharp-interface model of continuum mechanics is relaxed to one with a finite thickness fluid-fluid interface, capturing physics from mesoscopic length scales. The present work is devoted to the study of the contact line between two fluids confined by two parallel plates, i.e. a dynamically moving meniscus. Our approach is based on a coupled Navier-Stokes/Cahn-Hilliard model. This system of partial differential equations allows a tractable numerical solution to be computed, capturing diffusive and advective effects in a prototypical case study in a finite-element framework. Particular attention is paid to the static and dynamic contact angle of the meniscus advancing or receding between the plates. The results obtained from our approach are compared to the classical sharp-interface model to elicit the importance of considering diffusion and associated effects.

1We acknowledge financial support from European Research Council via Advanced Grant No. 247031.

2:47PM R33.00010 Non-isothermal spreading dynamics of self-rewetting droplets

DIMITRIS MAMALIS, KHELLIL SEFIANE, The University of Edinburgh, KIRTI CHANDRA SAHU, Indian Institute of Technology Hyderabad, GEORGE KARAPETSA, National Technical University of Athens, OMAR K. MATAR, Imperial College London — We study the spreading dynamics of droplets on uniformly heated substrates. More specifically, we consider the case of binary alcohol mixtures which exhibit a non-monotonic dependence of the surface tension on temperature; these systems are often referred in the literature as self-rewetting fluids. We show through experiments that the early-stage spreading exponents depend non-monotonically on the substrate temperature in contrast to the monotonic dependence of pure liquids. In addition, we observe through the use of IR thermography visualization the formation of spontaneous travelling waves which develop along and across the free surface of the evaporating droplet and influence the spreading behaviour. Finally, we develop a theoretical model based on lubrication theory and derive an evolution equation for the interface accounting for capillarity and thermocapillarity. Using this model we investigate the effect of varying droplet wettability, which is linked to the temperature of the solid surface, on the spreading dynamics.
3:00PM R33.00011 How to include the nonlinear Cox-Voinov law into sloshing dynamics? A weakly non linear approach, FRANCESCO VIOLA, LFMI EPFL, PIERRE-THOMAS BRUN, MIT, FRANÇOIS GALLAIRE, LFMI EPFL — Fluid sloshing in a glass is a common example of damped oscillator, with the frequency derived in the potential flow limit. The damping rate is then evaluated considering the viscous dissipation at the wall, in the bulk and at the free surface, respectively. This classical theoretical result however differs from what is often seen in the laboratory when the attenuation of gravity waves happens in a small basin. In particular, the damping rate is found to increase as the sloshing amplitude decreases. Here we show that this enhanced damping is due to capillary forces at the contact line between the liquid and the container. The angle $\theta_{12}$ made by the liquid interface with the container walls (contact angle) is modeled as a non-linear function of the interface speed $U$, (Cox-Voinov law $\theta_{12} \propto U$). We propose a multiple scale expansion scheme to consistently derive an amplitude equation using the Cox-Voinov law as boundary condition at the moving interface. The zero order problem reduces to the classical static meniscus problem, while the first order problem yields an eigenvalue problem defining the viscous sloshing modes. At an higher order, a compatibility condition has to be enforced, yielding an amplitude equation. Solving the latter, we recover the expected increase of the damping rate as the sloshing amplitude decreases, an effect thus attributed to capillary effects.

3:13PM R33.00012 Obtaining macroscopic quantities for the contact line problem from Density Functional Theory using asymptotic methods1, DAVID SIBLEY, Department of Mathematical Sciences, Loughborough University, ANDREAS NOLD, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London — Density Functional Theory (DFT), a statistical mechanics of fluids approach, captures microscopic details of the fluid density structure in the vicinity of contact lines, as seen in computations in our recent study [1]. Contact lines describe the location where interfaces between two fluids meet solid substrates, and have stimulated a wealth of research due to both their ubiquity in nature and technological applications and also due to their rich multiscale behaviour. Whilst progress can be made computationally to capture the microscopic to mesoscopic structure from DFT, complete analytical results to fully bridge to the macroscale are lacking. In this work, we describe our efforts to bring asymptotic methods to DFT to obtain results for contact angles and other macroscopic quantities in various parameter regimes. [1] A. Nold, D. N. Sibley, B. D. Goddard and S. Kalliadasis, “Fluid structure in the immediate vicinity of an equilibrium three-phase contact line and assessment of disjoining pressure models using density functional theory” Phys. Fluids 26, 072001 (2014).

Tuesday, November 24, 2015 12:50PM - 3:00PM Session R35 Drops, Bubbles and Foams: Collective Dynamics Ballroom B - Michael Rother, University of Minnesota Duluth

12:50PM R35.00001 The Effect of Slight Deformation on Binary Interactions of Sedimenting Drops with Partially Mobile Interfaces, MICHAEL ROTHER, University of Minnesota Duluth — Collision efficiencies are determined for two slightly deformable drops in gravitational motion with van der Waals forces and negligible inertia, using methodology borrowed from matched asymptotic expansions. The drop interfaces are free of surfactant and partially mobile, so that the drop-to-medium viscosity ratio is $\sim 10$. The outer solution for two sedimenting, spherical drops in the absence of van der Waals forces is used to find the driving force for the inner region. In the inner region, where deformation is confined to the area of close approach, the appropriate thin-film equations for partially mobile interfaces, including attractive molecular forces, are used to find the evolution of the gap between the drops. Solution of this system of integro-differential equations coupling the flow inside the drops and that within the small gap allows decoupling of trajectories leading to drop coalescence and separation. Full boundary-integral simulations have verified the accuracy of this quasi-asymptotic approach in the case of drops with partially mobile interfaces. This technique may also prove suitable for contaminated drops in some limiting cases, such as incompressible surfactant.

1:03PM R35.00002 Colloidal building blocks made with microfluidics, JOSHUA RICOUVIER, PATRICK TABELING, ESPCI — We discovered a novel strategy, using microfluidics, for designing colloidal building blocks. The strategy is based on the entrainment of droplets in microfluidic channels, where the droplets spontaneously aggregate and rapidly rearrange into an ensemble of well-defined structures. The physical origin of the phenomenon is a coupling between depletion forces and droplet-droplet dipolar interactions. By varying the flow parameters, we succeed in designing a wide array of building blocks such as chains, triangles, diamonds, tetrahedrons, and heterotrimers. These well-controlled structures possess geometrical, chemical, and/or magnetic anisotropies, which enable directional bonding. We demonstrate monodisperse ($98\%$) production of pentamers and production of $10^5$ monodisperse trimers. The liquid clusters can be photo-polymerized in situ and produced via a continuous flow process. The particles of the solidified clusters are tightly held together by sub-micrometric polymerized cords attached in-between them. We believe that this robust and inexpensive method could meet the demand for the efficient production of colloidal building blocks for various applications.

1:16PM R35.00003 Bursting of a bubble confined in between two plates1, MAYUKO MURANO, NATSUKI KIMONO, KO OKUMURA, Ochanomizu Univ. — Rupture of liquid thin films, driven by surface tension, has attracted interests of scientists for many years [1-4]. It is also a daily phenomenon familiar to everyone in the form of the bursting of soap films. In recent years, many studies in confined geometries (e.g. in a Hele-Shaw cell) have revealed physical mechanisms of the dynamics of bubbles and drops [5]. As for a liquid film sandwiched in between another liquid immiscible to the film liquid in the Hele-Shaw cell, it is reported that the thin film bursts at a constant speed and the speed depends on the viscosity of the surrounding liquid when the film is less viscous, although a rim is not formed at the bursting tip; this is because the circular symmetry of the hole in the bursting film is lost [6]. Here, we study the bursting speed of a thin film sandwiched between air instead of the surrounding liquid in the Hele-Shaw cell to seek different scaling regimes. By measuring the bursting velocity and the film thickness of an air bubble with a high speed camera, we have found a new scaling law in viscous regime. [1] L. Rayleigh, Nature 44 (1891) [2] F. E. Culick, J. Appl. Phys. 31 (1960) [3] G. Debregeas, P. Martin and F. Brochard-Wyart, Phys. Rev. Lett. 75 (1995) [4] E. Reyssat and D. Quere, Europhys. Lett. 76 (2006) [5] Maria YOKOTA and Ko OKUMURA, Proc. Nat. Acad. Sci. 108 (2011) [6] Ayako ERI and Ko OKUMURA, Phys. Rev. E Rapid Communication, 82 (2010)

1This research was partly supported by ImPACT Program of Council for Science, Technology and Innovation (Cabinet Office, Government of Japan).
1:29PM R35.00004 Approaching behavior of a pair of spherical bubbles in quiescent liquids. TOSHIYUKI SANADA, HIROAKI KUSUNO, Shizuoka University — Some unique motions related bubble-bubble interaction, such as equilibrium distance, wake induced lift force, have been proposed by theoretical analysis or numerical simulations. These motions are different from the solid spheres like DKT model (Drafting, Kissing and Tumbling). However, there is a lack of the experimental verification. In this study, we experimentally investigated the motion of a pair of bubbles initially positioned in-line configuration in ultrapure water or an aqueous surfactant solution. The bubble motion were observed by two high speed video cameras. The bubbles Reynolds number was ranged from 50 to 300 and bubbles hold the spherical shape in this range. In ultrapure water, initially the trailing bubble deviated from the vertical line on the leading bubble according to the wake of the leading bubble. And then, the slight difference of the bubble radius changed the relative motion. When the trailing bubble slightly larger than the leading bubble, the trailing bubble approached to the leading bubble due to it’s buoyancy difference. The bubbles attracted and collided only when the bubbles rising approximately side by side configuration. In addition, we will also discuss the motion of bubbles rising in an aqueous surfactant solution.

1:42PM R35.00005 Coalescence preference in dense packing of bubbles1, YESEUL KIM, BOPIL GIM, SKKU Advanced Institute of Nanotechnology(SAINT), Sungkyunkwan University, SU JIN LIM, Department of Bio and Brain Engineering, Korea Advanced Institute of Science and Technology (KAIST), BYUNG MOOK WEO, SKKU Advanced Institute of Nanotechnology(SAINT), Sungkyunkwan University — Coalescence preference is the tendency that a merged bubble from the contact of two original bubbles (parent) tends to be near to the bigger parent. Here, we show that the coalescence preference can be blocked by densely packing of neighbor bubbles. We use high-speed high-resolution X-ray microscopy to clearly visualize individual coalescence phenomenon which occurs in micro scale seconds and inside dense packing of microbubbles with a local packing fraction of ~40%. Previous theory and experimental evidence predict a power of -5 between the relative coalescence position and the parent size. However, our new observation for coalescence preference in densely packed microbubbles shows a different power of -2. We believe that this result may be important to understand coalescence dynamics in dense packing of soft matter. This work (NRF-2013R1A2A04008115) was supported by Mid-career Researcher Program through NRF grant funded by the MEST and also was supported by Ministry of Science, ICT and Future Planning (2009-0082580) and by Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry and Education, Science and Technology (NRF-2012R1A6A3A04039257).

1This work (NRF-2013R1A2A04008115) was supported by Mid-career Researcher Program through NRF grant funded by the MEST (South Korea) and also was supported by Sungkyun Research Fund, Sungkyunkwan University, 2014.

1:55PM R35.00006 Parallelizable flood fill algorithm and corrective interface tracking approach applied to the simulation of multiple finite size bubbles merging with a free surface. NATHAN LAFFERTY, ETH Zurich, HASSAN BADREDDINE, BOJAN NICENO, Paul Scherrer Institute, HORSTMICHAEL PRASSER, ETH Zurich — A parallelizable flood fill algorithm is developed for identifying and tracking closed regions of fluids, dispersed phases, in CFD simulations of multiphase flows. It is used in conjunction with a newly developed method, corrective interface tracking, for simulating finite size dispersed bubbles, where the bubbles are too small relative to the grid to be simulated accurately with interface tracking techniques and too large relative to the grid to use Lagrangian particle tracking techniques. The latter situation arising if local bubble induced turbulence is resolved, or modeled with LES. With corrective interface tracking the governing equations are solved on a static Eulerian grid. A correcting force, derived from empirical correlation based hydrodynamic forces, is applied to the bubble which is then advected using interface tracking techniques. This method results in accurate fluid-gas two-way coupling, bubble shapes, and terminal rise velocities. The flood fill algorithm and corrective interface tracking technique are applied to an air/water simulation of multiple bubbles rising and merging with a free surface. They are then validated against the same simulation performed using only interface tracking with a much finer grid.

2:08PM R35.00007 How does a pressure-driven foam jam in a straight channel? SHUBHA TEWARI, University of Massachusetts, Amherst and TIFR Centre for Interdisciplinary Sciences, Hyderabad, KARTHIK MENON, RAMA GOVINDARAJAN, TIFR Centre for Interdisciplinary Sciences, Hyderabad — A Newtonian fluid and a foam flow differently. We highlight this contrast in the pressure-driven flow of a foam through a straight channel. Unlike a Newtonian fluid, a foam in a straight channel does not flow below a threshold driving force. Just above this yield threshold, the flow is intermittent (stick-slip), and crosses over to smooth flow as the driving force is increased. We report on a numerical investigation of these different regimes using a modified version of Durian’s bubble model with an added short-ranged attraction potential to account for the effects of disjoining pressures. The crossover from one regime to the other is characterized by an evolution of the flow velocity profile from plug-like to one where the shear layer is much broader. The mean rate of neighbour changes per bubble increases as flow moves towards the steady regime with a distribution that broadens with the strength of the driving. We show that the stick-slip and steady flow regimes can be distinguished by the spectrum of energy fluctuations during the flow. We also vary the strength of the attractive potential and highlight the effect this has on the different regimes.

2:21PM R35.00008 Using DNS and Statistical Learning to Model Bubbly Channel Flow1, MING MA, JIACAI LU, GRETAR TRYGGVASON, University of Notre Dame — The transient evolution of laminar bubbly flow in a vertical channel is examined by direct numerical simulation (DNS). Near spherical bubbles, initially distributed evenly in a fully developed parabolic flow, are driven relatively quickly to the walls, where they increase the drag and reduce the flow rate on a longer time scale. Once the flow rate has been decreased significantly, some of the bubbles move back into the channel interior and the void fraction there approaches the value needed to balance the weight of the mixture and the imposed pressure gradient. A database generated by averaging the DNS results is used to model the closure terms in a simple model of the average flow. Those terms relate the average lateral flux of the bubbles, the velocity fluctuations and the averaged surface tension force to the fluid shear, the void fraction and its gradient, as well as the distance to the nearest wall. An aggregated neural network is used for the statistically leaning of unknown closures, and closure relationships are tested by following the evolution of bubbly channel flow with different initial conditions. It is found that the model predictions are in reasonably good agreement with DNS results.

1Supported by NSF
2:34PM R35.00009 Particle scavenging in a cylindrical ultrasonic standing wave field using levitated drops. TYLER MERRELL, J.R. SAYLOR, Clemson University — A cylindrical ultrasonic standing wave field was generated in a tube containing a flow of particles and fog. Both the particles and fog drops were concentrated in the nodes of the standing wave field where they combined and then grew large enough to fall out of the system. In this way particles were scavenged from the system, cleaning the air. While this approach has been attempted using a standing wave field established between disc-shaped transducers, a cylindrical resonator has not been used for this purpose heretofore. The resonator was constructed by bolting three Langevin transducers to an aluminum tube. The benefit of the cylindrical geometry is that the acoustic energy is focused. Furthermore, the residence time of the particle in the field can be increased by increasing the length of the resonator. An additional benefit of this approach is that tubes located downstream of the resonator were acoustically excited, acting as passive resonators that enhanced the scavenging process. The performance of this system on scavenging particles is presented as a function of particle diameter and volumetric flow rate. It is noted that, when operated without particles, the setup can be used to remove drops and shows promise for liquid aerosol retention from systems where these losses can be financially disadvantageous and/or hazardous.

2:47PM R35.00010 The catastrophic failures of plants hydraulic network examined through a model system. DIANE BIENAIMÉ, PHILIPPE MARMOTTANT, CNRS and University Grenoble Alpes, TIM BRODRIBB, University of Tasmania — Plants live a dangerous game: they have to facilitate water transport in their xylem conduits while minimizing the consequence of hydraulic failure. Indeed, as water flows under negative pressure inside these conduits, cavitation bubbles spontaneously occur. The failure dynamics of this hydraulic network is poorly studied, while it has important ecological and bioengineering implications. Here, by using dark-field transmission microscopy, we were able to directly visualize the spreading of cavitation bubbles within leaves, where the xylem conduits form a 2D and transparent network. We observe the surprising fact that the probability of cavitation increases in larger veins, where the majority of water flows. Next, in order to understand the physical mechanism of nucleation and propagation, we built artificial networks of channels made in hydrogel, where evaporation generates negative pressures. We find the hydraulic failure follows two stages: first a sudden bubble nucleation relaxing to the elastic stored of the system, and then a slow expansion driven by the flow of water in the surrounding medium. Channel constrictions slow the propagation of the bubble, similarly to the small valves that connect plants conduits.

Tuesday, November 24, 2015 12:50PM - 3:26PM — Session R36 Bubbles: Dynamics Ballroom C - Sadegh Dabiri, Purdue University

12:50PM R36.00001 Rising motion of a bubble layer near a vertical wall. SADEGH DABIRI, PRAMOD BHUVANKAR, Purdue University — Bubbly flows in vertical pipes and channels form a wall-peak distribution of bubbles under certain conditions. The dynamics of the bubbles near the wall is different than in an unbounded liquid. Here we report the rising motion of bubbles at a liquid/vapor interface in a vertical wall with a bubbly flow in a periodic domain on one side, an average pressure gradient is applied to the domain that balances the weight of the liquid phase. The upward flow is created by the rising motion of the bubbles. The bubbles are kept near the wall by the lateral lift force acting on them as a result of rising in a shear flow which is in turn generated by rising motion of bubbles. The rise velocity of the bubbles on the wall and the average rise velocity of the liquid depend on three dimensionless parameters, Archimedes number, Eotvos number, and the average volume fraction of bubbles near the wall. In the limit of small Eo, bubbles are nearly spherical and the dependency on Eo becomes negligible. In this limit, the scaling of the liquid Reynolds number with Archimedes number and the void fraction is presented.

1:03PM R36.00002 Bubble dynamics in a variable gap Hele-Shaw cell. SAUL PIEDRA, ROBERTO DOMINGUEZ, EDUARDO RAMOS, Universidad Nacional Autonoma de Mexico — We present observations of the dynamics of individual air bubbles ascending in a Hele-Shaw cell filled with water. Cells with gaps of 1 mm, 1.5 and 2.5 mm are used and the volume of the bubbles is such that we observe bubbles with apparent diameter from 2 mm to 7.3 mm. Given that we work with air and water in all experiments, the Morton number is constant and equal to 9.3 × 10^-11. The results are given in terms of the Eotvos, Archimedes and Reynolds numbers, and the trajectories and wakes of the bubbles are described as functions of the gap. In all cases we observe a linear relationship between the Reynolds and Archimedes numbers, but the proportionality constant varies with the gap. Also, although the wake is composed of alternating vortices similar to the von Karman vortex street, the size and location of the vortices vary with the gap. The analysis of some features of the observations and the description of the shape of the bubbles and dominant forces are made with a two dimensional numerical solution of the conservation equations using a front tracking strategy.

1:16PM R36.00003 Simulations of Non-spherical Bubble Collapse Dynamics in Visco- and Viscoelastic Media Near a Compliant Object. MAURO RODRIGUEZ, ERIC JOHNSEN, University of Michigan, Ann Arbor — Understanding the dynamics of cavitation bubbles and the shock waves emitted by their collapse in and near viscoelastic media is important for various naval and medical applications, particularly in the context of cavitation damage. Two examples are histotripsy, which utilizes this phenomenon for the ablation of pathogenic tissue, and erosion to elastomeric coatings on propellers. Although not fully understood, the damage mechanism combines the effect of the incoming pulses and cavitation produced by the high tension. Additionally, the influence of the shock on the material and the response of the material to the shock are not well known. A novel numerical approach for simulating shock and acoustic wave propagation in Zener-like viscoelastic media is proposed. This Eulerian method is based on a high-order accurate weighted essentially non-oscillatory scheme for shock capturing and introduces evolution equations for the components of the shear stress tensor. Validation studies between high-fidelity two-dimensional simulations of the bubble collapse dynamics for various experimental configurations (i.e. the viscous or viscoelastic material surrounding the bubble and the nearby compliant object are varied) will be presented.

1:29PM R36.00004 Bubbles in a superhydrophobic tube. HELENE DE MALEPRADE, CHRISTOPHE CLANET, LadHyX - Ecole polytechnique, DAVID QUERE, PMMH - ESPCI — When a capillary tube is put in contact to a bath of wetting liquid, liquid spontaneously rises with a dynamics that has been widely studied. However, if the tube is superhydrophobic, water will not tend to enter nor rise. Rather, the tube immersed in water and brought in contact with air will be invaded by air. The dynamics of this capillary dewetting process differs from the usual capillary rise, which we discuss together with the value of dynamic contact angles at the liquid/air interface.

1This work is supported by ONR grant N00014-12-1-0751.
Modelling of bubble trajectories in a pump impeller. A vertical rotating flow in an annulus gap with an increasing diameter is used to approximate the flow in a pump impeller. We study a spherical gas bubble released at the flow inlet, subject to turbulent drag and added mass forces. Bubbles trajectory have been computed for different geometries, rotation speeds and bubble size, showing a deviation from the liquid streamlines in the angular and radial directions. This effect is related to the pump performance in multiphase conditions: the velocity difference between the gas and the liquid phases changes the final pressure rise produced by the impeller. In some extreme cases, the centrifugal force can be large enough to prevent bubbles from exiting the impeller at all, leading to an unwanted gas accumulation and the blockage of the pump. We eventually quantify the effects of geometrical and operational parameters on the pump behaviour.

Bernoulli Suction Effect on Soap Bubble Blowing. As a model system for thin-film bubble with two gas-liquid interfaces, we experimentally investigated the pinch-off of soap bubble. Using the lab-built bubble blower and high-speed videography, we have found that the scaling law exponent of soap bubble pinch-off is $2/3$, which is similar to that of soap film bridge. Because air flowed through the decreasing neck of soap film tube, we studied possible Bernoulli suction effect on soap bubble pinch-off by evaluating the Reynolds number of airflow. Image processing was utilized to calculate approximate volume of growing soap film tube and the volume flow rate of the airflow, and the Reynolds number was estimated to be $800-3200$. This result suggests that soap bubbling may involve the Bernoulli suction effect.

The formation of soap bubbles created by blowing on soap films. Time-lapse photography was used to observe the bubble formation. We observe that the bubble size and shape are sensitive to the distance between the film and nozzle, and gas density, and we measure the gas velocity threshold above which bubbles are generated. The response is sensitive to confinement, that is, the ratio between film and jet sizes, and dissipation in the turbulent gas jet which is a function of the distance from the nozzle to the film. We observe four different regimes that we rationalize by comparing the dynamic pressure of a jet on the film and the Laplace pressure needed to create the curved surface of a bubble.

Bubble oscillation regimes including phase change. We study bubble formation and oscillation regimes in an expanding soap film tube by tracking the bubble interface. We observe that the oscillation frequency is influenced by the enthalpy of vaporization and the ratio of thermal conductivities. The proposed analysis is meant to be useful for the validation of full 3D numerical codes dealing with phase change processes.

Three-dimensional numerical simulations of a bubble rising in an unbounded weakly viscous fluid. A freely rising bubble in an unbounded low-viscosity fluid are performed to analyze the bubble trajectory for values of Galileo and Bond numbers close to the transition between vertical and non-vertical paths. The simulations are performed with the Gerris Flow Solver, based on the Volume of Fluid technique to track the interface, allowing deformations of the bubble during its rising motion. We find the existence of novel regimes of the bubble rise which we describe by tracking the bubble shape, path geometry and wake vortical structures, as well as the temporal evolution of the instantaneous Reynolds number. Besides the traditional rectilinear, zigzag and spiral paths, we observe chaotic, reflectional-symmetry-breaking or reflectional-symmetry-preserving regimes previously reported for axisymmetric solid bodies. The DNS results also allow us to check the accuracy of the neutral curve defining the region of the parameter space within which the vertical path of a buoyancy-driven bubble with fore-and-aft asymmetric shape is linearly stable.
3:00PM R36.00011 Rise of an argon bubble in liquid steel in the presence of a transverse magnetic field. — SURYA PRATAP VANKA, KAI JIN, PURUSHOTAM KUMAR, BRIAN THOMAS, University of Illinois at Urbana-Champaign — In this work, the motion of a single argon gas bubble rising in quiescent liquid steel under an external magnetic field is studied numerically using a Volume-Of-Fluid (VOF) method. To mitigate spurious velocities normally generated during numerical simulation of multiphase flows with large density differences, an improved algorithm for surface tension modeling, originally proposed by Wang and Tong (International Journal of Thermal Sciences 47, 221–228 (2008)) is implemented, validated and used in present computations. The governing equations are integrated by a second-order space and time accurate numerical scheme, and implemented on multiple Graphics Processing Units (GPU) with high parallel efficiency. The motion and the terminal velocities of the rising bubble under different magnetic fields are compared and a reduction in rise velocity is seen in cases with the magnetic field applied. The shape deformation and the path of the bubble are discussed. An elongation of the bubble along the second-order field direction is seen, and the physics behind these phenomena is discussed. The circulation inside of the bubble is seen to be affected by the magnetic field indirectly. The wake structures behind the bubble are visualized and effects of the magnetic field on the wake structures are presented.

3:13PM R36.00012 Dynamics of rising bubble inside a viscosity-stratified medium. — MANOJ TRIPATHI, PREMLATA A. R., KIRTI SAHU, Indian Institute of Technology Hyderabad, India — The rising bubble dynamics in an unconfined quiescent viscosity-stratified medium has been numerically investigated. This is frequently encountered in industrial as well as natural phenomena. In spite of the large number of studies carried out on bubbles and drops, very few studies have examined the influence of viscosity stratification on bubble rise dynamics. To the best of our knowledge, none of them have isolated the effects of viscosity-stratification alone, even though it is known to influence the dynamics extensively, which is the main objective of the present study. By conducting time-dependent simulations, we present a library of bubble shapes in the Galilei and the Eötvös numbers plane. Our results demonstrate some counter-intuitive phenomena for certain range of parameters due to the presence of viscosity stratification in the surrounding fluid. We found that in a linearly increasing viscosity medium, for certain values of parameters, bubble undergoes large deformation by forming an elongated skirt, while the skirt tends to physically separate the wake region from the rest of the surrounding fluid. This peculiar dynamics is attributed to the migration of less viscous fluid that is carried in the wake of the bubble as it rises, and thereby creating an increase in the wake viscosity.

Tuesday, November 24, 2015 12:50PM - 3:26PM

Session R37 Biofluids: Swimming Animals
Sheraton Back Bay A
Christophe Clanet, cole Polyclinique

12:50PM R37.00001 Diving birds. — CHRISTOPHE CLANET, LUCIEN MASSON, Ecole polytechnique, GARETH MCKINLEY, MIT-Mechanical Engineering, ROBERT COHEN, MIT-Chemical Engineering, ECOLE POLYTECHNIQUE COLLABORATION, MIT COLLABORATION — Many seabirds (gannets, pelicans, gulls, albatrosses) dive into water at high speeds (25 m/s) in order to capture underwater prey. Diving depths of 20 body lengths are reported in the literature. This value is much larger than the one achieved by men, which is of the order of 5. We study this difference by comparing the impact of slender vs bluff bodies. We show that, contrary to bluff bodies, the penetration depth of slender bodies presents a maximum value for a specific impact velocity that we connect to the velocity of diving birds.

1:03PM R37.00002 Calanoid Copepod Behavior in Thin Layer Shear Flows: Freshwater Versus Marine. — A. N. SKIPPER, D. R. WEBSTER, J. YEN, Georgia Tech — Marine copepods have been shown to behaviorally respond to vertical gradients of horizontal velocity and aggregate around thin layers. The current study addresses whether a freshwater copepod from an alpine lake demonstrates similar behavior response. *Hesperodiaptomus shoshone* is often the greatest biomass in alpine lakes and is the dominant zooplankton predator within its environment. The hypothesis is that *H. shoshone* responds to vertical gradients of horizontal velocity, which are associated with wind-driven outflow, in a similar manner as its marine counterpart. The two calanoid copepods studied here, *H. shoshone* (freshwater) and *Calanus finmarchicus* (marine), are of similar size (2 – 4 mm), have similar morphologies, and utilize cruising as their primary swimming mode. The two animals differ not only in environment, but also in diet; *H. shoshone* is carnivore, whereas *C. finmarchicus* is an herbivore. A laminar, planar jet (Bickley) was used in the laboratory to simulate a free shear flow. Particle image velocimetry (PIV) quantified the flow field. The marine species changed its swimming behavior significantly (increased swimming speed and turning frequency, and spent more time in the thin flow layer) from control to treatment. In contrast, the freshwater species exhibited very few changes in either swimming behavior or residence time. Swimming kinematics and residence time results were also similar between males and females. Unlike the marine copepod, the results suggest the environmental flow structure is unimportant to the freshwater species.

1:16PM R37.00003 Larvacean kinematics: a biological model of flapping flexible foils. — ALEXA BAUMER, The George Washington University, KAKANI KATIJA, Monterey Bay Aquarium Research Institute, MEGAN C. LEFTWICH, The George Washington University — Larvaceans are marine organisms found throughout the world’s oceans that create mucus houses to filter food (e.g., small plankton, detritus, and particulates) from adjacent waters. Estimates indicate that discarded mucus houses, which are eventually abandoned by larvaceans, are responsible for one-third of the particulate transported to the bottom of Monterey Bay in Central California. Once houses are abandoned, larvaceans swim freely to another location before generating a new one. Here we conduct a study of an undescribed larvacean, Bathocordaeus sp., to examine their free-swimming and in-house behaviors, and how changes in their body kinematics may alter fluid interactions. High definition videos captured by remotely operated vehicles (ROVs) in Monterey Bay (from 2003 to present) are analyzed to extract the kinematics of larvacean tail movement during these two distinct swimming behaviors. Using in-house Matlab algorithms, we reveal significant differences in stroke dynamics as traveling waves propagate along the larvacean tail. These kinematic differences may have important implications for swimming performance and fluid filtration rates through larvacean mucus houses.

1:29PM R37.00004 ABSTRACT WITHDRAWN

1:42PM R37.00005 Mechanical and scaling considerations for efficient jellyfish swimming. — ALEXANDER HOOVER, Tulane University, LAURA MILLER, BOYCE GRIFFITH, University of North Carolina at Chapel Hill — With a fossil record dating over half a billion years, jellyfish represent one of the earliest examples of how multicellular organisms first organized into moving systems. Lacking an agonist-antagonist muscle pairing, jellyfish swim via a process of elastic deformation and recoil. Jellyfish propulsion is generated via the coordinated contraction of its elastic bell by its contractile muscles and a companion re-expansion that is passively driven by stored elastic energy. Recent studies have found jellyfish to be one of the most efficient swimmers due to its low energy expenditure in their forward movement. Using an immersed boundary framework, we will further examine the efficiency of jellyfish swimming by incorporating material models that are informed by the musculature present in jellyfish into a model of the elastic jellyfish bell in three dimensions. The fully-coupled fluid structure interaction problem is solved using an adaptive and parallelized version of the immersed boundary method (IBAMR). This model is then used to explore how variability in the mechanical properties of the bell affect the work done by the bell as well as the cost of transport related to jellyfish locomotion. We then examine how the efficiency of this system is affected by the Reynolds number.
1:55PM R37.00006 Flapping, wobbling, and zig-zagging: Tomographic PIV measurements of Antarctic sea butterfly “flying” underwater. **D. ADHIKARI, D.R. WEBSTER, J. YEN,** Georgia Tech — A portable tomographic PIV technique was used to study the fluid dynamics and kinematics of sea butterflies in Antarctica. Antarctic pteropods (or sea butterflies), which are currently threatened by ocean acidification, swim in seawater with a pair of gelatinous parapodia (or “wings”) via a unique propulsion mechanism. Both power and recovery strokes propel the organism (1.5 – 5 mm in size) upward in a sawtooth-like trajectory with average speed of 14 – 30 mm/s and pitch the shell forwards-and-backwards at 1.9 – 3 Hz. The pitching motion effectively positions the parapodia such that they stroke downward during both the power and recovery strokes. Reynolds numbers defined for flapping, translating, and pitching (i.e. \(Re_f, Re_t, \) and \(Re_p\)) characterize the motion of the pteropod. For \(Re_f \leq 50\), the shell does not pitch and the pteropod swims abnormally. We present a detailed comparison of the volumetric fluid velocity fields induced by pteropods swimming upwards with \(Re_f = 80\) and 180. The pteropod at the lower \(Re_f\) creates an attached shear flow along the parapodia and pushes fluid in a method analogous to a paddle. In contrast, at higher \(Re_f\), the flow along the parapodia separates and generates complex vortex structures.

2:08PM R37.00007 Experimental investigation of crustacean swimming with variation of limb structures. **HONG KUAN LAI, MILAD SAMAEE, GEOFFREY DONNELL, ARVIND SANTHANAKRISHNAN,** Oklahoma State University, **ROBERT GUY, TIMOTHY LEWIS,** University of California, Davis — Crustaceans such as crayfish and krill swim by rhythmically paddling a set of four to five limbs (known as swimmerets or pleopods) originating from their abdomen. The limb motion in these animals has been observed to follow tail-to-head metachronal wave pattern with an approximate quarter-period inter-limb phase difference. The goal of this study is to investigate the hydrodynamics of this swimming mechanism as a function of inter-limb phase difference, inclusion of hinges in the limbs, and Reynolds number (\(Re\)). 2D PIV measurements were conducted on a scaled robotic model of metachronal paddling, consisting of a rectangular tank fitted with stepper motors coupled to a four-bar linkage that actuated four paddles immersed in water-glycerin fluid medium. The inter-limb phase difference was varied from 0% (synchronous paddling) through 50% across \(Re\) range of \(10-1000\). Two types of limb models were used, including a simple flat plate and a ‘split-paddle’ structure with two flat plates connected halfway with hinges. The results of the study show that limb models with hinges generated increased horizontal (thrust-producing direction) fluid velocity compared to the simple flat plate paddles, suggesting that asymmetry between power and return strokes is important to augment thrust.

2:21PM R37.00008 Bio-inspired robotic legs drive viscous recirculating flows. **DAISUKE TAKAGI, RINTARO HAYASHI,** University of Hawaii at Manoa — Crustaceans actuate multiple legs in a well-coordinated sequence to generate fluid flow for swimming and feeding. Inspired by tiny crustacean larvae operating at low \(Re\) number, we study a scaled-up model in which slender rods oscillate independently in a bath of glycerol. Experiments reveal qualitatively different flow patterns depending on the phase and orientation of actuated rods. The observations are analyzed in the framework of slender-body theory for Stokes flow. This study shows that simple oscillatory motion of multiple legs can produce complex recirculating flows, with potential applications for mixing and pumping.

2:34PM R37.00009 Volumetric flow around a swimming lamprey. **ANDREA M. LEHN,** MIT, SEAN P. COLIN, Roger Williams University, **JOHN H. COSTELLO,** Providence College. **MEGAN C. LEFTWICH,** GWU, **ERIC D. TYTELL,** Tufts University — A primary experimental technique for studying fluid-structure interactions around swimming fish has been planar dimensional particle image velocimetry (PIV). Typically, two components of the velocity vector are measured in a plane, in the case of swimming studies, directly behind the animal. While useful, this approach provides little to no insight about fluid structure interactions above and below the fish. For fish with a small height relative to body length, such as the long and approximately cylindrical lamprey, 3D information is essential to characterize how these fish interact with their fluid environment. This study presents 3D flow structures along the body and in the wake of larval lamprey, Petromyzon marinus, which are 10-15 cm long. Lamprey swim through a 1,000 cm \(^3\) field of view in a standard 10 gallon tank illuminated by a green laser. Data are collected using the three component velocimeter V3S system by TSI, Inc. and processed using Insight 4G software. This study expands on previous works that show two pairs of vortices each tail beat in the mid-plane of the lamprey wake.

2:47PM R37.00010 Investigation of Thunniform Swimming Using Material Testing, Biomimetic Robotics and Particle Image Velocimetry. **RUJIE ZHU, VISHAAL SARAIYA, JIANZHONG ZHU,** HILARY BART-SMITH, University of Virginia — Thunniform swimming is well recognized as an efficient method for high-speed long-distance underwater travelers such as tuna. Previous research has shown that tuna relies on contraction and relaxation of red muscle to generate angular motion of its large, crescent-shaped caudal fin and peduncle. This research project is composed of two parts, first of which is determining mechanical properties of components such as spine joints, tendons, fin rays and cartilage, from which the biomechanics of tuna tail can be better understood. The second part is building a robotic system mimicking a real tuna tail based on previously retrieved information, and testing the system inside a flow tank. With the help of PIV (Particle Image Velocimetry), fluid-structure interaction of the biomimetic fin is visualized and data such as swimming speed and power consumption are retrieved through the robotic system. The final outcome should explain how the material properties of tuna tail affect fluid dynamics of thunniform swimming.

This project is supported by Office of Naval Research (ONRBA13-022).

3:00PM R37.00011 Copepod Behavioral Response to Simulated Frontal Flows. **D.R. WEBSTER, A.C. TRUE,** M.J. WEISSBURG, J. YEN, Georgia Tech. **A. GENIN,** The Hebrew University of Jerusalem and The Interuniversity Institute for Marine Sciences of Eilat — When presented with a fine-scale upwelling or downwelling shear flow in a laboratory flume, two tropical copepods from the Red Sea, Acartia neglecta and Clausocalanus furcatus, performed a set of behaviors that resulted in apparent depth-keeping and the potential for producing patchiness. Analyses of free-swimming trajectories revealed a behavioral threshold shear deformation rate value of 0.05 s\(^{-1}\) for both species. This threshold triggered statistically significant changes in path kinematics (i.e., relative swimming speed and turn frequency) in the shear layer versus out-of-layer. Gross path characteristics (i.e., net-to-gross displacement ratio, NGDR, and proportional victimity time, PVT) were also significantly different in the shear layer treatments compared to controls. The vertical net-to-gross displacement ratio (VNGDR) was introduced here to explain a spectrum of depth-keeping behaviors. The mean value of VNGDR significantly increased in the shear layer treatments and, coupled with excited relative swimming speeds, suggested the potential to induce large vertical transport (at the 10 cm scale of the observation). However, histograms of VNGDR revealed a bimodality, which indicated a sizable portion of the population was also displaying depth-keeping behavior. Those copepod trajectories displaying large VNGDR predominately consisted of copepods swimming against the flow, thereby resisting vertical advection, which is another potential depth-keeping mechanism.
3:13PM R37.00012 Hydrodynamic role of longitudinal ridges in a leatherback turtle swimming\textsuperscript{1}, KYEONGTAE BANG, JOOHA KIM, SANG-IM LEE, HAEcheon CHO, Seoul National University — The leatherback sea turtle (Dermochelys coriacea), the fastest swimmer and the deepest diver among marine turtles, has five longitudinal ridges on its carapace. These ridges are the most remarkable morphological features distinguished from other marine turtles. To investigate the hydrodynamic role of these ridges in the leatherback turtle swimming, we model a carapace with and without ridges by using three dimensional surface data of a stuffed leatherback turtle in the National Science Museum, Korea. The experiment is conducted in a wind tunnel in the ranges of the real leatherback turtles Reynolds number (Re) and angle of attack (\(\alpha\)). The longitudinal ridges function differently according to the flow condition (i.e. Re and \(\alpha\)). At low Re and negative \(\alpha\) that represent the swimming condition of hatchlings and juveniles, the ridges significantly decrease the drag by generating streamwise vortices and delaying the main separation. On the other hand, at high Re and positive \(\alpha\) that represent the swimming condition of adults, the ridges suppress the laminar separation bubble near the front part by generating streamwise vortices and enhance the lift and lift-to-drag ratio.

\textsuperscript{1}Supported by the NRF program (2011-0028032)

Tuesday, November 24, 2015 12:50PM - 3:00PM — Session R38 Biofluids: Flexible Swimmers III — Sheraton Back Bay B - George Lauder, Harvard University

12:50PM R38.00001 Application of PIV-based pressure measurements to the study of aquatic propulsion, KELSEy LUCAS, Harvard Univ. JOHN DABIRI, Stanford Univ, GEORGE LAuder, Harvard Univ — Although it is relatively straightforward to image how fluid moves around a swimmer, translation of these motions to mechanisms that generate forces for propulsion is more difficult. This process is greatly facilitated by a recently developed technique for non-invasive pressure measurements that generate 2D pressure fields. Here, we explore how accurate a purely pressure-based calculation of propulsive forces can be. By comparing these calculations to forces and torques measured directly using a sensor on a robotic flapping foil system, we characterize the effects of motion frequency and out-of-plane flows on the calculation's accuracy. We then apply this calculation to study the dynamics of fish-like swimming of a foil model with non-uniform flexural stiffness, and to those of a freely swimming fish.

1:03PM R38.00002 Non-invasive 3D geometry extraction of a Sea lion foreflipper, CHEN FRIEDMAN, MARTHA WATSON, George Washington University, PAMELA ZHANG, Holton Arms school, MEGAN LEFTWICH, George Washington University — We are interested in underwater propulsion that leaves little traceable wake structure while producing high levels of thrust. A potential biological model is the California sea lion, a highly maneuverable aquatic mammal that produces thrust primarily with its foreflippers without a characteristic flapping frequency. The foreflippers are used for thrust, stability, and control during swimming motions. Recently, the flipper’s kinematics during the thrust phase was extracted using 2D video tracking. This work extends the tracking ability to 3D using a non-invasive Direct Linear Transformation technique employed on non-research sea lions. marker-less flipper tracking is carried out manually for complete dorsal-ventral flipper motions. Two cameras are used (3840 \( \times \) 2160 pixels resolution), calibrated in space using a calibration target inserted into the sea lion habitat, and synchronized in time using a simple light flash. The repeatability and objectivity of the tracked data is assessed by having two people tracking the same clap and comparing the results. The number of points required to track a flipper with sufficient detail is also discussed. Changes in the flipper pitch angle during the clap, an important feature for fluid dynamics modeling, will also be presented.

1:16PM R38.00003 Resistive and reactive force production in actuated elastic swimmers\textsuperscript{1}, RAMIRO GODOY-DIANA, MIGUEL PINEIRUA, BENJAMIN THIRIA, PMMH UMR7636 ESPCI ParisTech CNRS UPMC U Paris Diderot — We study the force production dynamics of undulating elastic plates as a model for fish-like inertial swimmers. Using a beam model coupled with Lighthill’s large-amplitude elongated-body theory, we explore different localized actuations at one extremity of the plate (heaving, pitching, and a combination of both) in order to quantify the reactive and resistive contributions to the thrust. The latter has only recently been pointed out as a crucial element in the force balance of large Reynolds number swimmers [Pineirua et al. Phys. Rev. E (2015)]. We show that this balance is modified as the frequency of excitation changes and the response of the elastic plate shifts between different resonant modes. In the heaving case for instance, higher frequencies and thus higher modes are associated to a stronger resistive contribution to the thrust, while in pitching case, at all frequencies, thrust production comes mostly from the reactive term.

\textsuperscript{1}We acknowledge support from EADS foundation through project “Fluids and elasticity in biomimetic propulsion”

1:29PM R38.00004 Optimality Principles of Undulatory Swimming\textsuperscript{1}, NISHANT NANGIA, Department of Engineering Sciences and Applied Mathematics, Northwestern University, RAHUL BALE, NEELESH PATANKAR, Department of Mechanical Engineering, Northwestern University — A number of dimensionless quantities derived from a fish’s kinematic and morphological parameters have been used to describe the hydrodynamics of swimming. In particular, body/caudal fin swimmers have been found to swim within a relatively narrow range of these quantities in nature, e.g., Strouhal number or the optimal specific wavelength. It has been hypothesized or shown that these constraints arise due to maximization of swimming speed, efficiency, or cost of transport in certain domains of this large dimensionless parameter space. Using fully resolved simulations of undulatory patterns, we investigate the existence of various optimality principles in fish swimming. Using scaling arguments, we relate various dimensionless parameters to each other. Based on these findings, we make design recommendations on how kinematic parameters for a swimming robot or vehicle should be chosen.

\textsuperscript{1}This work is supported by NSF Grants CBET–0828749, CMMI–0941674, CBET–1066575 and the National Science Foundation Graduate Research Fellowship under Grant No. DGE–1324589.
1:42PM R38.00005 The Hydrodynamics of Plesiosaurs
LUKE MUSCUTT, BHARATHRAM GANAPATHISUBRAMANI, GARETH DYKE, GABRIEL WEYMOUTH, University of Southampton — Plesiosaurs are extinct marine reptiles that existed at the same time as the dinosaurs, and are the only known animals to swim by actively flapping their four wing-like flippers. This can be viewed as a tandem flapping wing problem, where the hind wing is operating in the wake of the fore wing. Experiments using full-scale robotic plesiosaur flippers in a large flume tank have been used to investigate the kinematics and interaction of the flippers. The flippers are actuated in heave and pitch, and a combination of force measurements and flow visualization are used to analyze the characteristics of the vortex interaction between the flippers. Previous two-dimensional numerical simulations have shown that certain kinematics give an increase in thrust of the hind flipper of up to 50%. The current experiments determine if such thrust augmentation is present for a three-dimensional flowfield and determine the kinematics that give the highest possible thrust. This will help to answer paleo-biological questions about the function and evolution of the plesiosaur flippers, along with helping to determine if tandem flapping wings could be a viable propulsion system for autonomous underwater vehicles.

1:55PM R38.00006 Fluid Dynamics of Clap-and-Fling with Highly Flexible Wings inspired by the Locomotion of Sea Butterflies
YUANDA LI, Massachusetts Institute of Technology, PETER YEH, ALEXANDER ALEXEEV, Georgia Institute of Technology — We use three dimensional computer simulations to probe the hydrodynamics of oscillating flexible fins with varying thickness. The fin is modeled as an elastic two dimensional plate and introduce the effect of the thickness gradient by including an appropriate mass gradient and stiffness gradient along the plate. The flexible fin is actuated by a plunging motion at its leading edge. We evaluate the performance of the swimmer with a uniformly thick swimmer. These findings may shed insight into some of the physical mechanisms that allow fish to have high swimming efficiency. Ref1: supported by a grant from NSF.

2:08PM R38.00007 The role of spanwise-flexible propulsors in swimming and flying
YUANDA LI, MAI LAM, University of Minnesota — Natural swimmers and flyers span several orders of magnitude in both Reynolds number and fluid-propulsor mass ratio. Intuitively, one would expect different aerelastic strategies to be employed across these regimes. However, similar magnitudes of spanwise bending, as measured by flexion angle, have been observed across this entire range for cruise conditions. In this study, we hypothesize that propulsor flexion has converged to generate similar spanwise vorticity transport in order to control the dynamic-stall vortex ubiquitously in natural swimming and flight. In particular, it is believed that vorticity convection and vortex stretching delay vortex detachment by balancing vorticity generated at the leading-edge and by reducing the overall vortex size, respectively, as recently shown for flapping profiles. Moving forward, passive spanwise flexibility of propulsors is now abstracted as a spanwise-variation in effective incidence. This abstraction is realized as a pitching-flapping motion. By comparing passively flexible cases to rigid cases, the role of flexibility in controlling vorticity transport, and thus delaying vortex detachment is elucidated. Ref1: Lucas, K. N. et al., Nat. Commun. 5, 3293 (2014) Ref2: Wong, J. G., Rival, D. E., J. Fluid Mech. 766, 611-625 (2015)

2:21PM R38.00008 Strouhal number for free swimming
MEHDI SAADAT, University of Virginia, TYLER VAN BUREN, DANIEL FLORYAN, Princeton University, ALEXANDER SMITS, Princeton University, Monash University, HOSSEIN HAJ-HARIRI, University of Virginia — In this work, we present experimental results to explore the implications of free swimming for Strouhal number (as an outcome) in the context of a simple model for a fish that consists of a 2D virtual body (source of drag) and a 2D pitching foil (source of thrust) representing cruising with thunniform locomotion. The results validate the findings of Saadat and Haj-Hariri (2012): for pitching foils, the Strouhal number is a function of Strouhal number for all gaits having amplitude less than a certain critical value. Equivalently, given the balance of thrust and drag forces at cruise, Strouhal number is only a function of the shape, i.e. drag coefficient and area, and essentially constant for high enough swimming speeds for which the mild dependence of drag coefficient on the speed vanishes. Furthermore, a dimensional analysis generalizes the findings. A scaling analysis shows that the variation of Strouhal number with cruising speed is functionally related to the variation of body drag coefficient with speed. Ref: Saadat, M. and Haj-Hariri, H. Role of Strouhal number (St) in free swimming. American Physical Society, 65th Annual DFD Meeting, San Diego, CA, Nov 1820, 2012.

1Supported by ONR MURI Grant N00014-14-1-0533.

2:34PM R38.00009 A mechanism for efficient swimming
MEHDI SAADAT, AARON BRANDES, VISHAAL SARAIYA, HILARY BART-SMITH, University of Virginia — We present experimental measurements of hydrodynamic performance as well as wake visualization for a freely swimming 3D foil with pure pitching motion. The foil is constrained to move in its axial direction. It is shown that the iso-lines for speed and input power (or economy) coincide in the dimensionless frequency versus amplitude plane, up to a critical amplitude. The critical amplitude is independent from swimming speed. It is shown that all swimming gaits (combination of frequency and amplitude) share a single value for Strouhal number for all gait having amplitude less than a certain critical value. Equivalently, given the balance of thrust and drag forces at cruise, Strouhal number is only a function of the shape, i.e. drag coefficient and area, and essentially constant for high enough swimming speeds for which the mild dependence of drag coefficient on the speed vanishes. Furthermore, a dimensional analysis generalizes the findings. A scaling analysis shows that the variation of Strouhal number with cruising speed is functionally related to the variation of body drag coefficient with speed. Ref1: Saadat, H., Haj-Hariri, H. Role of Strouhal number (St) in free swimming. American Physical Society, 65th Annual DFD Meeting, San Diego, CA, Nov 1820, 2012.

2:47PM R38.00010 Thickness-varying flexible plunging fins swim more efficiently
YUANDA LI, Massachusetts Institute of Technology, PETER YEH, ALEXANDER ALEXEEV, Georgia Institute of Technology — We use three dimensional computer simulations to probe the hydrodynamics of oscillating flexible fins with varying thickness. The fin is modeled as an elastic rectangular plate with the thickest section at the leading edge, decreasing linearly until the trailing edge. The plate is modeled as infinitely thin, and we assume that the thickest part of the fin is much smaller compared to its other length scales. Therefore, we simulate the swimmer as two dimensional plate and introduce the effect of the thickness gradient by including an appropriate mass gradient and stiffness gradient along the length of the plate. The flexible fin is actuated by a plunging motion at its leading edge. We evaluate the performance of the swimmer by measuring the steady state thrust, free swimming velocity, input power, and swimming economy as a function of driving frequency and the magnitude of the thickness gradient. We find a wideband frequency range in which the swimming economy is increased as compared to a uniformly thick swimmer. These findings may shed insight into some of the physical mechanisms that allow fish to have high swimming efficiency.
Tuesday, November 24, 2015 12:50PM - 3:26PM
Session R39 Biofluids: Swimmer-Surface Interactions

12:50PM R39.00001 Microscopic suspension feeders near boundaries: Effects of external water flow, RACHEL PEPPER, University of Puget Sound, M.A.R. KOEHL, University of California Berkeley — Microscopic sessile suspension feeders are an important part of aquatic ecosystems and form a vital link in the transfer of carbon in aquatic food webs. These suspension feeders live attached to boundaries, consume bacteria and small detritus, and are in turn eaten by larger organisms. Many create a feeding current that draws fluid towards them, and from which they filter their food. In still water, the feeding current consists of recirculating eddies which form as a result of fluid forcing near a boundary. These recirculating eddies can be depleted of food and significantly decrease nutrient uptake; a variety of strategies have been proposed for how attached feeders increase their access to undepleted water. We investigate the interaction of the flow produced by a microscopic suspension feeder with external environmental flow, such as the current in a stream or ocean. We show through calculations that even very slow flow (on the order of microns per second) is sufficient to provide a constant supply of undepleted water to suspension feeders when the feeders are modeled with perfect nutrient capture efficiency and in the absence of diffusion. We also discuss which natural flow environments exceed the threshold to supply undepleted water and which do not, and we examine how characteristics of the suspension feeders themselves, such as stalk length and feeding disk size, influence feeding currents and their interactions with external flows.

1:03PM R39.00002 Does the stalk contractility of Vorticella convallaria depend on the stalk length?, EUN-GUL CHUNG, SANGJIN RYU, University of Nebraska-Lincoln — Vorticella convallaria is a sessile stalked ciliate living in water, and its stalk coils to move the cell body (zooid) towards its residence substrate at a maximum speed of ∼ 50 mm/s. Our previous microfluidics study shows that the isometric tension of the V. convallaria stalk is linearly proportional to the stalk length. Based on this observation, we hypothesize that the contractility of V. convallaria during normal contraction is also dependent on the stalk length. To investigate our hypothesis, we measured the contraction speed of V. convallaria using high-speed videography and evaluated the contractile force and energetics of V. convallaria using fluid dynamics modeling.

1:16PM R39.00003 Traction reveals mechanisms of wall-effects for microswimmers near boundaries, XINHUI SHEN, - MARCOS, Nanyang Technological University, Singapore, HENRY C. FU, University of Nevada, Reno — Swimming of microorganism near solid boundaries plays an important role in various biological processes, such as biofilm formation and the early stage of infection. The influence of a plane boundary on low-Reynolds number swimmers has frequently been studied using image systems for flow singularities. However, the effect of a boundary can also be expressed in terms of the flow caused by the force or traction exerted by the boundary on the fluid. Here we show that examining the traction pattern on the boundary caused by a nearby swimmer can yield physical insight into the effect of the boundary on swimming velocities. To illustrate this point, we investigate a three-sphere swimmer initially placed parallel to a solid planar wall. The three spheres are modelled as three stokeslets and the method of images for a stokeslet is employed to solve for the traction on the wall. When the swimmer is close to the boundary, the middle sphere and end spheres produce a quadrupolar and dipolar time-averaged traction, respectively, reflecting the internal structure of the swimmer. Far away from the boundary, the time-averaged traction of the swimmer is similar to that of a pure far-field quadrupole. Thus the traction patterns reveal how close the swimmer must be to the boundary for the internal structure of the swimmer to influence the boundary effects.

1:29PM R39.00004 Propulsion of flexible helical flagella near a rigid boundary, MOHAMMAD JAWED, PEDRO REIS, Massachusetts Institute of Technology — We study the locomotion of uni-flagellar bacteria in a viscous fluid at low Reynolds number near a rigid boundary, through a combination of computer simulations and experiments. In our analogue model experiments, we exploit the prominence of geometry of this class of problems to rescale the original micron-scale system onto the desktop-scale. We manufacture elastomeric filaments with fully customizable geometric and material properties, and rotate them in a gelatin bath at a finite distance away from a rigid boundary. The experimental results are compared against numerical simulations that employ the Discrete Elastic Rods method in conjunction with Lighthill Slender Body Theory. The non-slip boundary condition on the wall is implemented by the method of images. We first show that the filament buckles above a critical rotation frequency due to fluid loading, and then quantify the dependence of this critical threshold on the distance from the boundary, both experimentally and numerically. Excellent agreement is found between the two, with no fitting parameters. We then make use of our numerics to systematically investigate the change in the generated propulsion due to presence of a nearby boundary. We find that the propulsion depends strongly on the location of the boundary.

1:42PM R39.00005 The biofouling potential of flow on corrugated surfaces, GASTÓN L. MIÑO, ROBERTO RUSCONI, Civil and Environmental Engineering, MIT, USA, VASILY KANTSbler, Department of Physics, University of Warwick, United Kingdom, ROMAN STOKOWER, Institute of Environmental Engineering, ETH Zurich, Switzerland — Both natural and man-made surfaces are rarely smooth, and are instead often characterized by geometric heterogeneity or roughness over a broad range of scales. Because of the predicted importance of the local interaction between microorganisms and surfaces, roughness at the microbical scale can be an important element in determining the outcome of microbe-surface interactions, which represent the first step in biofilm formation and biofouling. In microbial habitats this interaction often occurs in flowing fluids, which can be important because regions with high hydrodynamic shear can induce a strong reorientation of bacteria towards surfaces, promoting attachment. Here we study the combination of flow and surface topography using video microscopy of Escherichia coli in corrugated microfluidic channels. We report that flow preferentially promotes attachment to specific regions of a corrugated surface, as result of the hydrodynamics of bacteria swimming in flow. We compute from the data a “Local Biofouling Potential” (LBP) and compare this successfully with predictions of a mathematical model, yielding one step towards the ability to mechanistically predict and thus ultimately either prevent or induce biofouling.

We appreciate support from UNL Layman Seed Grant and Nebraska EPSCoR First Award Grant.
1:55PM R39.00006 The Effect of Brownian Motion on the Trajectory of Diffusiophoretic Locomotors near a Solid Boundary, ALI MOZAFFARI, Department of Chemical Engineering and Levich Institute, City College of New York, NIMA SHARIFI-MOOD, Department of Chemical and Biomolecular Engineering, University of Pennsylvania, JOEL KOPLIK, Department of Physics and Levich Institute, City College of New York, CHARLES MALDARELLI, Department of Chemical Engineering and Levich Institute, City College of New York — Diffusiophoretically self-propelled locomotors are a class of active colloids in which a particle autonomously swims through the liquid as a result of an unbalanced interaction with solute molecules asymmetrically distributed around the colloid. This solute distribution is maintained by a reaction which produces the solute on one catalytically active side of the Janus motor colloid. For the simplest case of diffusiophoretic self-propulsion near a planar infinite wall with zero solute flux, and repulsive solute-colloid interactions, hydrodynamic solutions for deterministic Stokes flow have shown that that for large catalytically active areas pointed away from the wall, and for distances less than the particle radius, the particles can skim at a constant distance along the surface without rotation, or can become stationary. To examine the effect of thermal fluctuations on the stability of these regimes for small motor sizes, Brownian dynamics simulations including the hydrodynamic interaction with the wall are undertaken, and we identify critical Peclet numbers above which the skimming and stationary regimes are stable. Below these values, less predictable behavior is found in which the colloid can be repelled from or intersect with the wall.

2:08PM R39.00007 Interactions of micro-organisms near a wall in Stokes flow using a regularized image system, JIANJUN HUANG, SARAH OLSON, Worcester Polytechnic Institute — We present an extension of the regularized image system for Stokeslets, where regularization functions and parameters are chosen to satisfy zero flow at the wall for several different fundamental solutions. We study elastic rods near a wall using a Kirchhoff rod formulation. Results are presented for equilibrium states of straight rods and the effect of wall to the time required to reach equilibrium.

2:21PM R39.00008 Efficiency of air/liquid interfaces in detaching bacteria from a surface1, SEPIDEH KHODAPARAST, HOWARD STONE, Princeton University — Gas/liquid interfaces are known to be significantly more effective than shear stress in detaching microscale colloids from substrates by inducing surface tension forces. Providing that a three-phase contact is formed at the interface of a gas bubble, the liquid phase and the particle occurs, the magnitude of the surface tension force can potentially exceed by orders of magnitude the adhesion force, which keeps the micro particles on the surface. We investigate the ability of a moving air/liquid interface to stimulate the detachment of bacteria from a surface. Bacteria are micron-sized living organisms with strong tendency to attach to almost any substrate that they come into contact with. Attachment of bacteria on the surface is a complex process regulated by diverse characteristics of their growth medium, substrate, and cell surface. Moreover, once fixed on the surface, the microorganisms evolve in time to create intricate biofilm structures, which are highly challenging to be removed. The objective of this study is to characterise the efficiency of this detachment process as a function of bacterial attachment as well as hydrodynamic parameters such the surface tension and the interface velocity.

1Swiss National Science Foundation P2ELP2-158896

2:34PM R39.00009 Microfluidic experiments to quantify microbes encountering oil water interfaces1, JIAN SHENG, MARYAM JALALI, MEHDI MOLAEE, Texas Tech University — It is known that marine microbes are one of the components of biodegradation of crude oil. Biodegradation of crude oil is initiated by microbes encountering the droplet. To elucidate the key processes involved in bacterial encountering the rising oil droplets we have established microfluidic devices with hydrophilic interactions of micro-organisms near a wall in Stokes flow using a regularized image system, JIANJUN HUANG, SARAH OLSON, Worcester Polytechnic Institute — We present an extension of the regularized image system for Stokeslets, where regularization functions and parameters are chosen to satisfy zero flow at the wall for several different fundamental solutions. We study elastic rods near a wall using a Kirchhoff rod formulation. Results are presented for equilibrium states of straight rods and the effect of wall to the time required to reach equilibrium.

2:47PM R39.00010 Living on the edge: transfer and traffic of E. coli in a confined flow, NURIS FIGUEROA-MORALES, PMMH, UMR 7636 CNRS, ESPCI, France, GASTON MIÑO, EMG, MIT, U.S.A, ARAMIS RIVERA, ROGELIO CABALLERO, ERNESTO ALTSHULER, Facultad de Física, Universidad de la Habana, Cuba, ERIC CLÉMENT, ANKE LINDNER, PMMH, UMR 7636 CNRS, ESPCI, France — We quantitatively study the transport of E. coli near the walls of confined microfluidic channels, and in more detail along the edges formed by the interception of two perpendicular walls. Our experiments establish the connection between bacteria motion at the flat surface and at the edges and demonstrate the robustness of the upstream motion at the edges. Upstream migration of E. coli at the edges is possible at much larger flow rates compared to motion at the flat surfaces. Interestingly, the bacteria speed at the edges mainly results from collisions between bacteria moving along this single line. We show that upstream motion not only takes place at the edge but also in an “edge boundary layer” whose size varies with the applied flow rate. We quantify the bacteria fluxes along the bottom walls and the edges and show that the result from both the transport velocity of bacteria and the decrease of surface concentration with increasing flow rate due to erosion processes. We rationalize our findings as a function of the local variations of the shear rate in the rectangular channels and hydrodynamic attractive forces between bacteria and walls.

3:00PM R39.00011 Position and trajectories of helical microswimmers inside circular channels, HAKAN CALDAG, SERHAT YESILYURT, Sabanci University — This work reports the position and orientation of helical mm-sized microswimmers in circular channels obtained by image processing of recorded images. Microswimmers are biologically inspired structures with huge potential for medical practices such as delivery of potent drugs into tissues. In order to understand the hydrodynamic effects of confinement on the velocity and stability of trajectories of swimmers, we developed helical microswimmers with a magnetic head and a rigid helical tail, similar to those of E. coli bacteria. The experiments are recorded using a digital camera, which is placed above the experimental setup that consists of a circular channel containing the magnetic field. Microswimmers move in the direction of the head, i.e. pushed kinematically by the tail, has helical trajectories, which are more unstable in the presence of Poiseuille flow inside the channel; and the swimmers that are pulled by the tail, have trajectories that stabilize at the centerline of the channel.

1GoMRI

3:10PM R39.00012 A new mathematical model to predict the encounter rate of myobacteria with oil droplets and solid particles, JIANJUN HUANG, SARAH OLSON, Worcester Polytechnic Institute — Gas/liquid interfaces are known to be significantly more effective than shear stress in detaching microscale colloids from substrates by inducing surface tension forces. Providing that a three-phase contact is formed at the interface of a gas bubble, the liquid phase and the particle occurs, the magnitude of the surface tension force can potentially exceed by orders of magnitude the adhesion force, which keeps the micro particles on the surface. We investigate the ability of a moving air/liquid interface to stimulate the detachment of bacteria from a surface. Bacteria are micron-sized living organisms with strong tendency to attach to almost any substrate that they come into contact with. Attachment of bacteria on the surface is a complex process regulated by diverse characteristics of their growth medium, substrate, and cell surface. Moreover, once fixed on the surface, the microorganisms evolve in time to create intricate biofilm structures, which are highly challenging to be removed. The objective of this study is to characterise the efficiency of this detachment process as a function of bacterial attachment as well as hydrodynamic parameters such as the surface tension and the interface velocity.

3:40PM R39.00013 Efficiency of air/liquid interfaces in detaching bacteria from a surface1, SEPIDEH KHODAPARAST, HOWARD STONE, Princeton University — Gas/liquid interfaces are known to be significantly more effective than shear stress in detaching microscale colloids from substrates by inducing surface tension forces. Providing that a three-phase contact is formed at the interface of a gas bubble, the liquid phase and the particle occurs, the magnitude of the surface tension force can potentially exceed by orders of magnitude the adhesion force, which keeps the micro particles on the surface. We investigate the ability of a moving air/liquid interface to stimulate the detachment of bacteria from a surface. Bacteria are micron-sized living organisms with strong tendency to attach to almost any substrate that they come into contact with. Attachment of bacteria on the surface is a complex process regulated by diverse characteristics of their growth medium, substrate, and cell surface. Moreover, once fixed on the surface, the microorganisms evolve in time to create intricate biofilm structures, which are highly challenging to be removed. The objective of this study is to characterise the efficiency of this detachment process as a function of bacterial attachment as well as hydrodynamic parameters such as the surface tension and the interface velocity.

1Swiss National Science Foundation P2ELP2-158896
3:13PM R39.00012 Coupled Rapid Cell and Lattice Boltzmann Models to Simulate Hydrodynamics of Bacterial Transport in Response to Chemoattractant Gradients in Confined Domains, CAMERON MCKAY, HOA NGUYEN, Trinity University, HAKAN BASAGAOGLU, ALEXANDER CARPENTER, Southwest Research Institute, SAURO SUCCI, Southwest Research Institute, FRANK HEALY, Trinity University — The Rapid Cell (RC) model was developed to simulate motility and adaptation dynamics of flagellar bacterial chemotaxis in environments with spatiotemporal variations in chemoattractant gradients. RC is best suited to motility studies in unbounded domains within non-fluid environments; this limits its use as a simulation tool. In this study, we eliminate these constraints by dynamically coupling RC with the colloidal lattice-Boltzmann (LB) model, a versatile tool for simulating transport of particles (e.g., surrogate bacteria) of distinct shapes and finite sizes in transient flow fields in geometrically complex microchannels. This was accomplished by tracking positions of chemoreceptor clusters on the particle surface that vary with particle angular and translational velocities, and by including additional forces and torques due to particles tumbling and to running motions in particle force-balance and torque-balance equations. The coupled RC-LB model successfully simulated transport of multiple particles in confined domains with single- or multi-attraction sources in a variety of settings. The coupled RC-LB model represents a first step toward development of a new modeling capability to simulate chemotaxis-driven bacterial transport in a network of geometrically irregular flow channels typically observed in tumor vasculature in the context of targeted drug delivery.

Tuesday, November 24, 2015 12:50PM - 3:13PM — Session R40 Focus Session: Reconfiguration II
Sheraton Back Bay D - Mitul Luhar, University of Southern California

12:50PM R40.00001 On the dynamics of flexible blades in oscillatory flows, MITUL LUHAR, University of Southern California, HEIDI NEPF, Massachusetts Institute of Technology — We present an experimental and numerical study that describes the motion of flexible blades, scaled to be dynamically similar to natural aquatic plants, forced by wave-induced oscillatory flows. For the conditions tested, blade motion is governed primarily by two dimensionless variables: the Cauchy number, Ca, which represents the ratio of the hydrodynamic forcing to the restoring force due to blade stiffness, and the ratio of the blade length to the wave orbital excursion, L. For flexible blades with Ca >> 1, the relationship between drag and velocity can be described by two different scaling laws. For large excursions (L << 1), the flow resembles a unidirectional current and the scaling laws developed for steady-flow reconfiguration studies hold. For small excursions (L >> 1), the beam equations may be linearized and a different scaling law for drag applies. The numerical model employs the Morison force formulation, and adequately reproduces the experimentally measured forces and blade postures. In some cases with Ca ~ O(1), the measured forces generated by the flexible blades exceed those generated by rigid blades. Observations of blade motion suggest that this behavior is related to an unsteady vortex shedding event, which the quasi-steady numerical model cannot reproduce.

1:03PM R40.00002 Mechanisms and models which govern bending and reconfiguring of trees under water flow action, CATHERINE WILSON, Cardiff University, PETER WHITTAKER, JBA Consulting, HYDROENVIRONMENTAL RESEARCH CENTRE TEAM — A model for predicting the drag and reconfiguration of flexible vegetation under hydrodynamic loading is presented. The model is based on a refined “vegetative” Cauchy number to incorporate the magnitude and rate of a tree’s reconfiguration. In addition, analysis of data from a tree drag force study conducted at the Canal de Experiencias Hidrodinamicas de El Pardo, Madrid, is also presented. This data enables the analysis of the frontal projected and the side-view areas as well as the bending angle of the main tree stems over a full range of velocities. New physical mechanisms which link tree posture, permeability, and the Reconfiguration number—Cauchy number relationship for various key stages of reconfiguration are proposed. These mechanisms are mainly developed for multi-stem trees in their foliated state. In addition direct comparisons of mechanisms for foliated and defoliated states are also presented.

1:16PM R40.00003 Reconfiguration of a flexible fiber immersed in a 2D dense granular flow close to the jamming transition, EVELYNE KOLB1, NICOLAS ALCARRA, DAMIEN VANDEMBROUCO, PMMH, ESPCI, ARNAUD LAZARUS2, IJLRA, UPMC — We propose a new fluid/structure interaction in the unusual case of a dense granular medium flowing against an elastic fibre acting as a flexible intruder. We experimentally studied the deflection of a mylar flexible beam clamped at one side, the other free side facing a 2D granular flow in a horizontal cell moving at a constant velocity. We investigated the reconfiguration of the fibre as a function of the fibre’s rigidity and of the granular packing fraction close but below the jamming in 2D. Imposing the fibre geometry like its length or thickness sets the critical buckling force the fibre is able to resist if it was not supported by lateral grains, while increasing the granular packing fraction might laterally consolidate the fibre and prevent it from buckling. But on the other side, the approach to jamming transition by increasing the granular packing fraction will be characterized by a dramatically increasing size of the cluster of connected grains forming a solid block acting against the fibre, which might promote the fibre’s deflection. Thus, we investigated the granular flow fields, the fibre’s deflexion as well as the forces experienced by the fibre and compared them with theoretical predictions from elastica for different loadings along the fibre.

1:29PM R40.00004 Effect of herbivore damage on broad leaf motion in wind, NICHOLAS BURNETT, ADIT KOTHARI, University of California, Berkeley — Terrestrial plants regularly experience wind that imposes aerodynamic forces on the plants’ leaves. Passive leaf motion (e.g., fluttering) and reconfiguration (e.g., rolling into a cone shape) in wind can affect the drag on the leaf. In the study of passive leaf motion in wind, little attention has been given to the effect of herbivory. Herbivores may alter leaf motion in wind by making holes in the leaf. Also, a small herbivore (e.g. snail) on a leaf can act as a point mass, thereby affecting the leaf’s motion in wind. Conversely, accelerations imposed on an herbivore sitting on a leaf by the moving leaf may serve as a defense by dislodging the herbivore. In the present study, we investigated how point masses (>1 g) and holes in leaves of the tuliptree affected passive leaf motion in turbulent winds of 1 and 5 m s^-1. Leaf motion was unaffected by holes in the leaf surface (about 10% of leaf area), but an herbivore’s mass significantly damped the accelerations of fluttering leaves. These results suggest that an herbivore’s mass, but not the damage it inflicts, can affect leaf motion in the wind. Furthermore, the damping of leaf fluttering from an herbivore’s mass may prevent passive leaf motions from being an effective herbivore defense.
The reconfiguration of broad leaves in strong winds and currents
LAURA MILLER, University of North Carolina at Chapel Hill, ALEX HOOVER, Tulane University, JEREMY MARZUOLA, University of North Carolina at Chapel Hill — Flexible plants, fungi, and sessile animals are thought to reconfigure in the wind and water to reduce the drag forces that act upon them. In strong winds, for example, leaves roll up into cone shapes that reduce flutter and drag when compared to paper cut-outs with similar shape and flexibility. Simple mathematical models of a flexible beam immersed in a two-dimensional flow will also exhibit this behavior. What is less understood is how the mechanical properties of a two-dimensional leaf in a three-dimensional flow will passively allow roll up and reduce drag and flutter. In this project, we use computational fluid dynamics and particle image velocimetry to determine how leaves roll up into drag reducing shapes in extreme conditions. Force and flow measurements are taken on real broad leaves and simplified physical models. Corresponding numerical simulations using the immersed boundary method are used to understand which features of the flexible leaves result in proper reconfiguration and drag reduction.

Reconfiguration parameters for drag of flexible cylindrical elements
CHAPMAN JOHN, BRUCE WILSON, JOHN GULLIVER, University of Minnesota — This presentation compares parameters that characterize reconfiguration effects on flow resistance and drag. The drag forces occurring on flexible bluff bodies are different from the drag occurring on rigid bluff bodies due to reconfiguration. Drag force data, collected using a torque sensor in a flume, for simple cylindrical obstructions of the same shape and size but with different flexibility is used to fit drag parameters. The key parameter evaluated is a reference drag occurring on rigid bluff bodies due to reconfiguration. The reference velocity factor \( u \) to account for drag reduction due to reconfiguration, similar to a Vogel exponent. Our equations preserves the traditional exponent of the drag relationship, but places a factor onto the drag coefficient for flexible elements, rather than a Vogel exponent arrangement applied to the flow velocity. Additionally we relate the reference velocity factors to the modulus of elasticity of the material through the Cauchy Number. The use of a reference velocity factor \( u \) in place of a Vogel exponent appears viable to account for how the drag forces are altered by reconfiguration. The proposed formulation for drag reduction is more consistently estimated for the range of flexibilities in this study. Unfortunately, the mechanical properties of vegetation are not often readily available for reconfiguration relationships to the elastic modulus of vegetation to be of immediate practical use.

Enhancing wind turbines efficiency with passive reconfiguration of flexible blades
VINCENT P A COGNET, BENJAMIN THIRIA, PMMH (UMR 7636 CNRS - ESPCI - UPMC Paris 6 - UPD Paris 7 - PSL), SYLVAIN COURRECH DU PONT, MSC (UMR 7057 CNPS - UPD Paris 7), MSC TEAM1, PMMH TEAM2 — Nature provides excellent examples where flexible materials are advantageous in a fluid stream. By folding, leaves decrease the drag caused by air stream; and birds’ flapping is much more efficient with flexible wings. Motivated by this, we investigate the effect of flexible blades on the performance of a wind turbine. The effect of chordwise flexible blades is studied both experimentally and theoretically on a small wind turbine in steady state. Four parameters are varied: the wind velocity, the resisting torque, the pitch angle, and the blades bending modulus. We find an optimum efficiency with respect to the bending modulus. By tuning our four parameters, the wind turbine with flexible blades has a high-efficiency range significantly larger than rigid blades1, and, furthermore enhances the operating range. These results are all the more important as one of the current issues concerning wind turbines is the enlargement of their operating range. To explain these results, we propose a simple two-dimensional model by discretising the blade along the radius. We take into account the variation of drag and lift coefficients with the bending ability. This model matches experimental observations and demonstrates the contribution of the reconfiguration of the blade.

Drag reduction by reconfiguration in gorgonians
JULIEN DERR, ANNE-MIEK J. M. CORNELLISSEN, Université Paris Diderot, CLAUDE BOUCHON, YOLANDE BOUCHON, Université des Antilles et de la Guyane, JÉRÔME FOURNIER, CNRS/ MNHN, LIONEL MOISAN, Université Paris Descartes, PASCAL JEAN LOPEZ, CNRS/ MNHN, STÉPHANE DOUADY, Université Paris Diderot — Gorgonians are polyplon colonies over a flexible branched skeleton. Attached to the coral reefs, they are under the continuous oscillations of the swell. We investigate experimentally the drag, under continuous force traction, of Gorgonia Ventalina, which is particular as its branches are highly reconnect to form a flat net (see fan), perpendicular to the swell, and compare it with another branched species (candelstick). We observe a drag which is linear with speed, indicating a strong reconfiguration, which we also documented by imaging the gorgon shape, and transients showing that the gorgon do not always evolve along quasi-static curves. Depending on the size and shape of the gorgon, we observe different details, from a more rigid small gorgon to a flexible long one. A large gorgon with detached fingers, closing on themselves under the current, presents characteristics surprisingly close to a rigid candlestick one, with not much reconfiguration.

Interfering with the wake of cylinder by flexible filaments
ALFREDO PINELLI, MOHAMMAD OMIDYEUGANEH, City University London — This work is the very first attempt to understand and optimize the configuration of flexible filaments placed on the lee side of a bluff body to manipulate flow transitions and bifurcations. It is found that the presence of a sparse set of flexible filaments on the lee side of a cylinder can interfere with the 2D-3D transition process resulting in elongation of recirculation bubble, inhibition of higher order unstable modes, and narrowing the global energy content about a particular shedding frequency. Filaments become effective when spacing between them is smaller than the dominant unstable mode at each particular Reynolds number, i.e. A and B modes. In another study, by a particular arrangement the reconfigured filaments can reduce pressure fluctuations in the wake and drop lift fluctuations significantly (\( \approx 80\% \)).

Catenaries in viscous fluid
JAMES HANNA, BRATO CHAKRABARTI1, Virginia Polytechnic Institute and State University — Slender structures live in fluid flows across many scales, from towed instruments to plant blades to microfluidic valves. The present work details a simple model of a flexible structure in a uniform flow. We present analytical solutions for the translating, axially flowing equilibria of strings subjected to a uniform body force and linear drag forces. This is an extension of the classical catenaries to a five-parameter family of solutions, represented as trajectories in angle-curvature “phase space.” Limiting cases include neutrally buoyant towed cables and freely sedimenting flexible filaments.

1now at University of California, San Diego
3:00PM R40.00011 Numerical investigation on optimizing blast wave focusing effects for multiple munitions, SHI QIU, VERONICA ELIASSON, University of Southern California — The phenomenon of blast wave focusing onto a specified target has been studied. Simulations were performed in which multiple munitions were placed in a circular pattern around a target. The number of munitions was varied through multiple cases while the total energy distributed among all munitions was held constant. Previous research shows that there exists an optimal number of munitions to produce the most extreme conditions at the target while simultaneously reducing collateral damage. Two numerical approaches, inviscid Euler equations and geometrical shock dynamics were used to study the interaction between blast waves in order to further investigate the optimization problem. To generate initial conditions for geometrical shock dynamics simulations on interaction between blast waves, it was found that a transition point between regular reflection and irregular reflection needs to be determined in advance. Both experimental and theoretical investigation is included to study the transition condition. Optimization strategy for focusing blast waves is also discussed.

Tuesday, November 24, 2015 12:50PM - 3:26PM — Session R41 Minisymposium on Turbulence in Honor of John L. Lumley Sheraton Constitution A - Nadine Aubry, Northeastern University

12:50PM R41.00001 John Leask Lumley: In Memoriam, SIDNEY LEIBOVICH, Cornell University — John Lumley, a major contributor to turbulence research in the last half century, was a complex man of many talents. This talk is a personal reflection on a friendship of long standing.

1:16PM R41.00002 Fine-scale turbulence induced axial flow and instability of a vortex column, FAZLE HUSSAIN¹, Texas Tech University — Interaction of fine-scale turbulence with a coherent vortex column and its possible induction of axial flow leading to column instability is studied using direct numerical simulations. Vortex threads form from the fine-scale turbulence due to mean strain of the column and self-advect in the (primarily) axial and radial directions. Self-advective transport is dependent on the thread circulation and orientation, and radial advection causes similar threads with opposite circulations to radially separate, resulting in an axial flow. As axial flow increases both with Reynolds number (=vortex circulation/viscosity) and in time, instability due to axial flow (indicated by the ratio of maximum azimuthal velocity to maximum axial velocity, or the q criterion) can cause perturbation growth. For the simplified perturbation of two oppositely oriented vortex threads, axial flow is generated and, at sufficient amplitudes, perturbation amplification occurs, possibly leading to instability.

¹Co-authored by E. Stout

1:29PM R41.00003 Reduced order modeling of wall turbulence, PARVIZ MOIN, Center for Turbulence Research, Stanford University — Modeling turbulent flow near a wall is a pacing item in computational fluid dynamics for aerospace applications and geophysical flows. Gradual progress has been made in statistical modeling of near wall turbulence using the Reynolds averaged equations of motion, an area of research where John Lumley has made numerous seminal contributions. More recently, Lumley and co-workers² pioneered dynamical systems modeling of near wall turbulence, and demonstrated that the experimentally observed turbulence dynamics can be predicted using low dimensional dynamical systems. The discovery of minimal flow unit³ provides further evidence that the near wall turbulence is amenable to reduced order modeling. The underlying rationale for potential success in using low dimensional dynamical systems theory is based on the fact that the Reynolds number is low in close proximity to the wall. Presumably for the same reason, low dimensional models are expected to be successful in modeling of the laminar/turbulence transition region. This has been shown recently using dynamic mode decomposition. Furthermore, it is shown that the near wall flow structure and statistics in the late and non-linear transition region is strikingly similar to that in higher Reynolds number fully developed turbulence. In this presentation, I will argue that the accumulated evidence suggests that wall modeling for LES using low dimensional dynamical systems is a profitable avenue to pursue. The main challenge would be the numerical integration of such wall models in LES methodology.

¹Aubry et al., JFM, 192, 1988.
³Sayadi et al., JFM, 748, 2014.
⁴Sayadi et al., JFM, 724, 2013.

1:42PM R41.00004 POD analysis of turbulent pipe flow, ALEXANDER J. SMITS, Princeton University, Monash University, LEO HELSTROM, Princeton University, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton — Proper Orthogonal Decomposition was introduced into the analysis of turbulent flow by Lumley (1967, 1981). Turbulent flows pose particular challenges for POD analysis because the energy is distributed over a wide range of scales. It has recently been found, however, that POD can be a powerful experimental tool for identifying the largest scales, especially the Large Scale Motions (LSMs) and Very Large Scale Motions (VLSMs) in turbulent pipe flow. It has also been useful, for example, to identify the large-scale motions that dominate the unsteady behavior of the flow downstream of a right-angled bend. Here, we summarize some of these experimental results, and discuss their implications for the understanding of turbulence structure.

¹Supported under ONR Grant N00014-13-1-0174 and ERC Grant No. 277472.

1:55PM R41.00005 Low dimensional modeling of wall turbulence, NADINE AUBRY⁴, Northwestern University — In this talk we will review the original low dimensional dynamical model of the wall region of a turbulent boundary layer [Aubry, Holmes, Lumley and Stone, Journal of Fluid Dynamics 192, 1988] and discuss its impact on the field of fluid dynamics. We will also invite a few researchers who would like to make brief comments on the influence Lumley had on their research paths.

⁴In collaboration with Philip Holmes, Program in Applied and Computational Mathematics and Department of Mechanical and Aerospace Engineering, Princeton University, Princeton, NJ.
important consequences for single scale models. With Lumley, I carried out scalar decay measurements and found dependence of the scalar.

and others were making rapid progress in the development of second order and other models for the prediction of turbulent flows. It became

dependent on the decay of passive scalars (such as temperature fluctuations) in grid turbulence showed large variations, with important consequences for single scale models. With Lumley, I carried out scalar decay measurements and found dependence of the scalar variance decay rate on the initial scalar length scale. These experiments led to many more on passive scalars, both on their large and small-scale characteristics. Here I describe my interactions with John Lumley during this period, and relate it to the work of other groups. I also show that there are still unresolved problems in this area.

second-moment closures represent the simplest level at which the physics of turbulent flows can reasonably be represented. This is especially

true when the velocity field is coupled to scalar fields through buoyancy, as in the atmosphere and oceans. While Lumley was not the first to propose second-moment closures, he can be credited with establishing the rational approach to constructing such closures. This includes the application of various invariance principles and tensor representation theorems, imposing the constraints imposed by realizability, and of course appealing to experimental data in simple, canonical flows. These techniques are now well-accepted and have found application far beyond second-moment closures.

In collaboration with Adrien Thormann, Johns Hopkins University.

of kinetic energy due to pressure-velocity correlations, although its magnitude required to close the budget appears very large. Absence of dissipation enable us to obtain the spanwise gradient of the spatial flux. One possible explanation for the observations is upgrading transport.

In collaboration with Shravan Hanasoge, Tata Institute of Fundamental Research, Mumbai and Laurent Gizon, Max-Planck-Institut fuer Sonnensystemforschung, Goettingen.

A driving mechanism and provide an important benchmark for numerical simulations. This discrepancy challenges the models of internal differential rotation that rely on convective stresses as a driving mechanism and provide an important benchmark for numerical simulations.

New York University — Thermal convection is the dominant mechanism of energy transport in the outer envelope of the Sun (one-third by radius). It drives global fluid circulations and magnetic fields observed on the solar surface. Convection excites a broadband spectrum of acoustic waves that propagate within the interior and set up modal resonances. These acoustic waves, also called seismic waves, are observed at the surface of the Sun by space- and ground-based telescopes. Seismic sounding, the study of these seismic waves to infer the internal properties of the Sun, constitutes helioseismology. Here we review our knowledge of solar convection, especially that obtained through seismic inference. Several characteristics of solar convection, such as differential rotation, anisotropic Reynolds stresses, the influence of rotation on convection and supergranulation, are considered. On larger scales, several inferences suggest that convective velocities are substantially smaller than those predicted by theory and simulations. This discrepancy challenges the models of internal differential rotation that rely on convective stresses as a driving mechanism and provide an important benchmark for numerical simulations.

3:13PM R41.00011 Decay and Spatial Diffusion of Turbulent Kinetic Energy In The Presence of a Linear Kinetic Energy Gradient1, CHARLES MENEVEAU, Johns Hopkins University — A topic that elicited the interest of John Lumley is pressure transport in turbulence. In 1978 (JL, in Advances in Applied Mechanics, pages 123-176) he showed that pressure transport likely acts in the opposite direction to the spatial flux of kinetic energy due to triple velocity correlations. Here we examine a flow in which the interplay of turbulent decay and spatial transport is particularly relevant. Specifically, using a specially designed active grid and screens placed in the Corrsin wind tunnel, such a flow is realized. Data are acquired using X-wire thermal anemometry at different spanwise and downstream locations. In order to resolve the dissipation rate accurately, measurements are also acquired using the NSTAP probe developed and manufactured by Princeton researchers and kindly provided to us (M. Hultmark, Y. Fan, L. Smits). The results show power-law decay with downstream distance, with a decay exponent that becomes larger in the high kinetic energy side of the flow. Measurements of the dissipation enable us to obtain the spanwise gradient of the spatial flux. One possible explanation for the observations is upgrading transport of kinetic energy due to pressure-velocity correlations, although its magnitude required to close the budget appears very large. Absence of simultaneous pressure velocity measurement preclude us to fully elucidate the observed trends.

In collaboration with Adrien Thormann, Johns Hopkins University.

Financial support: National Science Foundation.