

69th Annual Meeting of the APS Division of Fluid Dynamics
Portland, Oregon
<http://www.aps.org/meetings/meeting.cfm?name=DFD16>

Sunday, November 20, 2016 8:00AM - 9:44AM – Session A1 Mini-Symposium: Geophysical Turbulence Induced by Flow over Topography | A105 - Ali Mashayek, Massachusetts Institute of Technology

8:00AM A1.00001 Internal Wave Breaking in Stratified Flow over Topography W

RICHARD PELTIER, Univ of Toronto — In both atmosphere and oceans, internal waves generated by stratified flow over topography “break” when a critical Froude number is exceeded. In the oceans, the global field of such waves forced by the flow of the barotropic tide over bottom topography constitutes an “internal tide”, the turbulent dissipation of which contributes significantly to the diapycnal diffusivity of mass in the abyss. In the atmosphere, the vertical flux of horizontal momentum in the wave field plays an important role in mediating the strength of the mid-latitude jet streams in the troposphere through the “gravity wave drag” that is applied to the mean zonal flow when the waves break. Early work on the atmospheric problem based upon the application of LES methods demonstrated that, in the restricted case of topographically forced 2-D flows, wave breaking aloft led to the development of an intense low level jet in the lee of the topographic maximum, in which an intense secondary instability of Kelvin-Helmholtz type developed which became intensely turbulent. The same methods were later applied to the oceans, initially to develop an understanding of the tidally induced breaking wave mechanics in the Knight Inlet “flume”. Similar dynamical interactions, to those observed in the atmosphere in connection with severe downslope windstorm formation, have been observed to occur in the deep ocean in the lee of ocean bottom topographic extrema. Current work is underway to determine the extent to which DNS methods applied to the oceanographic context are able to recover the phenomenology revealed by the atmospheric LES analyses.

8:26AM A1.00002 Tidal bores, turbulence and mixing above deep-ocean slopes , KRAIG

WINTERS, Scripps Institution of Oceanography, UCSD — A tidally driven, stably-stratified turbulent boundary layer over supercritically sloping topography is simulated numerically using a spectral LES approach (Winters, 2015, 2016). The near boundary flow is characterized by quasi-periodic, bore-like motions, whose temporal signature is compared to the high-resolution ocean mooring data of van Haren (2006). The relatively thick bottom boundary layer remains stably stratified owing to the regular cycling of unmixed ambient fluid into the turbulent boundary layer and episodic expulsion events where fluid is ejected into the stratified interior. The effective diffusivity of the flow near the boundary is estimated by means of a synthetic dye tracer experiment. The average dissipation rate within the dye cloud is computed and combined with the diffusivity estimate to yield an overall mixing efficiency of 0.15. Both the estimated diffusivity and dissipation rates are in reasonable agreement with the microstructure observations of Kunze et al (2012) when scaled to the environmental conditions at the Monterey and Soquel Canyons and to the values estimated by van Haren and Gostiaux (2012) above the sloping bottom of the Great Meteor Seamount in the Canary Basin.

8:52AM A1.00003 Tidally driven mixing: breaking lee waves, hydraulic jumps and the influence of subinertial internal tides , RUTH MUSGRAVE, Massachusetts Institute of Technology — We present observations of tidally driven turbulence that were obtained in a small channel that transects the crest of the Mendocino Ridge, a site of mixed (diurnal and semidiurnal) tides. At this latitude the diurnal tide is subinertial and evanescent away from the topography, in contrast to the semidiurnal tide which is superinertial and radiating. During the larger of the daily tides, strong turbulence (10 W/m^2) is observed, and using a high resolution, two-dimensional, nonhydrostatic simulation, we interpret observed flow features and concomitant turbulent dissipation to arise from both breaking internal lee waves (above the crest of the topography), and turbulent hydraulic jumps (on the flanks of the topography). To understand the nature of the regional scale flows we employ a three-dimensional tidally forced model, which illustrates the presence of diurnal bottom-trapped internal waves. These energetic waves are of leading order importance in determining the timing of the dissipative processes, alternately enhancing and canceling near bottom flows that are of critical importance to near-topographic turbulence.

9:18AM A1.00004 Parameterizing turbulence over abrupt topography¹ , JODY KLYMAK,

University of Victoria — Stratified flow over abrupt topography generates a spectrum of propagating internal waves at large scales, and non-linear overturning breaking waves at small scales. For oscillating flows, the large scale waves propagate away as internal tides, for steady flows the large-scale waves propagate away as standing “columnar modes”. At small-scales, the breaking waves appear to be similar for either oscillating or steady flows, so long as in the oscillating case the topography is significantly steeper than the internal tide angle of propagation. The size and energy lost to the breaking waves can be predicted relatively well from assuming that internal modes that propagate horizontally more slowly than the barotropic internal tide speed are arrested and their energy goes to turbulence. This leads to a recipe for dissipation of internal tides at abrupt topography that is quite robust for both the local internal tide generation problem (barotropic forcing) and for the scattering problem (internal tides incident on abrupt topography). Limitations arise when linear generation models break down, an example of which is interference between two ridges. A single “super-critical” ridge is well-modeled by a single knife-edge topography, regardless of its actual shape, but two supercritical ridges in close proximity demonstrate interference of the high modes that makes knife-edge approximations invalid. Future direction of this research will be to use more complicated linear models to estimate the local dissipation. Of course, despite the large local dissipation, many ridges radiate most of their energy into the deep ocean, so tracking this low-mode radiated energy is very important, particularly as it means dissipation parameterizations in the open ocean due to these sinks from the surface tide cannot be parameterized locally to where they are lost from the surface tide, but instead lead to non-local parameterizations.

¹US Office of Naval Research; Canadian National Science and Engineering Research Council

Sunday, November 20, 2016 8:00AM - 9:44AM –

Session A2 Wind Turbines: Airfoils A106 - Jonathan Naughton, University of Wyoming

8:00AM A2.00001 Design and analysis of small wind turbine blades with wakes similar to those of industrial scale turbines¹ , ARASH HASSANZADEH, JONATHAN NAUGHTON, University of Wyoming — A new design approach has been developed for wind turbine blades to be used in wind tunnel experiments that study wind turbine wakes. The approach allows wakes of small scale (2 m diameter) wind turbine rotors to simulate the important physics of wakes generated by a “parent” industrial scale wind turbine rotor despite the difference in size. The design approach forces the normalized normal and tangential force distributions of the small scale wind turbine blades to match those of the “parent” industrial scale wind turbine blades. The wake arises from the interaction between the flow and the blade, which imparts a momentum deficit and rotation to the flow due to the forces created by the blade on the flow. In addition, the wake dynamics and stability are affected by the load distribution across the blade. Thus, it is expected that matching normalized force distributions should result in similar wake structure. To independently assess the blades designed using this approach, the “parent” industrial scale and small scale wind turbine rotors are modeled using a free vortex wake method to study the generation and evolution of the two wakes.

¹This work was supported by the U.S. Department of Energy, Office of Science, Basic Energy Sciences, under Award DE-SC0012671.

8:13AM A2.00002 Three-dimensional effects on airfoil measurements at high Reynolds numbers¹, JANIK KIEFER, MARK MILLER, MARCUS HULTMARK, MARTIN HANSEN, Princeton Univ — Blade Element Momentum codes (BEM) are widely used in the wind turbine industry to determine a turbines operational range and its limits. Empirical two-dimensional airfoil data serve as the primary and fundamental input to the BEM code. Consequently, the results of BEM simulations are strongly dependent on the accuracy of these data. In this presentation, an experimental study is described in which airfoils of different aspect ratios were tested at identical Reynolds numbers. A high-pressure wind tunnel facility is used to achieve large Reynolds numbers of $Re_c = 3 \times 10^6$, even with small chord lengths. This methodology enables testing of very high aspect ratio airfoils to characterize 3-D effects on the lift and drag data. The tests were performed over a large range of angles of attack, which is especially important for wind turbines. The effect of varying aspect ratio on the aerodynamic characteristics of the airfoil is discussed with emphasis on the outcome of a BEM simulation.

¹The project was partially funded by NSF CBET-1435254 (program manager Dr. Gregory Rorrer).

8:26AM A2.00003 Wind turbine airfoil investigations in customized turbulent inflow¹, HENDRIK HEISSELMANN, JOACHIM PEINKE, MICHAEL HOELLING, ForWind - Institute of Physics, University of Oldenburg — Experimental airfoil characterizations are usually performed in laminar or unsteady periodical flows. Neither of these matches the flow conditions of natural atmospheric flows as experienced by wind turbine blades. In the presented experimental study, an active grid is used to generate turbulent inflow with customized properties, like reduced frequencies or inflow angles. This is used not only to tune flow properties, but also to mimic time series of measured atmospheric wind speeds and inflow angles in the wind tunnel. Experiments were performed on a wind turbine dedicated DU 00-W-212 airfoil to obtain highly resolved force data and chord-wise pressure distributions at $Re=500,000$ and $Re=900,000$. Additional to a laminar baseline case, unsteady sinusoidal inflow fluctuations were applied as well as three different turbulent inflows with comparable turbulence intensity, but different inflow angle fluctuations to grasp the impact of inflow characteristics on the airfoil performance. In comparison with the laminar inflow case, the lift peak of the polar is shifted to higher angles of attack in the turbulent flows. While the laminar lift polars show a rather sudden transition to stall, a softer transition with an extended stall region is found for all turbulent cases.

¹The presented work was performed within the project AVATAR and is funded from the European Unions Seventh Program for research, technological development and demonstration under grant agreement No FP7-ENERGY-2013-1/n 608396.

8:39AM A2.00004 Comparison of Blade Element Momentum Theory to Experimental Data Using Experimental Lift, Drag, and Power Data, TARA NEALON, MARK MILLER, JANIK KIEFER, MARCUS HULTMARK, Princeton Univ — Blade Element Momentum (BEM) codes have often been used to simulate the power output and loads on wind turbine blades without performing CFD. When computing the lift and drag forces on the blades, the coefficients of lift and drag are normally calculated by interpolating values from standard airfoil data based on the angle of attack. However, there are several empirical corrections that are needed. Due to a lack of empirical data to compare against, the accuracy of these corrections and BEM in general is still not well known. For this presentation, results from an in-house written BEM code computed using experimental lift and drag coefficient data for the airfoils of the V27 wind turbine will be presented. The data is gathered in Princeton University's High Reynolds Number Testing Facility (HRTF) at full scale Reynolds numbers and over a large range of angles of attack. The BEM results are compared to experimental data of the same wind turbine, conducted at full scale Reynolds number and TSR, also in the HRTF. Conclusions will be drawn about the accuracy of the BEM code, and the corrections, regarding the usage of standard airfoil data versus the experimental data, as well as future applications to potentially improve large-eddy simulations of wind turbines in a similar manner.

8:52AM A2.00005 ABSTRACT WITHDRAWN —

9:05AM A2.00006 Synthetic-jet-based dynamic stall control on a scaled finite span wind turbine S817 blade, THOMAS RICE, KEITH TAYLOR, MICHAEL AMITAY, Rensselaer Polytechnic Institute — As wind turbines increase in size, so do many of the adverse effects associated with unsteady flow fields. Yawed flow, unsteady gusts, atmospheric boundary layers, and even free stream turbulence can cause unsteady loading, which are detrimental to the blades' structure. In order to decrease unsteady loading, synthetic jet actuators were installed on a scaled finite span cantilevered wind turbine blade having an S817 airfoil shape. The S817 airfoil shape is of the blade tips on the NREL CART3, which will be used next year on full scale field testing of active flow control. The model has been tested in the wind tunnel with and without active flow control, using load, surface pressure, and PIV measurements to characterize the airfoil's stall behavior during static and dynamic conditions, and the effect of flow control on its aerodynamic performance. Surface-mounted microphones were also used to detect dominant frequencies in the flow field. Dynamic stall was also simulated by pitching the airfoil through stall in a sinusoidal pitching motion. Synthetic jets, placed near the leading edge, were shown to increase lift both in the static and dynamic cases, in addition to attaching the flow and reducing hysteresis during dynamic pitching, showing a decrease in structural loading.

9:18AM A2.00007 A comparison between 2-and 3-bladed wind turbine rotors with focus on wake characteristics, FRANZ MHLE, MUJIWA SAMUEL ADARAMOLA, NMBU, s, LARS STRAN, NTNUI, Trondheim — Due to cost benefit and weight reduction, 2-bladed wind turbines have the potential to become more important for offshore wind applications. In order to optimize the arrangement of wind turbines in wind farms and for accurate forecasts of the power production, a detailed knowledge of the wake flow is needed. In the presented study, three different rotors with varying number of blades and similar performance behavior have been designed and manufactured using the 3-dimensional (3D) printing technology. The performance characteristics of these rotors as well as their wake features are measured experimentally in wind tunnel tests and compared. The velocity deficit is seen to vary only insignificantly for the wakes in distances of 3D (where D is the rotor diameter), 5D and 7D behind the turbine. However, higher turbulence intensity levels are recorded in the wake of the 2-bladed rotors. This could have potential for a faster wake recovery and thus a narrower turbine spacing.

9:31AM A2.00008 Evolution of the shear layer during unsteady separation over an experimental wind turbine blade, MATTHEW MELIUS, RAUL CAL, Portland State University, KAREN MULLENERS, École Polytechnique Fédérale de Lausanne — Unsteady flow separation in rotationally augmented flow fields plays a significant role in the aerodynamic performance of industrial wind turbines. Current computational models underestimate the aerodynamic loads due to the inaccurate prediction of the emergence and severity of unsteady flow separation in the presence of rotational augmentation. Through the use of time-resolved particle image velocimetry (PIV), the unsteady separation over an experimental wind turbine blade is examined. By applying Empirical Mode Decomposition (EMD), perturbation amplitudes and frequencies within the shear-layer are identified. The time dependent EMD results during the dynamic pitching cycle give insight into the spatio-temporal scales that influence the transition from attached to separated flow. The EMD modes are represented as two-dimensional fields and are analyzed together with the spatial distribution of vortices, the location of the separation point, and velocity contours focusing on the role of vortex shedding and shear layer perturbation in unsteady separation and reattachment.

Sunday, November 20, 2016 8:00AM - 9:57AM –

Session A3 Vortex Dynamics: Cylinders B110-111 - Yahya Modarress-Sadeghi, University of Massachusetts, Amherst

8:00AM A3.00001 The effect of boundary conditions on VIV of a fully submerged flexible cylinder, MAHDIAH EDRAKI, University of Massachusetts, Amherst, BANAFSHEH SEYED-AGHAZADEH, Miami University, YAHYA MODARRES-SADEGHI, University of Massachusetts, Amherst — A series of experiments was conducted in a re-circulating water tunnel, in which Vortex-Induced Vibration (VIV) of a fully submerged, tension-dominated cylinder with different boundary conditions was studied. While in most previous studies, either the cylinder was not fully submerged in flow or the boundary conditions for the cylinder were different at the two ends, in the current study the cylinder is fully submerged and the boundary conditions are carefully controlled. The cylinder was held fixed at both ends and was placed perpendicular to the uniform incoming flow direction. Different symmetric and asymmetric boundary conditions for the cylinder, i.e., clamped-clamped, simply supported, and clamped-hinged were tested. Continuous response of the cylinder in both the crossflow and inline directions were reconstructed from limited number of measurement points based on modal expansion theorem modified using Modal Assurance Criterion (MAC). Amplitudes and frequencies of oscillations were studied in the reduced velocity range of $U^*=5.5\text{--}32.5$ and the Reynolds number range of $Re = 200\text{--}1220$. Modes up to four were excited in the crossflow direction for a cylinder with a length of $L=0.3$ m and an aspect ratio of 73.

8:13AM A3.00002 Boundary layers and forces on a cylinder in vortex-induced vibrations, RAVI CHAITHANYA MYSA, Agency for Science, Technology and Research (A*STAR), SUMANA INAGANTI, Nanyang Technology University, VINH-TAN NGUYEN, Agency for Science, Technology and Research (A*STAR), CHANG WEI KANG, Institute of High Performance Computing, (A*STAR) — The flow, load on the cylinder and its motion are coupled to each other in a vortex-induced vibration (VIV). The boundary layer on the cylinder is affected by the shedding of vortices and motion of the cylinder. The motion of the cylinder will be in certain phase with respect to shedding vortices depending on the reduced velocity. Based on their phase difference, the cylinder motion dynamics can affect boundary layers differently. This, in turn changes force acting on the cylinder and subsequently its motions. It is believed that boundary layer and its movements on the cylinder play an important role in explaining the responses in vortex induced vibrations. Numerical experiments have been performed on the vortex induced vibrations of the cylinder in laminar flow environment for different mass ratios to better understand its complex coupled physics. The boundary layer on the cylinder is calculated at different locations on the cylinder during its VIV responses. The variation of the boundary layer is characterized in pre-lock-in, lock-in and post-lock-in regions with respect to the pressure and viscous forces generated on the cylinder. The connection between the forces generated, shed vortices and the motion will be presented with the help of boundary layer variation and flow contours. Their respective correlations will be established while the dependence of pattern of shedding vortices on the phase of the cylinder motion will be discussed.

8:26AM A3.00003 Symmetry breaking in flow-induced vibration of rotating prisms with non-circular cross-sections, BANAFSHEH SEYED-AGHAZADEH, Miami University, YAHYA MODARRES-SADEGHI, University of Massachusetts, Amherst — Flow-induced vibration of a flexibly-mounted triangular prism, free to oscillate in the crossflow direction with imposed rotation about its axis, was studied experimentally. Depending on the angle of attack, non-rotating prisms with triangular cross-sections could experience both Vortex-Induced Vibration (VIV) and galloping. In particular, the objective of this study was to investigate how the imposed rotation could affect the galloping instability for such prisms. The rotation rate, α , defined as the ratio of the surface velocity and the free stream velocity, was varied in the range of $\alpha=0\text{--}2.6$. The amplitudes and frequencies of oscillations were measured in a Reynolds number range of $Re=420\text{--}2100$ and reduced velocity of $Vr=2.8\text{--}14$. The oscillation was found to be limited to a range of reduced velocities, and the lock-in range became narrower at higher rotation rates where the oscillation ceased beyond the rotation rate around $\alpha=2.4$. This rotation rate at which the oscillation ceased and the decrease in the width of the lock-in range was very similar to those observed previously for VIV of a rotating circular cylinder. The tests were repeated for a square-cross-section prism and similarly, the oscillation was observed to cease at the same range of rotation rate around $\alpha=2.4$.

8:39AM A3.00004 Optical Tracking Measurement on Vortex Induced Vibration of Flexible Riser with Short-Length Buoyance Module, DIXIA FAN, Massachusetts Inst of Tech-MIT, HONGLIN DU, Tianjin University, MICHAEL TRIANTAFYLLOU, Massachusetts Inst of Tech-MIT — We address experimentally the vortex induced vibrations (VIV) of long flexible cylinders. We employ optical tracking, using an array of high speed cameras. Compared to strain gauges and accelerometers, this non-intrusive approach, allows direct measurement of the flexible cylinder displacement with far denser spatial distribution. The measurements reveal essential features of flexible cylinder VIV, including complex geometries such as cylinders containing short-length buoyancy modules, with module to cylinder diameter ratio of 1:3.2 and module to bare cylinder length ratio of 1:1. The experiments are conducted with aspect ratio of 170 and 3 different coverage ratios, of 100%, 50% and 20%. The measurements demonstrate bi-frequency response due to excitation from both buoyancy module and bare cylinder, at low Strouhal number, down to values of 0.08, and the generation of traveling wave patterns.

8:52AM A3.00005 Multiple Long-Time Solutions for Intermediate Reynolds Number Flow past a Circular Cylinder with a Nonlinear Inertial and Dissipative Attachment¹, ANTOINE B. E. BLANCHARD, LAWRENCE A. BERGMAN, ALEXANDER F. VAKAKIS, ARNE J. PEARLSTEIN, University of Illinois at Urbana-Champaign — We consider two-dimensional flow past a linearly-sprung cylinder allowed to undergo rectilinear motion normal to the mean flow, with an attached “nonlinear energy sink” consisting of a mass allowed to rotate about the cylinder axis, and whose rotational motion is linearly damped by a viscous damper. For $Re < 50$, where the flow is expected to be two-dimensional, we use different inlet transients to identify multiple long-time solutions, and to study how they depend on Re and a dimensionless spring constant. For fixed values of the ratio of cylinder density to fluid density, dimensionless damping coefficient, and ratio of the rotating mass to the total mass, we find that different inlet transients lead to different long-time solutions, including solutions that are steady and symmetric (with a motionless cylinder), time-periodic, quasi-periodic, and chaotic. The results show that over a wide range of the parameters, the steady symmetric motionless-cylinder solution is locally, but not globally, stable.

¹Supported by NSF Grant CMMI-1363231

9:05AM A3.00006 Relating surface pressure to Lagrangian wake topology around a circular cylinder in cross flow¹, MATTHEW ROCKWOOD, MELISSA GREEN, Syracuse University — The tracks of Lagrangian saddles, identified as non-parallel intersections of positive and negative-time finite-time Lyapunov exponent (FTLE) ridges, have been shown to indicate the timing of von Karman vortex shedding in the wake of bluff bodies. The saddles are difficult to track in real-time, however, since future flow field data is needed for the computation of the FTLE fields. In order to detect the topological changes without direct access to the FTLE, the saddle dynamics are correlated to measurable surface quantities on a circular cylinder in cross flow. The Lagrangian saddle found upstream of a forming and subsequently shedding vortex has been shown to accelerate away from the cylinder surface as the vortex sheds. In previous numerical results at $Re = 150$, this acceleration coincides with the peak in lift force over the cylinder, and also with a minimum in the static pressure at a location slightly upstream of the mean separation location. In the current work, this result is compared with experimental data at $Re = O(10,000)$. Successful validation would provide a strategy for locating sensitive regions on the cylinder surface where vortex shedding could be detected using simple pressure transducers.

¹This work was supported by the Air Force Office of Scientific Research under AFOSR Award No. FA9550-14-1-0210.

9:18AM A3.00007 Two-dimensional wakes of a variable diameter cylinder, WENCHAO YANG, MARK STREMLER, Virginia Tech — It is well known that periodic variations in the position of a circular cylinder can produce a variety of complex vortex wake patterns. We will discuss what we believe is the first investigation of the wake patterns produced by a stationary circular cylinder undergoing periodic variations in the cylinder diameter. In our experiments, cylinder variations are produced by oscillating a cone perpendicularly through a flowing soap film. The wake flow generates thickness variations in the thin soap film, allowing direct observation of wake patterns through visualization of interference fringes. We consider diameter variations ranging from 0.1 to 0.5 times the mean diameter, with the Reynolds number varying from 50 to 150. The frequency of the diameter's variation influences the wake patterns. When the variation frequency is negligible compared to the vortex shedding frequency, the wake is a quasi-steady representation of fixed cylinder shedding. We will discuss wake pattern bifurcations that occur as the variation frequency becomes comparable to the vortex shedding frequency. Comparisons will be made with the wake patterns generated by a constant-diameter circular cylinder forced to oscillate transverse to the free stream.

9:31AM A3.00008 Wake structure of an oscillating cylinder in a flowing soap film, MARK STREMLER, WENCHAO YANG, Virginia Tech — When a circular cylinder oscillates with respect to a uniform background flow, a variety of wake patterns can be observed in which multiple vortices are generated during each shedding cycle. Thorough investigations of the possible wake patterns behind a cylinder undergoing forced oscillations have been conducted by C.H.K. Williamson using two-dimensional characterization of a three-dimensional flow. Attempts to reproduce the structural bifurcations using two-dimensional computational models have been only moderately successful. A flowing soap film, an experimental system with quasi-two-dimensional flow, provides an alternative method for investigating the role of system dimensionality in the structure and dynamics of complex vortex wakes. Wake patterns are observed directly through interference fringes caused by thickness variations in the soap film. Such systems have been used for decades to visualize wake structure, but they have not previously been used to conduct an analog of Williamson's work. We will discuss the results of an ongoing parametric study of the wake structure produced by a circular cylinder undergoing forced oscillations transverse to the background flow in an inclined soap film system.

9:44AM A3.00009 Investigation of Vortical Flow Patterns in the Near Field of a Dynamic Low-Aspect-Ratio Cylinder¹, SAMANTHA GILDERSLEEVE, MICHAEL AMITAY, Rensselaer Polytech Inst — The flowfield and associated flow structures of a low-aspect-ratio cylindrical pin were investigated experimentally in the near-field as the pin underwent wall-normal periodic oscillations. Under dynamic conditions, the pin is driven at the natural wake shedding frequency with an amplitude of 33% of its mean height. Additionally, a static pin was also tested at various mean heights of 0.5, 1.0, and 1.5 times the local boundary layer thickness to explore the effect of the mean height on the flowfield. Three-dimensional flowfields were reconstructed and analyzed from SPIV measurements where data were collected along streamwise planes for several spanwise locations under static and dynamic conditions. The study focuses on the incoming boundary layer as it interacts with the pin, as well as two main vortical formations: the arch-type vortex and the horseshoe vortex. Under dynamic conditions, the upstream boundary layer is thinner, relative to the baseline, and the downwash in the wake increases, resulting in a reduced wake deficit. These results indicate enhanced strength of the aforementioned vortical flow patterns under dynamic conditions. The flow structures in the near-field of the static/dynamic cylinder will be discussed in further detail.

¹Supported by The Boeing Company

Sunday, November 20, 2016 8:00AM - 9:44AM – Session A4 Acoustics I: Aeroacoustics (General) B112 - Philip McCarthy, University of Toronto

8:00AM A4.00001 Acoustic mode coupling of two facing, shallow cylindrical cavities¹, PHILIP MCCARTHY, ALIS EKMEKCI, University of Toronto — Cavity mode excitation by grazing flows is a well-documented source for noise generation. Similarly to their rectangular equivalents, single cylindrical cavities have been shown to exhibit velocity dependent self-sustaining feedback mechanisms that produce significant tonal noise. The present work investigates the effect of cavity mode coupling on the tonal noise generation for two facing, shallow cylindrical cavities. This geometric arrangement may occur for constrained flows, such as those within ducts, silencers or between aircraft landing gear wheels. For the latter configuration, the present study has observed that the tonal frequency dependence upon the freestream Mach number, associated with the single cavity feedback mechanism, no longer holds true. Instead, two simultaneously present and distinct large amplitude tones that are independent (in frequency) of speed, propagate to the far field. These two, fixed frequency tones are attributable to the first order transverse mode, and the first order transverse and azimuthal modes for the two combined cavities and the volume between them. Altering either the cavity aspect ratio or the inter-cavity spacing thus changes the acoustic resonant volume and translates the centre frequencies of the observed tones correspondingly.

¹The authors would like to thank Bombardier and Messier-Bugatti-Dowty for their continued support.

8:13AM A4.00002 ABSTRACT WITHDRAWN –

8:26AM A4.00003 Direct numerical simulation of broadband trailing edge noise from a NACA 0012 airfoil¹, MOHAMMAD MEHRABADI, DANIEL BODONY, University of Illinois at Urbana-Champaign — Commercial jet-powered aircraft produce unwanted noise at takeoff and landing when they are close to near-airport communities. Modern high-bypass-ratio turbofan engines have reduced jet exhaust noise sufficiently such that noise from the main fan is now significant. In preparation for a large-eddy simulation of the NASA/GE Source Diagnostic Test Fan, we study the broadband noise due to the turbulent flow on a NACA 0012 airfoil at zero degree angle-of-attack, a chord-based Reynolds number of 408,000 and a Mach number of 0.115 using direct numerical simulation (DNS) and wall-modeled large-eddy simulation (WMLES). The flow conditions correspond to existing experimental data. We investigate the roughness-induced transition-to-turbulence and sound generation from a DNS perspective as well as examine how these two features are captured by a wall model. Comparisons between the DNS- and WMLES-predicted noise are made and provide guidance on the use of WMLES for broadband fan noise prediction.

¹AeroAcoustics Research Consortium

8:39AM A4.00004 Large Eddy Simulation of Surface Pressure Fluctuations on a Stalled Airfoil, SANJIVA LELE, Stanford University, JOSEPH KOCHEEMOOLAYIL, STC Corp. in NASA Ames Research Center — The surface pressure fluctuations beneath the separated flow over a turbine blade are believed to be responsible for a phenomenon known as Other Amplitude Modulation (OAM) of wind turbine noise. Developing the capability to predict stall noise from first-principles is a pacing item within the context of critically evaluating this conjecture. We summarize the progress made towards using large eddy simulations to predict stall noise. Successful prediction of pressure fluctuations on the airfoil surface beneath the suction side boundary layer is demonstrated in the near-stall and post-stall regimes. Previously unavailable two-point statistics necessary for characterizing the surface pressure fluctuations more completely are documented. The simulation results indicate that the space-time characteristics of pressure fluctuations on the airfoil surface change drastically in the near-stall and post-stall regimes. The changes are not simple enough to be accounted for by straight-forward scaling laws. The eddies responsible for surface pressure fluctuations and hence far-field noise are significantly more coherent across the span of the airfoil in the post-stall regime relative to the more canonical attached configurations.

8:52AM A4.00005 On the intermittent route to resonance for flow through an orifice in a duct, MANU KAMIN¹, JOSEPH MATHEW², Indian Inst of Science — In experiments done by Vineeth et. al. (2015) on flow in a duct-orifice arrangement, flow enters through the duct inlet, and leaves into the atmosphere through the orifice exit, whistling was observed at a Reynolds number of 4200, where large amplitude pressure oscillations were observed. At slightly lower Reynolds numbers, bursts of smaller amplitudes of pressure oscillations were observed to appear intermittently. For a similar configuration, LES were carried out. Both whistling and intermittency were observed in the simulations. As air flows from the duct into the orifice, it turns sharply around the corner at the duct-orifice interface, and hence, flow separation occurs, and a shear layer is formed at the mouth of the orifice. The mechanism of whistling was found to be the shear layer within the orifice flapping about and hitting the trailing edge of the orifice periodically, thus causing the shear layer to break and roll up into a vortex. Hurst exponent was measured in the time series data obtained. It was found to gradually drop to zero as the flow approached the state of whistling, since the growth rates of all the long term and short term trends in the time series vanish. A loss of multifractality in the time series was also observed as flow approached whistling.

¹Research Scholar (MSc. Engg.), Aerospace Engineering Dept., Indian Institute of Science

²Professor, Aerospace Engineering Dept., Indian Institute of Science

9:05AM A4.00006 Computation and analysis of rotor-noise generation in grid turbulence¹, JUNYE WANG, KAN WANG, MENG WANG, University of Notre Dame — The noise of a ten-bladed rotor interacting with a grid-generated turbulent flow at low Mach number is computed using large-eddy simulation and the Ffowcs Williams-Hawkins extension to Lighthill's theory. The grid turbulence is approximated as convected homogeneous and isotropic turbulence generated by a separate simulation and provided as inflow boundary conditions. The sound pressure spectrum predicted by the simulation exhibits overall agreement with previous experimental measurements in terms of the spectral shape and level. The turbulence ingestion noise is broadband with small peaks at the blade passing frequency and its harmonics. It is significantly stronger than the rotor self-noise generated by blade trailing-edge vortex shedding. Consistent with experimental observations, decreasing the rotor advance ratio at fixed mean inflow velocity leads to an increase in the sound pressure level. Different levels of acoustic compactness assumptions are examined, and the results indicate that the blade chord is acoustically compact over the frequency range of interest. Blade to blade correlations of the acoustic dipole sources are shown to be small.

¹Supported by ONR Grant N00014-14-1-0129

9:18AM A4.00007 Large-eddy simulation of propeller noise¹, JACOB KELLER, KRISHNAN MAHESH, Univ of Minnesota - Twin Cities — We will discuss our ongoing work towards developing the capability to predict far field sound from the large-eddy simulation of propellers. A porous surface Ffowcs-Williams and Hawking (FW-H) acoustic analogy, with a dynamic endcapping method (Nitzkorski and Mahesh,2014) is developed for unstructured grids in a rotating frame of reference. The FW-H surface is generated automatically using Delaunay triangulation and is representative of the underlying volume mesh. The approach is validated for tonal trailing edge sound from a NACA 0012 airfoil. LES of flow around a propeller at design advance ratio is compared to experiment and good agreement is obtained. Results for the emitted far field sound will be discussed.
Nitzkorski and Mahesh, Phys. Fluids, 2014, 26(11):115101

¹This work is supported by ONR.

9:31AM A4.00008 Prediction of Flow-Induced Noise Over a Realistic Automotive Vehicle¹, JAEYONG JEONG, JUNSHIN PARK, DONGHYUN YOU, Pohang Univ of Sci & Tech — Turbulent flow interacting with the front parts of an automotive vehicle, such as the cowl-top, A-pillars, and side mirrors are known to be significant sources of acoustic noise. In the present study, sources and propagation of acoustic noise generated over the front parts of a realistic automotive vehicle, known as the DrivAer model are predicted using a novel hydrodynamics-acoustics splitting method. Large eddy simulations are conducted to predict the turbulent flow field which is employed to compute noise sources, while of which accuracy is validated against experimental data. Acoustic fields are predicted using immersed-boundary linearized perturbed compressible equations. Discussion on turbulent flow fields, acoustic sources, and acoustic wave propagation are presented.

¹Supported 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

Sunday, November 20, 2016 8:00AM - 9:57AM –

Session A5 DFD GPC: Geophysical Fluid Dynamics: Air-sea Interaction B113 - James Duncan, University of Maryland

8:00AM A5.00001 Air-sea exchange from bubble-induced jetting: How viscous forces suppress droplet production from small bubbles¹, ELENA FLYNN, PETER WALLS, JAMES BIRD, Boston Univ — When a bubble ruptures in the ocean, it frequently produces a jet that releases aerosols into the atmosphere. The number of jet drops ejected is important because droplets may contain sea salt and other cloud condensation nuclei. It is generally accepted that the smallest bubbles produce the largest number of jet drops. However, if the bubble is sufficiently small, viscosity prevents droplet production altogether. Here we investigate the number of jet drops produced by small bubbles. Using a combination of high-speed microscopy, similitude, and numerical simulations, we quantify the extent that viscous forces inhibit this droplet production.

¹We acknowledge support from NSF under Grant No. 1351466

8:13AM A5.00002 Numerical investigation of wind over two progressive waves, TAO CAO, University of Minnesota — Wind-wave interaction is important in many applications and critical for the fundamental understanding of the wind-wave growth mechanism. It has been found in present study that the growth of short wave is suppressed in the presence of long waves, which is called the sheltering effect. In the present study, we have carried out a direct numerical simulation (DNS) of wind over two surface waves to improve the understanding of the sheltering effect. We have observed that the sheltering effect on the short wave strongly depends on the wave age of the long wave, thus the wave growth rate of the long wave. For the slow and fast long waves, the magnitude of wave growth rate of the short wave is significantly reduced compared with the cases with short wave only. But for the intermediate long wave, this reduction is relatively small. Based on the DNS data, the budget of energy conservation in the wave boundary layer is analyzed in detail.

8:26AM A5.00003 Numerical study of airflow over breaking waves, ZIXUAN YANG, LIAN SHEN, Univ of Minnesota - Twin Cities — We present direct numerical simulation (DNS) results on airflow over breaking waves. Air and water are simulated as a coherent system. The initial condition for the simulation is a fully-developed turbulent airflow over strongly-forced steep waves. The airflow is driven by a shear stress at the top. The effects of the initial wave steepness and wave age are studied systematically. Because wave breaking is an unsteady process, we use ensemble averaging of a large number of runs to obtain turbulent statistics. Simulation results show that the airflow above does not see the wave trough during wave breaking. Vortex structures at different stages of wave breaking are analyzed based on a linear stochastic estimation method. It is found that the wave breaking alters the pattern of vortex structures.

8:39AM A5.00004 Observations of Equatorial Kelvin Waves and their Convective Coupling with the Atmosphere/Ocean Surface Layer¹, PATRICK CONRY, H. J. S. FERNANDO, LAURA LEO, BYRON BLOMQUIST, University of Notre Dame, VINCENT AMELIE, NELSON LALANDE, Seychelles Meteorological Authority, ED CREEGAN, CHRIS HOCUT, BEN MACCALL, YANSEN WANG, US Army Research Laboratory, S. U. P. JINADASA, National Aquatic Resources Research and Development Agency, CHIEN WANG, LIK-KHIAN YEO, Singapore-MIT Alliance for Research and Technology — Intraseasonal disturbances with their genesis in the equatorial Indian Ocean (IO) are an important component of global climate. The disturbances, which include Madden-Julian Oscillation and equatorial Kelvin and Rossby waves in the atmosphere and ocean, carry energy which affects El Niño, cyclogenesis, and monsoons. A recent field experiment in IO (ASIRI-RAWI) observed disturbances at three sites across IO with arrays of instruments probing from surface layer to lower stratosphere. During the field campaign the most pronounced planetary-scale disturbances were Kelvin waves in tropical tropopause layer. In Seychelles, quasi-biweekly westerly wind bursts were documented and linked to the Kelvin waves aloft, which breakdown in the upper troposphere due to internal shear instabilities. Convective coupling between waves' phase in upper troposphere and surface initiates rapid (turbulent) vertical transport and resultant wind bursts at surface. Such phenomena reveal linkages between planetary-scale waves and small-scale turbulence in the surface layer that can affect air-sea property exchanges and should be parameterized in atmosphere-ocean general circulation models.

¹Funded by ONR Grants N00014-14-1-0279 and N00014-13-1-0199

8:52AM A5.00005 A dynamic framework for subgrid-scale parametrization of mesoscale eddies in geophysical flows, OMER SAN, ROMIT MAULIK, Oklahoma State University - Stillwater — This study puts forth a modular dynamic subgrid-scale modeling framework for large eddy simulation of quasigeostrophic turbulence based upon minimizing the errors between structural and functional subgrid-scale models. The approximate deconvolution procedure (AD) is used to estimate the free modeling parameters for the eddy viscosity coefficient parameterized in space and time using the Smagorinsky and Leith models. The novel idea here is to estimate the modeling parameters using the AD method rather than a test filter. The proposed model is applied to a wind-driven quasigeostrophic four-gyre ocean circulation problem, which is a standard prototype of more realistic ocean dynamics. Results show that the proposed model captures the quasi-stationary ocean dynamics and provides the time averaged four-gyre circulation patterns. Taking into account for local resolved flow characteristics, the model dynamically provides higher eddy viscosity values for lower resolutions. Furthermore, our first step in the numerical assessment for solving the quasigeostrophic turbulence problem addresses the intimate relationship between the eddy viscosity coefficients and the numerical resolution employed by the quasigeostrophic models.

9:05AM A5.00006 Dynamical properties of breaking waves: dissipation, air entrainment and spray generation, NICK PIZZO, LUC DEIKE, W. KENDALL MELVILLE, Scripps Institution of Oceanography, UC San Diego, STEPHANIE POPINET, Sorbonne Universites, UPMC, CNRS, Institut Jean Le Rond d'Alembert. — 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 parametrization for ocean-atmosphere exchange in weather and climate models. Here, we present 2D and 3D direct numerical simulations of breaking waves, compared with laboratory measurements. The dissipation due to breaking in the 2D and 3D simulations is found to be in good agreement with experimental observations and inertial-scaling arguments [1,2]. We discuss the transition from a 2D to a 3D flow during breaking. We present a model for air entrainment and bubble statistics that describes well the experimental and numerical data, and is based on turbulent fragmentation of the bubbles and a balance between buoyancy forces and viscous dissipation [2]. Finally we discuss the generation of large drops during the impact and splashing process. [1] Deike, L., Popinet, S., and Melville, W.K. 2015. Journal of Fluid Mechanics. vol 769, p541-569.[2] Deike, L., Melville, W.K., and Popinet, S. 2016. Journal of Fluid Mechanics. vol 801, p91- 129.

9:18AM A5.00007 A Laboratory Study of a Water Surface in Response to Rainfall

, REN LIU, XINAN LIU, JAMES DUNCAN, Department of Mechanical Engineering, University of Maryland, College Park — The shape of a water surface in response to the impact of raindrops is studied experimentally 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 hypodermic needles that are attached to the bottom of an open-surface water tank. The tank is connected to a 2D translation stage to provide a small-radius horizontal circular or oval motion to the needles, thus avoiding repeated drop impacts at the same location under each needle. The drop diameter is about 2.6 mm and the height of the water tank above the water surface of the pool is varied from 1 m to 4.8 m to provide different impact velocities. The water surface features including stalks, crowns and ring waves are measured with a cinematic laser-induced-fluorescence (LIF) technique. It is found that the average stalk height is strongly correlated to the impact velocities of raindrops and the phase speeds of ring waves inside the rain field are different from that measured outside the rain field.

9:31AM A5.00008 Instabilities in Non-Boussinesq Density Stratified Long and Narrow Lakes¹

, ANIRBAN GUHA, MIHIR SHETE, Indian Inst of Tech-Kanpur — We have discovered a new type of instability that can potentially occur in density stratified long and narrow lakes. The non-Boussinesq air-water interface plays a major role in this instability mechanism. A two layered lake driven by wind is considered; in such wind driven scenarios circulation sets up in each layer of the lake. The flow is assumed to be two dimensional, inviscid and incompressible. A surface gravity wave exists on the interface between air and water while an interfacial gravity wave exists on the interface between the two water layers (interface between warm and cold water). The resonant interactions between these two waves under a suitable doppler shift gives rise to normal mode growth rates leading to instability. We verify these claims analytically by piecewise linear velocity and density profiles. Furthermore we also use a realistic velocity and density profiles that are smooth and perform a linear stability analysis using a non-Boussinesq Taylor-Goldstein equation solver. We find that the normal mode instabilities are instigated by realistic wind velocities.

¹Planetary Science and Exploration (PLANEX) Programme Grant No. PLANEX/PHY/2015239

9:44AM A5.00009 Seasonal variability of atmospheric surface layer characteristics and weather pattern in Qatar¹

, DHRUBAJYOTI SAMANTA, WAY LEE CHENG, REZA SADR, Texas AM University at Qatar — Qatar's economy is based on oil and gas industry, which are mostly located in coastal regions. Therefore, better understanding of coastal weather, characteristics of surface layer and turbulence exchange processes is much needed. However, the turbulent atmospheric layer study in this region is severely limited. To support the broader aim and study long term precise wind information, a micro-meteorological field campaign has been carried out in a coastal location of north Qatar. The site is based on a 9 m tower, installed at Al Ghariya in the northern coast of Qatar, equipped with three sonic anemometers, temperature-humidity sensor, radiometer and a weather station. This study shows results based on the period August 2015 to July 2016. Various surface layer characteristics and modellings coefficients based on Monin Obukhov similarity theory is studied for the year and seasonal change is noted. Along with the seasonal variabilities of different weather parameters also observed. We hope this long term field observational study will be very much helpful for research community especially for modelers. In addition, two beach and shoreline monitoring cameras installed at the site could give first time information on waves and shoreline changes, and wind-wave interaction in Qatar.

¹An Preliminary Analysis of Wind-Wave Interaction in Qatar in the Context of Changing Climate

Sunday, November 20, 2016 8:00AM - 9:57AM –

Session A6 Aerodynamics: Control Strategies B114 - James Buchholz, The University of Iowa

8:00AM A6.00001 Manipulation of Leading-Edge Vortex Evolution by Applied Suction¹

, JAMES BUCHHOLZ, JAMES AKKALA, The University of Iowa — The generation and shedding of vortices from unsteady maneuvering bodies can be characterized within a framework of vorticity transport, accounting for the effects of multiple sources and sinks of vorticity on the overall circulation of the vortex system. On a maneuvering wing, the diffusive flux of secondary vorticity from the surface is a critical contributor to the strength and dynamics of the leading-edge vortex, suggesting that flow control strategies targeting the manipulation of the secondary vorticity flux and the secondary vortex may provide an effective means of manipulating vortex development. Suction has been applied in the vicinity of the secondary vortex during the downstroke of a periodically-plunging flat-plate airfoil, and the flow evolution and aerodynamic loads are compared to the baseline case in which suction is not applied. Observation of the resulting surface pressure distribution and flow evolution suggest that the secondary flux of vorticity and the evolution of the flow field can be altered subject to appropriate position of the suction ports relative to the developing vortex structures, and at a specific temporal window in the development of the vortex.

¹This work was supported by the Air Force Office of Scientific Research, grant number FA9550-16-1-0107 and NSF EPSCoR grant number EPS1101284.

8:13AM A6.00002 Performance of active and passive control of an airfoil using CFD¹

, DANIEL ASSELIN, JAY YOUNG, C.H.K. WILLIAMSON, Cornell University — Birds and fish employ flapping motions of their wings and fins in order to produce thrust and maneuver in flight and underwater. There is considerable interest in designing aerial and submersible systems that mimic these motions for the purposes of surveillance, environmental monitoring, and search and rescue, among other applications. Flapping motions are typically composed of combined pitch and heave and can provide good thrust and efficiency (Read, et al. 2003). In this study, we examine the performance of an airfoil actuated only in the heave direction. Using a cyber-physical fluid dynamics system (Mackowski & Williamson 2011, 2015, 2016), we simulate the presence of a torsion spring to enable the airfoil to undergo a passively controlled pitching motion. The addition of passive pitching combined with active heaving ("Active-Passive" or AP) provides significantly improved thrust and efficiency compared with heaving alone. In many cases, values of thrust and efficiency are comparable to or better than those obtained with two actively controlled degrees of freedom ("Active-Active" or AA). By using carefully-designed passive dynamics in the pitch direction, we can eliminate one of the two actuators, saving cost, complexity, and weight, while maintaining or improving performance.

¹This work was supported by the Air Force Office of Scientific Research Grant No. FA9550-15-1-0243, monitored by Dr. Douglas Smith.

8:26AM A6.00003 Feedback Control of an Ahmed Body Flow Exhibiting Symmetry-Breaking Regimes, OLGA EVSTAFYeva, AIMEE MORGANS, Imperial College London — At motorway speeds two-thirds of usable engine energy of square-back vehicles is spent overcoming the aerodynamic drag. The main source of drag is the bi-stable low pressure wake which forms at the back of the body as the boundary layers separate over the rear edges of the vehicle. Identifying large coherent structures and describing the physics of the wake is, therefore, of great practical importance for understanding the sources of drag and informing drag-reduction strategies. Present work investigates numerically the flow past the Ahmed body - a commonly used test-case for a simplified vehicle geometry, at Reynolds numbers $310 < Re_H < 435$. Previously reported experimental results on the bifurcation scenario for symmetry breaking of the Ahmed body wake (Grandemange et. al., 2012) are reproduced in Large Eddy Simulations and using data from the full 3D flow-field, the destabilising dynamics of the wake and vortex systems are investigated further. Dynamic Mode Decomposition is performed to identify the main coherent structures and their frequencies and growth rates. A practical feedback control strategy is then implemented to achieve base pressure recovery yielding a concomitant drag reduction.

8:39AM A6.00004 Identifying Sources of Lift Production on Rapidly Pitching Trailing Edge Flaps, PETER MANCINI, ANYA JONES, Univ of Maryland-College Park, MICHAEL OL, AFRL — Recent work has delved into the design and quantification of the aerodynamic response of large trailing edge flaps. Ultimately, these flaps would be used as a control mechanism to provide an immediate aerodynamic response to the vehicle, e.g. in the event of a gust encounter. The present work explores the individual sources and contributions of lift in the case of a large, rapidly pitching trailing edge flap. The flap is 50% of the chord length, and thus produces large acceleration and pitch rate terms that dominate the lift production. In the experiment and simulations presented here, the front element remains fixed at a constant angle of attack, while the rear element pitches to a final incidence angle, which in this study ranges from 5 degrees to 40 degrees. Although the front element does not pitch throughout the motion, it is important to consider the time history of the lift distribution on that wing section and assess whether the rapid pitching of the aft element affects the forces experienced on the stationary front element. These results are then used to suggest a simplified method for predicting lift production of a wing with a large trailing flap.

8:52AM A6.00005 In-Flight Active Wave Cancellation with Delayed-x-LMS Control Algorithm in a Laminar Boundary Layer¹, BERNHARD SIMON, Tech Univ Darmstadt, NICOLO FABBIANE, KTH Stockholm, TIMOTHEUS NEMITZ, Tech Univ Darmstadt, SHERVIN BAGHERI, DAN HENNINGSON, KTH Stockholm, SVEN GRUNDMANN, University of Rostock — This manuscript demonstrates the first successful application of the delayed-x-LMS (dxLMS) control algorithm for TS-wave cancellation. Active wave cancellation of two-dimensional broad-band Tollmien-Schlichting (TS) disturbances is performed with a single DBD plasma actuator. The experiments are conducted in flight on the pressure side of a laminar flow wing glove, mounted on a manned glider. The stability properties of the controller are investigated in detail with experimental flight data, DNS and stability analysis of the boundary layer. Finally, a model-free approach for dxLMS operation is introduced to operate the controller as a “black box” system, which automatically adjusts the controller settings based on a group speed measurement of the disturbance wave packets. The modified dxLMS controller is operated without a model and is able to adapt to varying conditions that may occur during flight in atmosphere.

¹DFG No.GR3524/4-1

9:05AM A6.00006 Three dimensional breakdown of an impulsively forced laminar separation bubble, THEODOROS MICHELIS, MARIOS KOTSONIS, Delft University of Technology — The spatio-temporal behaviour of a short laminar separation bubble is investigated experimentally. The bubble develops on a flat plate driven by an adverse pressure gradient wall at Reynolds number based on displacement thickness at separation of $Re_{\delta_s^*} = 975$. The boundary layer is impulsively forced by means of AC dielectric barrier discharge plasma actuator located upstream of the separation point. The full four-dimensional flow development is captured by time resolved tomographic PIV measurements using the multi-pass light amplification technique. Immediately after forcing, a convectively unstable wave packet emerges due to selective amplification of modes which interacts with the reattachment process. The interaction becomes non-linear at the reattachment region, where Λ structures typical of laminar separation bubbles are captured before the occurrence of breakdown. The structures and breakdown are characterised in terms of temporal evolution, spanwise coherence and energy budget. The diminishing of Λ structures triggers a sharp reduction in size of the separation bubble by interfering with the natural shedding process. As a result, the bubble significantly elongates without shedding undergoing bursting before recovering to its unperturbed state.

9:18AM A6.00007 Aerodynamic Impact of an Aft-Facing Slat-Step on High Re Airfoils, GEOFFREY KIBBLE, CHRIS PETRIN, JAMEY JACOB, BRIAN ELBING, Oklahoma State University, PETER IRELAND, BUDDY BLACK, Edge Aerodynamix — Typically, the initial aerodynamic design and subsequent testing and simulation of an aircraft wing assumes an ideal wing surface without imperfections. In reality, however the surface of an in-service aircraft wing rarely matches the surface characteristics of the test wings used during the conceptual design phase and certification process. This disconnect is usually deemed negligible or overlooked entirely. Specifically, many aircraft incorporate a leading edge slat; however, the mating between the slat and the top surface of the wing is not perfectly flush and creates a small aft-facing step behind the slat. In some cases, the slat can create a step as large as one millimeter tall, which is entirely submerged within the boundary layer. This abrupt change in geometry creates a span-wise vortex behind the step and in transonic flow causes a shock to form near the leading edge. This study investigates both experimentally and computationally the implications of an aft-facing slat-step on an aircraft wing and is compared to the ideal wing surface for subsonic and transonic flow conditions. The results of this study are useful for design of flow control modifications for aircraft currently in service and important for improving the next generation of aircraft wings.

9:31AM A6.00008 Control of a flexible, surface-piercing hydrofoil for high-speed, small-scale applications., GABRIEL BOUSQUET, MICHAEL TRIANTAFYLLOU, JEAN-JACQUES SLOTINE, Massachusetts Inst of Tech-MIT — In recent years, hydrofoils have become ubiquitous in the design of high performance surface vehicles such as sailboats. They have proven particularly useful at small scales: while the speed of displacement-hull sailboats of length L is limited by their hull speed $\propto \sqrt{gL}$, due to wave making resistance, such limitations do not apply to hydrofoil crafts and sailboats. Such crafts of length $O(1 - 10 \text{ m})$ are capable of reaching speeds in excess of 45 kts, often far faster than the wind. Besides, in the quest for super-maneuverability, actuated hydrofoils enable the efficient generation and control of large forces. With the intent to ultimately enable the design of small-scale, high-speed, and super-maneuverable surface vehicles, we investigate the problem of controlling the lift force generated by a flexible, surface-piercing hydrofoil traveling at high speed through a random wave field. We design a test platform composed of a rudder-like vertical foil, which is actuated in pitch, and instrumented with velocity, force, and immersion sensors. We present a feedback linearization controller, designed to operate over a wide range of velocities and sea states. Validation experiments are carried out on-the-field at speeds ranging from 3 to 10+m/s.

9:44AM A6.00009 Tomographic-PIV and TSP for Airfoil Flow-Control Studies , ADAM STOLT, JOHN OTTE, JESSTIN KRECH, JORDI ESTEVADEORDAL, North Dakota State University — Airfoil blades can experience a significant change of angle of attack during operation cycles that can lead to boundary layer separation and dynamic stall. It is unclear how elements distributed at the leading edge would affect the aerodynamic performance, boundary layer separation and transition, and stall behaviors. In the present study, various passive flow control structures, such as distributed dimples and bumps have been investigated and compared to airfoil geometries including the baseline smooth NACA0015 airfoil. Along with standard particle image velocimetry (PIV), a curved-laser sheet PIV, tomographic PIV, and Temperature Sensitive Paint (TSP) techniques have been combined to reveal spanwise flow information in the curved surface of the airfoil. Results show the effects and induced flow patterns of the various elements on boundary layer separation and stall at various angles of attack and compare them with the smooth models.

Sunday, November 20, 2016 8:00AM - 9:57AM –

Session A7 Separation Flow Around Obstacles B115 - Bharathram Ganapathisubramani, University of Southampton

8:00AM A7.00001 Effects of freestream turbulence on the characteristics of separation and re-attachment in flow past obstacles.¹ , JACQUES VAN DER KINDEERE, ROBERT HEARST, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton — We study the characteristics of separation and reattachment in the presence of freestream turbulence (FST) on flows on ribs. This two-dimensional obstacle represents a canonical geometry in industrial aerodynamics. It consists of a forward-facing step, FFS, followed by a backward-facing step, BFS. An experiment at Reynolds number 20000 based on rib height, H, was carried out. The rib was fully submerged in a boundary layer and the freestream was subjected to varying turbulence intensities: 0.5, 3.5, 7.5 and 9.0%. Three rib lengths of L/H=1, 4 and 8 were tested. Particle Image Velocimetry measurements show that increasing freestream turbulence consistently decreases recirculation lengths both on top and in the wake of ribs. The shrinkage of recirculation bubbles is also dependent on rib length because of the interaction between FFS and BFS. Snapshot Proper Orthogonal Decomposition suggests that FST modifies the dominant motions in the flow. The shape and ranking of POD modes of velocity above short ribs (L/H=1) seems unaffected by FST until the 4th mode. In contrast, the longer ribs (L/H=4 and 8) produce different dominant modes for each FST intensity. This signifies the dominant motions in the flow are affected by FST which could explain the different recirculation characteristics.

¹The authors acknowledge the financial support of the European Research Council (ERC Grant agreement No. 277472) and the Leverhulme Trust for the Philip Leverhulme Prize.

8:13AM A7.00002 Surface obstacles in pulsatile flow¹ , IAN A. CARR, MICHAEL W. PLESNIAK, George Washington University — Flows past obstacles mounted on flat surfaces have been widely studied due to their ubiquity in nature and engineering. For nearly all of these studies, the freestream flow over the obstacle was steady, i.e. constant velocity unidirectional flow. Unsteady, pulsatile flows occur frequently in biology, geophysics, biomedical engineering, etc. Our study is aimed at extending the comprehensive knowledge base that exists for steady flows to considerably more complex pulsatile flows. Beyond the important practical applications, characterizing the vortex and wake dynamics of flows around surface obstacles embedded in pulsatile flows can provide insights into the underlying physics in all wake and junction flows. In this study, we experimentally investigated the wake of four canonical surface obstacles: hemisphere, cube, and circular cylinders with aspect ratio of 1:1 and 2:1. Phase-averaged PIV and hot-wire anemometry are used to characterize the dynamics of coherent structures in the wake and at the windward junction of the obstacles. Complex physics occur during the deceleration phase of the pulsatile inflow. We propose a framework for understanding these physics based on self-induced vortex propagation, similar to the phenomena exhibited by vortex rings.

¹This material is based in part upon work supported by the National Science Foundation under Grant Number CBET-1236351, and GW Center Biomimetics and Bioinspired Engineering (COBRE).

8:26AM A7.00003 A different approach on the onset of separation in the flow around a circular cylinder. , NIKOLAOS MALAMATARIS, George Mason University / ATEI of Thessaloniki, I. SARRIS, TEI of Athens, D. PAZIS, Aristotle University of Thessaloniki, A. LIAKOS, US Naval Academy — The onset of separation in the flow around a cylinder is revisited with new insight. The goal of the research is to compute the smallest Reynolds number where the separation actual occurs rather than computing small eddies and extrapolating to the value of the Reynolds number where separation may occur. To this purpose, an accurate home made code is designed with Galerkin finite elements. The computational domain is chosen as the laboratory experiments by Taneda. It is found that in all six different choices of Taneda's diameters of the cylinders he used, separation is not observed for $Re < 6.1$. Actually, separation is computed in all of his six cases for $Re = 6.14$. Images of this smallest eddy are shown for the first time where all characteristics of eddies are recognisable (vortex centre, separation length etc). The vorticity of the flow is computed along the cylinder surface and it is shown that, at separation, vorticity changes sign. Byproducts of this research is the computation of the drag coefficient for Reynolds numbers starting from $1 \cdot 10^{-5}$ up to 40. In addition, the separation angle (point where vorticity changes sign) is computed for $6.14 \leq Re \leq 40$. This research aims to be the most thorough work done on that subject so far.

8:39AM A7.00004 Three-Dimensional, Laminar Flow Past a Short, Surface-Mounted Cylinder , ANASTASIOS LIAKOS, United States Naval Academy, NIKOLAOS MALAMATARIS, George Mason University/ATEI of Thessaloniki — The topology and evolution of three-dimensional flow past a cylinder of slenderness ratio $SR = 1$ mounted in a wind tunnel is examined for $0.1 \leq Re \leq 325$ (based on the diameter of the cylinder) where steady-state solutions have been obtained. Direct numerical simulations were computed using an in-house parallel finite element code. Results indicate that symmetry breaking occurs at $Re = 1$, while the first prominent structure is a horseshoe vortex downstream from the cylinder. At $Re = 150$, two foci are observed, indicating the formation of two tornadolike vortices downstream. Concurrently, another horseshoe vortex is formed upstream from the cylinder. For higher Reynolds numbers, the flow downstream is segmented to upper and lower parts, whereas the topology of the flow on the solid boundaries remains unaltered. Pressure distributions show that pressure, the key physical parameter in the flow, decreases everywhere except immediately upstream from the cylinder. In addition, creation of critical points from saddle-node-type bifurcations occur when the streamwise component of the pressure gradient changes sign. Finally, at $Re = 325$, an additional horseshoe vortex is formed at the wake of the cylinder

8:52AM A7.00005 Flow past a rotating cylinder at high Reynolds number using PANS method, RAJESH KUMAR, Department of Mechanical Engineering, NIT Kurukshetra — In the present study, high-Reynolds number flow past a rotating cylinder has been simulated using Partially-Averaged Navier-Stokes (PANS) method. The simulations are performed at $Re = 140000$. The spin ratio of the cylinder, which is defined by the ratio of the circumferential speed of the cylinder to the free-stream speed, varies from $a = 0$ to $a = 4$. The resolved and the modeled physical scales have been compared with the corresponding LES data for better understanding of the efficacy of the PANS method. The comparison of PANS results with the LES results showed good agreement. It has been recognized that the PANS simulation is able to produce fairly acceptable results using even a coarse-mesh. It is recognized that the time-averaged flow statistics obtained using PANS and URANS simulations are approximately same. However the vortex structure is much better captured by the PANS method. With the increase in the spin ratio, decrease in the time-averaged drag and increase in the time-averaged lift force acting on the cylinder have been observed. The vortices in far wake region are displaced and deformed but those in the vicinity of the cylinder are stretched at the bottom and accumulated over the top of the cylinder.

9:05AM A7.00006 Laminar flow separation subject to control by zero-net-mass-flux jet, OLAF MARXEN, Univ of Surrey, RAJAT MITTAL, TAMER ZAKI, Johns Hopkins University — The flow around slender bodies at moderate Reynolds numbers often features a laminar separation bubble. Convective amplification of small-amplitude perturbations leads to the formation of two-dimensional large-scale vortices that are shed from the bubble. These perturbations can be triggered through a zero-net-mass-flux actuator in order to control the bubble size and shedding frequency. Using data from Navier-Stokes simulations for the flow around a canonical airfoil-like geometry, it is found that linear modes with intermediate frequencies exhibit strongest convective amplification caused by Kelvin-Helmholtz instability. Forcing at these frequencies is most effective. For low frequencies, the front part of the bubble still diminishes due to the interaction of a vortex that starts from the actuator with the wall. This vortex transiently amplifies downstream due to the Orr mechanism. Actuation at high frequencies leads to visible, amplified instability waves in the shear layer, but is not effective in reducing the size of the bubble.

9:18AM A7.00007 Experimental study on the flow around in-line array of spheres¹, DAEHYEON CHOI, HYUNGMIN PARK, Seoul National University — In this study, we investigate the flow around a in-line array of spheres focusing on the interactions between the wakes flows. In a circulating water tunnel, 12.7 mm-diameter spheres have been aligned in line with the direction of flow, with each sphere held by a 0.1 mm thin stainless-steel wire. Considered Reynolds number for a single sphere is 1000 and the number of spheres is increased up to five with varying the distance between them, as well. To measure the flow field, we use dye visualization and PIV together, and the drag forces of each sphere are indirectly measured using two-dimensional optical micrometer. As the center-to-center distance increases, the wake instability in the gap between them is enhanced, and the axisymmetric structure of wake collapses and the turbulence levels becomes large. Based on this observation, flow structure around the sphere array is classified depending on the symmetricity, steadiness and turbulent intensity between spheres and the wake behind a following sphere. The drag on each sphere will be analyzed on the basis of this classification.

¹Supported by a BK21 Plus program funded by the Ministry of Education, Korean Government.

9:31AM A7.00008 Experimental and modeling study of the flow over a skewed bump¹, DAVID S. CHING, CHRISTOPHER J. ELKINS, JOHN K. EATON, Stanford Univ — Three-dimensional separated flows can be very sensitive to geometry and inlet conditions, such that a small change in the geometry or the upstream boundary layer could cause the flow structure to change drastically. This study examines the geometric sensitivity of a skewed bump with axis ratio 4/3 by changing the angle of the bump with respect to the flow. The three-dimensional, three-component mean velocity field was acquired using Magnetic Resonance Velocimetry (MRV) for several bump angles. The flow is dominated by large coherent vortices in the wake. For a symmetric case, two counter-rotating vortices exist in the wake, but when the bump is skewed relative to the oncoming flow one vortex structure is much stronger and overwhelms the other vortex. A comparison to RANS simulations found that the RANS simulations predict the velocity fields with reasonable accuracy within the separation bubble, but are very inaccurate downstream of reattachment. Using a time-resolved MRV sequence, the shedding frequency of the wake was determined for two bump angles. Hot-wire anemometry confirmed the shedding frequencies found from the MRV data and observed that the shedding frequency is sensitive to the bump angle at low bump angles, but is insensitive at high bump angles.

¹Funding provided by the Office of Naval Research

9:44AM A7.00009 Optimization of air injection parameters toward optimum fuel saving effect for ships, INWON LEE, SEONG HYEON PARK, Pusan Natl Univ — Air lubrication method is the most promising commercial strategy for the frictional drag reduction of ocean going vessels. Air bubbles are injected through the array of holes or the slots installed onto the flat bottom surface of vessel and a sufficient supply of air is required to ensure the formation of stable air layer by the coalescence of the bubbles. The air layer drag reduction becomes economically meaningful when the power gain through the drag reduction exceeds the pumping power consumption. In this study, a model ship of 50k medium range tanker is employed to investigate air lubrication method. The experiments were conducted in the 100m long towing tank facility at the Pusan National University. To create the effective air lubrication with lower air flow rate, various configurations including the layout of injection holes, employment of side fences and static trim have been tested. In the preliminary series of model tests, the maximum 18.13%(at 15kts) of reduction of model resistance was achieved. This research was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MEST) through GCRC-SOP (Grant No. 2011-0030013).

Sunday, November 20, 2016 8:00AM - 9:57AM –
Session A8 Nonlinear Dynamics: Coherent Structures B116 - Tom Solomon, Bucknell University

8:00AM A8.00001 Streamwise asymptotics of spatially localized solutions in plane Poiseuille flow, ROMAN GRIGORIEV, JOSHUA BARNETT, Georgia Institute of Technology — Numerical advances of recent years have enabled us to find localized solutions in various canonical shear flows, for instance, relative periodic orbits in plane and pipe Poiseuille flow. These solutions have interesting properties such as the exponential decay of the leading and trailing fronts which appear to also shape the fronts of "turbulent puffs" that are found during intermittent turbulence. While arguably quite important, these exponential asymptotics are not well understood theoretically. This talk will discuss how they can be derived, or at least constrained, analytically using wave propagation theory.

8:13AM A8.00002 Hyperbolic neighborhoods as organizers of finite-time exponential stretching¹, SANJEEVA BALASURIYA, Univ of Adelaide, NICHOLAS OUELLETTE, Stanford University — Hyperbolic points and their unsteady generalization, hyperbolic trajectories, drive the exponential stretching that is the hallmark of nonlinear and chaotic flow. Typical experimental and observational velocity data is unsteady and available only over a finite time interval, and in such situations hyperbolic trajectories will move around in the flow, and may lose their hyperbolicity at times. Here we introduce a way to determine their region of influence, which we term a hyperbolic neighborhood, which marks fluid elements whose dynamics are instantaneously dominated by the hyperbolic trajectory. We establish, using both theoretical arguments and numerical verification from model and experimental data, that the hyperbolic neighborhoods profoundly impact Lagrangian stretching experienced by fluid elements. In particular, we show that fluid elements traversing a flow experience exponential boosts in stretching while within these time-varying regions, that greater residence time within hyperbolic neighborhoods is directly correlated to larger Finite-Time Lyapunov Exponent (FTLE) values, and that FTLE diagnostics are reliable only when the hyperbolic neighborhoods have a geometrical structure which is regular in a specific sense.

¹Future Fellowship Grant FT130100484 from the Australian Research Council (SB), and a Terman Faculty Fellowship from Stanford University (NO)

8:26AM A8.00003 Simple computation of null-geodesics, with applications to vortex boundary detection, MATTIA SERRA, GEORGE HALLER, Institute for Mechanical Systems, ETH Zurich — Recent results show that boundaries of coherent vortices (elliptic coherent structures) can be computed as closed null-geodesics of appropriate Lorentzian metrics defined on the physical domain of the underlying fluid. Here we derive a new method for computing null-geodesics of general Lorentzian metrics, founded on the geometry of geodesic flows. We also derive the correct set of initial conditions for the computation of closed null-geodesics, based on simple topological properties of planar closed curves. This makes the computation of coherent vortex boundaries fully automated, simpler and more accurate compared to the existing procedure. As an illustration, we compute objective coherent vortex boundaries in Oceanic and Atmospheric Flows.

8:39AM A8.00004 Coherent structure coloring: identification of coherent structures from sparse flow trajectories using graph theory¹, KRISTY SCHLUETER, JOHN DABIRI, Stanford University — Coherent structure identification is important in many fluid dynamics applications, including transport phenomena in ocean flows and mixing and diffusion in turbulence. However, many of the techniques currently available for measuring such flows, including ocean drifter datasets and particle tracking velocimetry, only result in sparse velocity data. This is often insufficient for the use of current coherent structure detection algorithms based on analysis of the deformation gradient. Here, we present a frame-invariant method for detecting coherent structures from Lagrangian flow trajectories that can be sparse in number. The method, based on principles used in graph coloring algorithms, examines a measure of the kinematic dissimilarity of all pairs of flow trajectories, either measured experimentally, e.g. using particle tracking velocimetry; or numerically, by advecting fluid particles in the Eulerian velocity field. Coherence is assigned to groups of particles whose kinematics remain similar throughout the time interval for which trajectory data is available, regardless of their physical proximity to one another. Through the use of several analytical and experimental validation cases, this algorithm is shown to robustly detect coherent structures using significantly less flow data than is required by existing methods.

¹This research was supported by the Department of Defense (DoD) through the National Defense Science & Engineering Graduate Fellowship (NDSEG) Program.

8:52AM A8.00005 Dynamically dominant exact coherent structures in turbulent Taylor-Couette flow¹, MICHAEL KRYGIER, ROMAN GRIGORIEV, Georgia Institute of Technology — Unstable Exact Coherent Structures (ECS), which are solutions to the Navier-Stokes equation, provide a connection between turbulence and dynamical systems and offer a method for exploiting the low dimensionality of weakly turbulent flows. We investigate ECS in an intermittent Taylor-Couette flow (TCF) found in a small-aspect-ratio geometry with counter-rotating cylinders ($\eta = 0.5$, $\Gamma = 1$, $Re_i = -1200$, $Re_o = 1200$). The presence of end-caps breaks the axial translational symmetry of TCF, but continuous rotational symmetry remains, which suggest that typical ECS should be the relative versions of equilibria and time-periodic orbits. Indeed, previous studies (Meseguer et al., 2009 and Deguchi, Meseguer & Mellibovsky, 2014) found several unstable traveling wave solutions (relative equilibria). We have shown that the dynamically dominant ECS for weakly turbulent TCF in the small-aspect-ratio geometry are relative periodic orbits (not relative equilibria), as evidenced by the frequent visits of their neighborhoods by the turbulent flow.

¹This work is supported by a grant from the Army Research Office (Contract W911NF-15-1-0471).

9:05AM A8.00006 Low-order invariant solutions in plane Couette flow¹, MUHAMMAD AHMED, ATI SHARMA, Univ of Southampton — Ten new equilibrium solutions of the Navier-Stokes equations in plane Couette flow are presented. The new solutions add to the inventory of known equilibria in plane Couette flow found by Nagata JFM 1990, Gibson JFM 2008, 2009, and Halcrow JFM 2008, who together found 13. These new solutions elucidate the low-dimensional nature of exact coherent structures, which are essential to defining simplified mechanisms that explain the self-sustaining nature of wall-bounded flows. In particular, one of the solutions found has a one-dimensional unstable manifold in the symmetry-invariant subspace and otherwise, like the lower branch equilibrium solution found by Nagata JFM 1990. A new method for generating initial guesses for Newton-Krylov-hookstep (NKH) searches is also presented. This method allows the NKH algorithm to find equilibrium solutions that are derived from previous solutions.

¹Air Force Office of Scientific Research (European Office of Aerospace Research and Development) under award FA9550-14-1-0042

9:18AM A8.00007 Burning invariant manifolds and reaction front barriers in three-dimensional vortex flows¹, JJ SIMONS, MINH DOAN, Bucknell University, KEVIN MITCHELL, UC-Merced, TOM SOLOMON, Bucknell University — We describe experiments on the effects of three-dimensional fluid advection on the motion of the excitable, Ruthenium-catalyzed Belousov-Zhabotinsky chemical reaction. The flow is a superposition of horizontal and vertical vortices produced by magnetohydrodynamic forcing and measured with particle image velocimetry. We visualize the propagating fronts in three dimensions with a scanning, laser-induced fluorescence technique that benefits from the fluorescence of the reduced Ru indicator. The experiments reveal a combination of tube- and sheet-like barriers that block the propagating reaction fronts. We study the dependence of the structure of these barriers on the front propagation speed (normalized by a characteristic flow velocity). The locations and blocking properties of these barriers are interpreted with a six-dimensional *burning invariant manifold*² theory that follows the evolution of front elements in the flow.

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

²J. Mahoney, D. Bargteil, M. Kingsbury, K. Mitchell and T. Solomon, *Europhys. Lett.* **98**, 44005 (2012).

9:31AM A8.00008 Equilibria and Travelling wave solutions for Couette and channel flows with longitudinal grooves , SABARISH VADAREVU, ATI SHARMA, BHARATHRAM GANAPATHISUBRAMANI,

University of Southampton — Several classes of exact solutions for canonical flows have been computed by earlier researchers. These solutions are known to inform the flow of turbulence in state-space. We extend two classes of exact solutions, equilibria and travelling wave solutions, from flat-walled Couette and channel flows to grooved geometries with groove-amplitudes as high as 20% of channel half-height. These solutions provide insight into the mechanics of how a wavy wall could influence turbulent flow. Plotting scalars such as the average shear stress at the wall and the bulk velocity (for channel flows) allows us to identify branches of solutions that could have greater contributions to turbulence, and reconcile the curious phenomenon of drag reduction observed in some riblet-mounted boundary layer flows. Earlier researchers have proposed using modified boundary conditions (imposed on flat surfaces) as a substitute to imposing the traditional no-slip and impermeability conditions on a rough wall. We compare solutions for grooved flows to those for flat-walled flows with modified boundary conditions to evaluate the validity of such simplification to non-laminar solutions.

9:44AM A8.00009 The orientation field of fibers advected by a two-dimensional chaotic flow , BARDIA HEJAZI, Wesleyan University, BERNHARD MEHLIG, University of Gothenburg, GREG VOTH, Wesleyan University

— We examine the orientation of slender fibers advected by a 2D chaotic flow. The orientation field of these fibers show fascinating structures called scar lines, where they rotate by π over short distances. We use the standard map as a convenient model to represent a time-periodic 2D incompressible fluid flow. To understand the fiber orientation field, we consider the stretching field, given by the eigenvalues and eigenvectors of the Cauchy-Green strain tensors. The eigenvector field is strongly aligned with the fibers over almost the entire field, but develops topological singularities at certain points which do not exist in the advected fiber field. The singularities are points that have experienced zero stretching, and the number of such points increases rapidly with time. A key feature of both the fiber orientation and the eigenvector field are the scar lines. We show that certain scar lines form from fluid elements that are initially stretched in one direction and then stretched in an orthogonal direction to cancel the initial stretching. The scar lines that satisfy this condition contain the singularities of the eigenvector field. These scar lines highlight the major differences between the passive director field and the much more widely studied passive scalar field.

Sunday, November 20, 2016 8:00AM - 9:57AM –

Session A9 General Fluid Dynamics: Rotating Flows B117 - Wei Zhang, Cleveland State University

8:00AM A9.00001 Giant Hydrodynamic Fluctuations Due to Coriolis Redistribution of Turbulent Kinetic Energy , CHARLES PETTY, ANDRE BENARD, Michigan State University

— The influence of chemical reaction on turbulent mixing of a chemically reactive constituent in a rotating channel flow indicates that the transverse transport of the reactive constituent is mitigated by a coupling between the shear component of the Reynolds stress and the longitudinal component of the mean flux of the reactive constituent. In the region of zero intrinsic vorticity, the dispersion coefficient in the cross flow direction is significantly larger than the dispersion coefficient in the spanwise direction. The dispersion coefficient in the longitudinal direction is relatively small. Koppula, K.S., A. Bénard, and C. A. Petty, 2011, "Turbulent Energy Redistribution in Spanwise Rotating Channel Flows", Ind. Eng. Chem. Res., 50 (15), 8905-8916.

8:13AM A9.00002 Suppressing Taylor vortices in a Taylor–Couette flow system with free surface , A. BOUABDALLAH, Université des Sciences et de la Technologie Houari Boumediene, Algiers, Algeria, H. OUALLI, M.

MEKADEM, École Militaire Polytechnique, Algiers, Algeria, M. GAD-EL-HAK, Virginia Commonwealth University, Richmond, Virginia, USA — Taylor–Couette flows have been extensively investigated due to their many industrial applications, such as catalytic reactors, electrochemistry, photochemistry, biochemistry, and polymerization. Mass transfer applications include extraction, tangential filtration, crystallization, and dialysis. A 3D study is carried out to simulate a Taylor–Couette flow with a rotating and pulsating inner cylinder. We utilize FLUENT to simulate the incompressible flow with a free surface. The study reveals that flow structuring is initiated with the development of an Ekman vortex at low Taylor number, $Ta = 0.01$. For all encountered flow regimes, the Taylor vortices are systematically inhibited by the pulsatile motion of the inner cylinder. A spectral analysis shows that this pulsatile motion causes a rapid decay of the free surface oscillations, from a periodic wavy movement to a chaotic one, then to a fully turbulent motion. This degenerative free surface behavior is interpreted as the underlying mechanism responsible for the inhibition of the Taylor vortices.

8:26AM A9.00003 A slowly rotating impeller in a rapidly rotating fluid. , NATHANIEL MACHICOANE, FREDERIC MOISY, PIERRE-PHILIPPE CORRET, Université Paris-Saclay, INSTABILITY, WAVES AND TURBULENCE TEAM

— We characterize the two-dimensionalization process in the turbulent flow produced by an impeller rotating at a rate ω in a fluid rotating at a rate Ω around the same axis for Rossby number $Ro = \omega / \Omega$ down to 0.01. The flow can be described as the superposition of a large-scale vertically invariant global rotation and small-scale shear layers detached from the impeller blades. As Ro decreases, the large-scale flow is subjected to azimuthal modulations. In this regime, the shear layers can be described in terms of wakes of inertial waves traveling with the blades, originating from the velocity difference between the non-axisymmetric large-scale flow and the blade rotation. The wakes are well defined and stable at low Rossby number, but they become disordered and interact nonlinearly at Ro of order of 1.

8:39AM A9.00004 The generalized Onsager model and DSMC simulations of high-speed rotating flows with product and waste baffles , DR. SAHADEV PRADHAN, Department of Chemical Engineering, Indian Institute of Science, Bangalore- 560 012, India

— The generalized Onsager model for the radial boundary layer and of the generalized Carrier-Maslen model for the axial boundary layer in a high-speed rotating cylinder ((S. Pradhan & V. Kumaran, J. Fluid Mech., 2011, vol. 686, pp. 109-159); (V. Kumaran & S. Pradhan, J. Fluid Mech., 2014, vol. 753, pp. 307-359)), are extended to a multiply connected domain, created by the product and waste baffles. For a single component gas, the analytical solutions are obtained for the sixth-order generalized Onsager equations for the master potential, and for the fourth-order generalized Carrier-Maslen equation for the velocity potential. In both cases, the equations are linearized in the perturbation to the base flow, which is a solid-body rotation. An explicit expression for the baffle stream function is obtained using the boundary layer solutions. These solutions are compared with direct simulation Monte Carlo (DSMC) simulations and found excellent agreement between the analysis and simulations, to within 15%, provided the wall-slip in both the flow velocity and temperature are incorporated in the analytical solutions.

8:52AM A9.00005 Turbulent strength in ultimate Taylor-Couette turbulence¹ , RODRIGO

EZETA, University of Twente, SANDER G. HUISMAN, Univ Lyon, Ens de Lyon, Univ Claude Bernard, CNRS, Laboratoire de Physique, F-69342 Lyon, France, CHAO SUN, Tsinghua University, DETLEF LOHSE, University of Twente — We provide the local scaling of the Taylor-Reynolds number (Re_λ) as a function of driving strength (Ta), in the ultimate regime of Taylor-Couette flow for the inner cylinder rotation case. The calculation is done via local flow measurements using Particle Image Velocimetry (PIV) to reconstruct the velocity fields. We approximate the value of the local dissipation rate $\epsilon(r)$ using the scaling for the second order structure functions in the longitudinal and transversal directions within the inertial regime where Taylor's hypothesis is not invoked. We find an effective local scaling of $\langle \epsilon(r) \rangle_r / (\nu^3 d^{-4}) \sim Ta^{1.4}$, which is the same as the global dissipation rate obtained from both torque measurements and Direct Numerical Simulations (DNS). Additionally, we calculate the Kolmogorov length scale and find $\langle \eta(r) \rangle_r / d \sim Ta^{-0.35}$. The turbulence intensity is also calculated and it is found to scale with the driving strength as $i_\theta \sim Ta^{-0.056}$. Finally, with both the local dissipation rate and the local fluctuations available we find that the Taylor-Reynolds number scales as $Re_\lambda \sim Ta^{0.18}$.

¹Stichting voor Fundamenteel Onderzoek der Materie (FOM)

9:05AM A9.00006 Critical-band vortex in a precessing sphere¹ , SHIGEO KIDA, Doshisha University

— We consider the motion of an incompressible viscous fluid in a rotating sphere with strong precession, where the spin and precession axes are assumed to be orthogonal to each other. By an asymptotic analysis we determine the structure of the steady flow in the entire sphere in the leading order of the asymptotic expansion. It is found that the boundary layer is developed on the whole spherical surface, outside of which the flow is stationary in the leading order. The boundary-layer approximation breaks down on a great circle perpendicular to the precession axis. A partial differential equation which describes the velocity field in the vicinity of this great circle, called the critical band, is derived theoretically and solved numerically to find a pair of vortices localized in the critical band. In the meeting we present the three-dimensional structure of these vortices with streamlines as well as streamsurfaces.

¹Numerical computation in this work was carried out at the Yukawa Institute Computer Facility.

9:18AM A9.00007 Rotational Flow of Nonlinear Drilling Mud , NARIMAN ASHRAFI, MEHDI

YEKTAPUR, Young Researchers and Elites Club, Science and Research Branch, Islamic Azad University, Tehran, Iran — To analyze the drilling process, the pseudoplastic flow between coaxial cylinders is investigated. Here, the inner cylinder is assumed to rotate and, at the same time, slide along its axis. A numerical scheme based on the spectral method is used to derive a low-order dynamical system from the conservation of mass and momentum equations under mixed boundary conditions. It is found that the Azimuthal stress develops far greater than other stress components. All stress components increase as pseudoplasticity is decreased. The flow loses its stability to the vortex structure at a critical Taylor number. The emergence of the vortices corresponds to the onset of a supercritical bifurcation. The Taylor vortices, in turn, lose their stability as the Taylor number reaches a second critical number corresponding to the onset of a Hopf bifurcation. The rotational and axial velocities corresponding to the optimum drilling conditions are evaluated. Furthermore, complete stress and viscosity maps are presented for different scenarios in the flow regime.

9:31AM A9.00008 Oscillatory Convection in Rotating Liquid Metals¹ , VINCENT BERTIN²,

ENS Paris, ALEX GRANNAN, JONATHAN AURNOU, UCLA Earth and Space Sciences — We have performed laboratory experiments in a aspect ratio $\Gamma \simeq 2$ cylinder using liquid gallium ($Pr \simeq 0.023$) as the working fluid. The Ekman number varies from $E = 4 \times 10^{-5}$ to 4×10^{-6} and the Rayleigh number varies from $Ra = 3 \times 10^5$ to 2×10^7 . Using heat transfer and temperature measurements within the fluid, we characterize the different styles of low Pr rotating convective flow. The convection threshold is first overcome in the form of a container scale inertial oscillatory mode. At stronger forcing, wall-localized modes develop, coexisting with the inertial oscillatory modes in the bulk. When the strength of the buoyancy increases further, the bulk flow becomes turbulent while the wall modes remain. Our results imply that rotating convective flows in liquid metals do not develop in the form of quasi-steady columns, as in $Pr \simeq 1$ planetary and stellar dynamo models, but in the form of oscillatory motions. Therefore, convection driven dynamo action in low Pr fluids can differ substantively than that occurring in typical $Pr \simeq 1$ numerical models. Our results also suggest that low wavenumber, wall modes may be dynamically and observationally important in liquid metal dynamo systems.

¹We thank the NSF Geophysics Program for support of this project.

²VB will be in France during DFD, so one of the other authors will present this paper.

9:44AM A9.00009 Linear stability analysis of natural convection in an inclined rotating cavity.¹ , DIANA PEREZ-ESPEJEL, RUBEN AVILA, Universidad Nacional Autonoma de Mexico

— The linear stability analysis of the natural convection in an inclined rectangular cavity with rotation is presented. The critical parameters for the onset of longitudinal rolls are obtained by solving the stability equations with the Tau-Chebyshev spectral method. We report under what conditions of the inclination angle, Rayleigh and Taylor numbers, the onset of longitudinal rolls appears. The rectangular cavity with a small aspect ratio (depth/length) is heated from below, cooled from above and thermally isolated at the rest of the boundaries. The rotation axis is orthogonal to the hot and cold surfaces and passes through the center of these surfaces, while the inclination angle varies from 0 to 90° . Based on the results of the linear stability analysis, it was possible to perform non linear, three dimensional numerical simulations based on a spectral element method for a Boussinesq fluid. Our preliminary results show the effect of the rotation rate and the tilted angle on the convective patterns, temperature distribution and heat transfer rate.

¹Thanks to DGAPA-PAPIIT Project: IN117314-3.

Sunday, November 20, 2016 8:00AM - 9:57AM –
Session A10 DFD GPC: Convection and Buoyancy Driven Flows: Environmental
B118-119 - Kiran Bhaganagar, University of Texas, San Antonio

8:00AM A10.00001 Simulation of plume rise: Study the effect of stably stratified turbulence layer on the rise of a buoyant plume from a continuous source by observing the plume centroid

SUDHEER REDDY BHIMIREDDY, Univ of Texas, San Antonio, KIRAN BHAGANAGAR, University of Texas, San Antonio — Buoyant plumes are common in atmosphere when there exists a difference in temperature or density between the source and its ambience.

In a stratified environment, plume rise happens until the buoyancy variation exists between the plume and ambience. In a calm no wind ambience, this plume rise is purely vertical and the entrainment happens because of the relative motion of the plume with ambience and also ambient turbulence. In this study, a plume centroid is defined as the plume mass center and is calculated from the kinematic equation which relates the rate of change of centroids position to the plume rise velocity. Parameters needed to describe the plume are considered as the plume radius, plumes vertical velocity and local buoyancy of the plume. The plume rise velocity is calculated by the mass, momentum and heat conservation equations in their differential form. Our study focuses on the entrainment velocity, as it depicts the extent of plume growth. This entrainment velocity is made up as sum of fractions of plume's relative velocity and ambient turbulence. From the results, we studied the effect of turbulence on the plume growth by observing the variation in the plume radius at different heights and the centroid height reached before loosing its buoyancy.

8:13AM A10.00002 Monitoring of Carbon Dioxide and Methane Plumes from Combined Ground-Airborne Sensors¹

JAMEY JACOB, TAYLOR MITCHELL, WES HONEYCUTT, NICHOLAS MATERER, TYLER LEY, PETER CLARK, Oklahoma State University — A hybrid ground-airborne sensing network for real-time plume monitoring of CO₂ and CH₄ for carbon sequestration is investigated. Conventional soil gas monitoring has difficulty in distinguishing gas flux signals from leakage with those associated with meteorologically driven changes. A low-cost, lightweight sensor system has been developed and implemented onboard a small unmanned aircraft and is combined with a large-scale ground network that measures gas concentration. These are combined with other atmospheric diagnostics, including thermodynamic data and velocity from ultrasonic anemometers and multi-hole probes. To characterize the system behavior and verify its effectiveness, field tests have been conducted with simulated discharges of CO₂ and CH₄ from compressed gas tanks to mimic leaks and generate gaseous plumes, as well as field tests over the Farnsworth CO₂-EOR site in the Anadarko Basin. Since the sensor response time is a function of vehicle airspeed, dynamic calibration models are required to determine accurate location of gas concentration in space and time. Comparisons are made between the two tests and results compared with historical models combining both flight and atmospheric dynamics.

¹Supported by Department of Energy Award DE-FE0012173

8:26AM A10.00003 Dynamics of Downdrafts

EMILY KRUGER, HENRY BURRIDGE, JAMIE PARTRIDGE, Univ of Cambridge, GABRIEL ROONEY, MET Office, PAUL LINDEN, Univ of Cambridge — Downdraft moving cold air within thunderstorms, known as downdrafts, can be used to determine the severity of a storm. Therefore an understanding of them is useful for weather forecasting. Typically in weather forecasting these downdrafts are modelled using the theory of a plume from Morton, Taylor and Turner (1956), which inherently assumes that the plume is long and thin. Downdrafts are generally wider than they are high and hence deviate from the Morton, Taylor and Turner theory. We perform experiments using finite releases of dense fluid from large area sources, releasing a range of volumes of fluids from a cylinder, at a range of heights above the ground which encompasses the typical geometries of downdrafts. By tracking the edges of the release we compare the dynamics of both the fall and the resulting gravity currents of our experimental data to that of previous results. In doing so we find that the resulting gravity current behaves like an axi-symmetric finite release gravity current, whereas the fall doesn't seem to resemble anything previously studied. We hope that these results and future work will allow us to better inform forecasting of weather arising from such downdrafts.

8:39AM A10.00004 Experiments on horizontal convection at high Rayleigh and Prandtl numbers.¹

PIERRE-YVES PASSAGGIA, ALBERTO SCOTTI, BRIAN WHITE, Department of Marine Sciences, University of North Carolina, Chapel Hill, NC 27599, USA — Horizontal convection is a flow driven by a differential buoyancy forcing across a horizontal surface. It has been considered as a simple model to study the influence of heating, cooling and fresh water fluxes at the ocean surface on the meridional overturning circulation. In order to investigate the flow properties and energetics of horizontal convection at high Prandtl numbers, the flow is driven by the diffusion of salt in water across membranes localized at the surface. The resulting experiments are examined for a Prandtl number $\text{Pr} \approx 500$ and Rayleigh numbers up to $\text{Ra} \approx 10^{16}$. Time resolved particle image velocimetry is performed together with planar laser induced fluorescence. To quantify the salt concentration and therefore the density of the fluid, sodium bisulfate is added to the salt water to decrease its pH and thereby reduce the emission rate of the fluorescein dye. Rhodamine WT, insensitive to pH variations, is also introduced to correct for the spatial nonuniformity of the intensity of the laser sheet, a technique also known as ratiometric PLIF (Coppeta & Rogers, 1998). The local turbulent energetics are finally investigated using the local approach to available potential energy of Scotti & White (2014).

¹The authors acknowledge the support by the National Science Foundation Grant N° OCE-1155558.

8:52AM A10.00005 Energetic dynamics of a rotating horizontal convection model of an ocean basin with wind forcing¹

VARVARA ZEMSKOVA, BRIAN WHITE, ALBERTO SCOTTI, University of North Carolina at Chapel Hill — We analyze the energetic dynamics in a rotating horizontal convection model, where flow is driven by a differential buoyancy forcing along a horizontal surface. This model is used to quantify the influence of surface heating and cooling and surface wind stress on the Meridional Overturning Circulation. We study a model of the Southern Ocean in a rectangular basin with surface cooling on one end (the South pole) and surface warming on the other end (mid-latitudes). Free-slip boundary conditions are imposed in the closed box, while zonally periodic boundary conditions are enforced in the reentrant channel. Wind stress and differential buoyancy forcing are applied at the top boundary. The problem is solved numerically using a 3D DNS model based on a finite-volume AMR solver [Santilli and Scotti, J. Comp. Phys., 2015] for the Boussinesq Navier-Stokes equations with rotation. The overall dynamics, including large-scale overturning, baroclinic eddying, turbulent mixing, and resulting energy cascades are investigated using the local Available Potential Energy framework introduced in [Scotti and White, J. Fluid Mech., 2014]. We study the relative contributions of surface buoyancy and wind forcing along with the effects of bottom topography to the energetic balance of this dynamic model.

¹This research is part of the Blue Waters sustained-petascale computing project, supported by the NSF (awards OCI-0725070, ACI-1238993 and ACI-14-44747) and the state of Illinois.

9:05AM A10.00006 Modeling of quasi-constant volume gravity currents due to open-water sediment disposal, JENN WEI ER, ADRIAN WING-KEUNG LAW, Nanyang Technological University, E ERIC ADAMS, Massachusetts Institute of Technology — The near field transport of a sediment cloud in the water column after open-water disposal generally experiences two sequential phases: (i) convective descent phase, during which the cloud behavior is dominated by gravity, and (ii) bottom collapse phase, which upon impact the momentum and buoyancy of the cloud then drive the propagation along the seabed as a gravity current. The spreading of gravity current determines the zone of influence by the disposal event at the seabed. In this study, a modified Box-Model was proposed to assess the behavior of the gravity current. In particular, for the case of a split-barge, which is commonly used for land reclamation and contaminated sediment disposal, the model took into account the finite time period for the barge to fully discharge. Within this period, the sediments were continuously released from the barge, and supplied into the gravity current (as constant flux current). Beyond that, the gravity current continued to spread as a constant volume current instead. The interplay from constant flux to constant volume, which has significant implications on the engineering outcome, has not been addressed before. In addition, the modified Box-Model also included the geometry of the barge opening with a usual rectangular shape, generating differential spreading in both axes, and leading to the final elliptical zone of deposition. A laboratory study was carried out for model verification. The model predicted well the experimental results, while existing engineering models were inadequate due to their oversimplified representations.

9:18AM A10.00007 Clear salt water above sediment-laden fresh water: Interfacial instabilities, NATHAN KONOPLIV, BARTHO SCHULTE, ECKART MEIBURG, Univ of California - Santa Barbara — The stability of an interface separating less dense, clear salt water above from more dense, sediment-laden fresh water below is explored via direct numerical simulations. We find that the destabilizing effects of double-diffusion and particle settling amplify each other above the diffusive interface, whereas they tend to cancel each other below. For moderate settling velocities, plumes form both above and below the interface, whereas for large settling velocities plume formation below the interface is suppressed. We identify the dimensionless parameter that determines in which regime a given flow takes place, along with the critical value at which the transition between the regimes takes place.

9:31AM A10.00008 Laboratory Experiments Modelling Sediment Transport by River Plumes¹, BRUCE SUTHERLAND, MURRAY GINGRAS, CALLA KNUDSON, University of Alberta, LUKE STEVERANGO, McGill University, CHRIS SURMA, University of Alberta — Through lock-release laboratory experiments, the transport of particles by hypopycnal (surface) currents is examined as they flow into a uniform-density and a two-layer ambient fluid. In most cases the tank is tilted so that the current flows over a slope representing an idealization of a sediment-bearing river flowing into the ocean and passing over the continental shelf. When passing into a uniform-density ambient, the hypopycnal current slows and stops as particles rain out, carrying some of the light interstitial fluid with them. Rather than settling on the bottom, in many cases the descending particles accumulate to form a hyperpycnal (turbidity) current that flows downslope. This current then slows and stops as particles both rain out to the bottom and also rise again to the surface, carried upward by the light interstitial fluid. For a hypopycnal current flowing into a two-layer fluid, the current slows as particles rain out and accumulate at the interface of the two-layer ambient. Eventually these particles penetrate through the interface and settle to the bottom with no apparent formation of a hyperpycnal current. Analyses are performed to characterize the speed of the currents and stopping distances as they depend upon experiment parameters.

¹Natural Sciences and Engineering Research Council

9:44AM A10.00009 Suspension-Driven Gravity Surges on Horizontal Surfaces: Effect of the Initial Shape.¹, NADIM ZGHEIB, University of Florida, THOMAS BONOMETTI, IMFT CNRS-INPT-UPS, S. BALACHANDAR, University of Florida — We present results from fully-resolved direct numerical simulations of canonical (axisymmetric and planar) and non-canonical (rectangular) configurations of horizontal suspension-driven gravity surges. We show that the dynamics along the initial minor and major axis of a rectangular release are roughly similar to that of a planar and axisymmetric current, respectively. However, contrary to expectation, we observe under certain conditions the final extent of the deposit from finite releases to surpass that from an equivalent planar current. This is attributed to a converging flow of the particle-laden mixture towards the initial minor axis, a behaviour that was previously reported for scalar-driven currents on uniform slopes. This flow is observed to be correlated with the travelling of a perturbation wave generated at the extremity of the longest side that reaches the front of the shortest side in a finite time. A semi-empirical explicit expression (based on established relations for planar and axisymmetric currents) is proposed to predict the extent of the deposit in the entire x-y plane. Finally we observe that for the same initial volume of a suspension-driven gravity surge, a release of larger initial horizontal aspect-ratio is able to retain particles in suspension for longer periods of time.

¹ExxonMobil Upstream Research (EM 09296); NSF (OISE-0968313); CALMIP (P1525)

Sunday, November 20, 2016 8:00AM - 9:57AM – Session A11 Electrokinetic Flows: Concentration Polarization and Instability C120-121-122 - Ali Mani, Stanford University

8:00AM A11.00001 From electroconvective vortices to current hot spots on ion selective membranes subject to concentration polarization, KAREN WANG, ALI MANI, Stanford University — Electroconvective instabilities near ion-selective surfaces have been shown to greatly enhance ion transport and play a significant role in a wide range of applications in electrochemistry. When the driving voltage exceeds a threshold, electroconvection becomes chaotic and leads to intermittent spikes of current density on the ion-selective surface. We present an investigation of this phenomenon by considering a canonical setting consisting of a symmetric binary electrolyte next to a flat, ion-selective membrane subject to an external driving voltage. By tracking individual rolls of vortices, we reveal the common mechanism under which the three-way coupled fluid dynamics, ion transport, and electrostatic effects lead to advective displacement of ion concentration field, sustained vortices and vortex migration, and current hot spots on the membrane.

8:13AM A11.00002 The Equivalent Electrokinetic Circuit Model of Ion Concentration Polarization Layer: Electrical Double Layer, Extended Space Charge and Electroconvection

, INHEE CHO, KEON HUH, Seoul National University, RHOKYUN KWAK, Korea Institute of Science and Technology, HYOMIN LEE, SUNG JAE KIM, Seoul National University — The first direct chronopotentiometric measurement was provided to distinguish the potential difference through the extended space charge (ESC) layer which is formed with the electrical double layer (EDL) near a perm-selective membrane. From this experimental result, the linear relationship was obtained between the resistance of ESC and the applied current density. Furthermore, we observed the step-wise distributions of relaxation time at the limiting current regime, confirming the existence of ESC capacitance other than EDLs. In addition, we proposed the equivalent electrokinetic circuit model inside ion concentration polarization (ICP) layer under rigorous consideration of EDL, ESC and electro-convection (EC). In order to elucidate the voltage configuration in chronopotentiometric measurement, the EC component was considered as the dependent voltage source which is serially connected to the ESC layer. This model successfully described the charging behavior of the ESC layer with or without EC, where both cases determined each relaxation time, respectively. Finally, we quantitatively verified their values utilizing the Poisson-Nernst-Planck equations. Therefore, this unified circuit model would provide a key insight of ICP system and potential energy-efficient applications.

8:26AM A11.00003 Micro/nanofluidic Diode using Asymmetric Ion Concentration Polarization Layer

, SEOYUN SOHN, INHEE CHO, SUNG JAE KIM, Seoul National University — Recent developments of ion concentration polarization (ICP) theory would suggest that an over-limiting conductance (OLC) of the device is subject to the morphology of ICP layer and a micro-structure is able to alter the morphology. In this study, we demonstrated an ion rectification resulted only from asymmetric microscale structures, while conventional nanofluidic diode applications have usually employed a nanoscale asymmetry which requires sophisticate and expensive fabrication processes. We designed two dead-end microchannels incorporated with the nanoporous membrane. The difference in width of the microchannels was designed to yield asymmetry to the device. Cyclic voltammetry measurement was conducted to investigate the OLC behaviors on both forward and reverse bias. The diodic characteristics on I-V responses were observed at various ratio of the different microchannel width. In addition, we experimentally verified the logarithmical linearity between the ratios and the rectification ratios of OLC. This quantitative analysis would guide the further application utilizing microscale asymmetric diode device that now can be realized with minimum fabrication endeavors.

8:39AM A11.00004 Spontaneous water filtration of bio-inspired membrane¹

, KIWOONG KIM, HYEJEONG KIM, SANG JOON LEE, Pohang Univ of Sci & Tech — Water is one of the most important elements for plants, because it is essential for various metabolic activities. Thus, water management systems of vascular plants, such as water collection and water filtration have been optimized through a long history. In this view point, bio-inspired technologies can be developed by mimicking the nature's strategies for the survival of the fittest. However, most of the underlying biophysical features of the optimized water management systems remain unsolved. In this study, the biophysical characteristics of water filtration phenomena in the roots of mangrove are experimentally investigated. To understand water-filtration features of the mangrove, the morphological structures of its roots are analyzed. The electrokinetic properties of the root surface are also examined. Based on the quantitatively analyzed information, filtration of sodium ions in the roots are visualized. Motivated by this mechanism, spontaneous desalination mechanism in the root of mangrove is proposed by combining the electrokinetics and hydrodynamic transportation of ions. This study would be helpful for understanding the water-filtration mechanism of the roots of mangrove and developing a new bio-inspired desalination technology.

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

8:52AM A11.00005 Hydrodynamic Model of Desalination by “Overlimiting” Electrodialysis with Electroconvective Vortices

, RHOKYUN KWAK, Korea Institute of Science and Technology, VAN SANG PHAM, Hanoi University of Science and Technology, JONGYOON HAN, Massachusetts Institute of Technology — In 1968, Sonin and Probstein developed a hydrodynamic theory of desalination by electrodialysis (Desalination, 5, 293-329, 1968). Under a laminar flow between ion exchange membranes, linear ion concentration gradients are developed near the membranes by ion concentration polarization (ICP) in Ohmic-limiting current regimes. This linear ICP determines the relations between current, voltage, and desalting performance. Here, we revisit the hydrodynamic model with nonlinear ICP phenomenon at overlimiting currents. In this regime, electroconvective vortices on the membrane induce flat and extremely low concentration zones. Based on the previous prediction of the vortex height under shear flow (Kwak et al., PRL, 110, 114501, 2013), we verify that the height directly represents the amount of the removed salt because there is almost no ion in the vortices. Next, from the mass continuity of ions, the amount of the removed salts is equal to the ion flux through the membrane (i.e. current); as a result, we can develop the relations between current, voltage, and salt removal. Lastly, from these relations, power consumption and desalination cost can be calculated to find the optimal operating condition of overlimiting electrodialysis.

9:05AM A11.00006 Controlling turbulent drag across electrolytes using electric fields

, ALPHA LEE, RODOLFO OSTILLA-MNICO, Harvard University — Controlling friction is a crucial problem in engineering science. Using direct numerical simulation, we investigate the phenomenology of turbulent Couette flows in electrolytes sheared by charged surfaces. We show how the presence of large shear rates affects the structure, dynamics and stress generation in the electrical double layer. The constant injection of energy from the sheared boundaries drives the double layer far from thermodynamic equilibrium, thus placing conventional statistical physical intuitions on a more tenuous footing. Critically, we uncover regimes where friction associated with turbulent dissipation could be controlled by applying an electric field. The implications of our results on chaotic electrokinetic flows and the non-equilibrium electrical double layer in other electrokinetic settings will also be discussed.

9:18AM A11.00007 Electrokinetic Fingering In Hele-Shaw Cells

, MOHAMMAD MIRZADEH, MARTIN BAZANT, Department of Chemical Engineering, MIT — Large scale flow problems in porous media, such as those encountered in underground oil reservoirs, are typically described by the Darcys law. However, it is well known that many underground rock formations contain surface groups and minerals that dissociate in the presence of water. Convection of these charges by the pressure driven flow can then set up streaming current and streaming potential that affects the flow. Furthermore, electric fields that are often used to enhance oil recovery, e.g. by reducing the oil's viscosity through electro-thermal heating, drive electro-osmotic flows that could set up very large pressure in small pores. The full description of fluid flow thus requires a solution to the fully coupled electrokinetic problem. In their seminal work, Saffman and Taylor showed that the moving interface between two immiscible fluids in a porous medium becomes unstable if pushed by the low-viscosity fluid. Here we report on the role of electrokinetic phenomena on stability of these viscous fronts in Hele-Shaw cells by using analytic as well as numerical approaches. Interestingly, we find that the instability could be suppressed if the right physical conditions are met or otherwise enhanced, leading to greater mixing of two fluids.

9:31AM A11.00008 Energy harvesting through charged nanochannels using external flows of different salt concentrations, SOURAYON CHANDA, PEICHUN AMY TSAI, Mechanical Engineering, University of Alberta — Renewable electricity may be generated by mixing of two solutions of different salt concentrations through charged nanochannels or pores, by leveraging ion-selective effect of the nano-confinements. We numerically investigate such a continuous power generation system using reverse electrodialysis (RED) with external flows. In the simulation model, two reservoirs are connected using a nanochannel of constant surface charge density. Solutions of high and low concentrations flow through the two reservoirs at a constant velocity. We examine the effects of (salt) concentration gradients and nanochannel dimensions on the power generation. Moreover, the effect of external flow velocity on the process is analyzed. Our results show that the maximum surface charge density, open circuit voltage, channel resistance, and energy conversion efficiency of the process are significantly affected by the difference of the high and low concentrations and the nanochannel dimension ratio.

9:44AM A11.00009 Coherent structures of electrokinetic instability in microflows, KAUSHILENDRA DUBEY, AMIT GUPTA, SUPREET SINGH BAHGA, Indian Inst of Tech-New Delhi — Electrokinetic instabilities occur in fluid flow where gradients in electrical properties of fluids, such as conductivity and permittivity, lead to a destabilizing body force. We present an experimental investigation of electrokinetic instability (EKI) in a microchannel flow with orthogonal conductivity gradient and electric field, using time-resolved visualization of a passive fluorescent scalar. This particular EKI has applications in rapid mixing at low Reynolds number in microchannels. Previous studies have shown that such EKI can be characterized by the electric Rayleigh number (Rae) which is the ratio of diffusive and electroviscous time scales. However, these studies were limited to temporal power spectra and time-delay phase maps of fluorescence data at a single spatial location. In the current work, we use dynamic mode decomposition (DMD) of time-resolved snapshots of EKI to investigate the spatio-temporal coherent structures of EKI for a wide range of Rae . Our analysis yields spatial variation of modes in EKI along with their corresponding temporal frequencies. We show that EK instability with orthogonal conductivity-gradient and electric field can be characterized by transverse and longitudinal coherent structures which depend strongly on Rae .

Sunday, November 20, 2016 8:00AM - 9:57AM –
Session A12 Multiphase Flows: Recent Advances in Numerical Methods C123 - Brian Turnquist, Montana State University

8:00AM A12.00001 Intrusive Method for Uncertainty Quantification in a Multiphase Flow Solver¹, BRIAN TURNQUIST, MARK OWKES, Montana State University — Uncertainty quantification (UQ) is a necessary, interesting, and often neglected aspect of fluid flow simulations. To determine the significance of uncertain initial and boundary conditions, a multiphase flow solver is being created which extends a single phase, intrusive, polynomial chaos scheme into multiphase flows. Reliably estimating the impact of input uncertainty on design criteria can help identify and minimize unwanted variability in critical areas, and has the potential to help advance knowledge in atomizing jets, jet engines, pharmaceuticals, and food processing. Use of an intrusive polynomial chaos method has been shown to significantly reduce computational cost over non-intrusive collocation methods such as Monte-Carlo. This method requires transforming the model equations into a weak form through substitution of stochastic (random) variables. Ultimately, the model deploys a stochastic Navier Stokes equation, a stochastic conservative level set approach including reinitialization, as well as stochastic normals and curvature. By implementing these approaches together in one framework, basic problems may be investigated which shed light on model expansion, uncertainty theory, and fluid flow in general.

¹NSF Grant Number 1511325

8:13AM A12.00002 Marker Re-Distancing and Sharp Reconstruction for High-Order Multi-Material Interface Evolution¹, ROBERT NOURGALIEV, PATRICK GREENE, SAM SCHOFIELD, Lawrence Livermore National Laboratory — A new method for high-order front evolution on arbitrary meshes is introduced. The method is a hybrid of a Lagrangian marker tracking with a Discontinuous Galerkin projection based level set re-distancing. This Marker-Re-Distancing (MRD) method is designed to work accurately and robustly on unstructured, generally highly distorted meshes, necessitated by applications within ALE-based hydro-codes. Since no PDE is solved for re-distancing, the method does not have stability time step restrictions, which is particularly useful in combination with AMR, used here to efficiently resolve fine interface features. A high-order (implemented up to the 6th-order) level set approach is utilized for a new sharp treatment of mix elements, which reconstructs multiple-per-element solution fields (one for each material present in the mix element). Reconstruction incorporates interfacial jump conditions, which are enforced in the least-squares sense at the interfacial marker positions provided by MRD. Since no explicit differentiation across the interface is involved in the assembly of residuals for mass, momentum and energy equations, the method is capable of capturing discontinuous solutions at multi-material interfaces with high order, and without Gibbs oscillations.

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

8:26AM A12.00003 A Finite Element Method for Simulation of Compressible Cavitating Flows¹, EHSAN SHAMS, FAN YANG, YU ZHANG, ONKAR SAHNI, MARK SHEPHARD, ASSAD OBERAI, Rensselaer Polytechnic Institute — This work focuses on a novel approach for finite element simulations of multi-phase flows which involve evolving interface with phase change. Modeling problems, such as cavitation, requires addressing multiple challenges, including compressibility of the vapor phase, interface physics caused by mass, momentum and energy fluxes. We have developed a mathematically consistent and robust computational approach to address these problems. We use stabilized finite element methods on unstructured meshes to solve for the compressible Navier-Stokes equations. Arbitrary Lagrangian-Eulerian formulation is used to handle the interface motions. Our method uses a mesh adaptation strategy to preserve the quality of the volumetric mesh, while the interface mesh moves along with the interface. The interface jump conditions are accurately represented using a discontinuous Galerkin method on the conservation laws. Condensation and evaporation rates at the interface are thermodynamically modeled to determine the interface velocity. We will present initial results on bubble cavitation the behavior of an attached cavitation zone in a separated boundary layer.

¹We acknowledge the support from Army Research Office (ARO) under ARO Grant W911NF-14-1-0301.

8:39AM A12.00004 Numerical simulation of particle dynamics at a fluid interface¹

, PENGTAO YUE, Virginia Tech — Particles straddling a fluid interface exhibit rich dynamics due to the coexistence of moving boundaries, fluid interfaces, and moving contact lines. For instance, as a particle falls onto a liquid surface, it may sink, float, or even bounce off depending on a wide range of parameters. To better understand the dynamics of such a multiphase system, we develop a finite-element based arbitrary Lagrangian-Eulerian-phase-field method. The governing equations for particles and fluids are solved in a unified variational framework that satisfies an energy law. We first validate our code by computing three problems found in literature: sinking of a horizontal cylinder through an air-water interface, sinking of a sphere through an air-oil interface at small Reynolds numbers, and bouncing of a sphere after its normal impact onto an air-water interface. Our numerical results show good agreements with experimental data. We then investigate the effect of wetting properties, including static contact angle, slip length, and wall energy relaxation, on particle dynamics at the fluid interface.

¹This work is supported by NSF DMS-1522604.

8:52AM A12.00005 An inviscid regularization technique for the simulation of compressible multiphase flow , BAHMAN ABOULHASANZADEH, KAMRAN MOHSENI, University of Florida

— A common feature of flow problems involving shocks, turbulence, and/or two-phase flows is the k -infinity irregularity. We present an inviscid regularization technique, dubbed observable regularization, for the simulation of compressible multiphase flows. In this technique, we use the observable divergence theorem to derive the conservation equations considering the observability limit in any computational or physical system. To avoid contamination of the result with numerical diffusion a pseudo-spectral technique is used to discretize the conservation equations. This methodology has been tested successfully for regularizing single-phase problems with shocks and/or turbulence. Using observable Euler equations, shock-bubble and shock-drop interactions are simulated and the results are compared with available experimental data from literature, showing good agreement. Observable equations are capable of simultaneously regularizing problems with shocks, turbulence, and/or sharp interfaces without the need for treating each aspect separately.

9:05AM A12.00006 Continuity waves in fully resolved simulations of settling particles¹ , DANIEL WILLEN, ADAM SIERAKOWSKI, Johns Hopkins University, ANDREA PROSPERETTI, University of Houston, University of Twente

— Fully resolved simulations of 500 to 2,000 particles settling in a fluid have been conducted with the Physalis method. A new approach to the reconstruction of pseudo-continuum fields is described and is used to examine the results with the purpose of identifying concentration waves. The velocity of concentration waves is successfully deduced from the simulations. A comparison of the results with continuity wave theory shows good agreement. Several new insights about the particle microstructure conditionally averaged on volume fraction and velocity are also described.

¹This work is supported by NSF award CBET1335965.

9:18AM A12.00007 Physics-Based Preconditioning for the Numerical Solution of the All-Speed Compressible Navier-Stokes Equations with Laser-Induced Phase Change , BRIAN WESTON, University of California, Davis, ROBERT NOURGALIEV, Lawrence Livermore National Laboratory, JEAN-PIERRE DELPLANQUE, University of California, Davis, ANDY ANDERSON, Lawrence Livermore National Laboratory

— The numerical simulation of flows associated with metal additive manufacturing processes such as selective laser melting and other laser-induced phase change applications present new challenges. Specifically, these flows require a fully compressible formulation since rapid density variations occur due to laser-induced melting and solidification of metal powder. We investigate the preconditioning for a recently developed all-speed compressible Navier-Stokes solver that addresses such challenges. The equations are discretized with a reconstructed Discontinuous Galerkin method and integrated in time with fully implicit discretization schemes. The resulting set of non-linear and linear equations are solved with a robust Newton-Krylov (NK) framework. To enable convergence of the highly ill-conditioned linearized systems, we employ a physics-based operator split preconditioner (PBP), utilizing a robust Schur complement technique. We investigate different options of splitting the physics (field) blocks as well as different block solvers on the reduced preconditioning matrix. We demonstrate that our NK-PBP framework is scalable and converges for high CFL/Fourier numbers on classic problems in fluid dynamics as well as for laser-induced phase change problems.

9:31AM A12.00008 Development of multiphase CFD flow solver in OpenFOAM , CHAD ROLLINS, HONG LUO, North Carolina State University, Department of Mechanical & Aerospace Engineering, NAM DINH, North Carolina State University, Department of Nuclear Engineering

— We are developing a pressure-based multiphase (Eulerian) CFD solver using OpenFOAM with Reynolds-averaged turbulence stress modeling. Our goal is the evaluation and improvement of the current OpenFOAM two-fluid (Eulerian) solver in boiling channels with a motivation to produce a more consistent modeling and numerics treatment. The difficulty lies in the presence of the many forces and models that are tightly non-linearly coupled in the solver. Therefore, the solver platform will allow not only the modeling, but the tracking as well, of the effects of the individual components (various interfacial forces/heat transfer models) and their interactions. This is essential for the development of a robust and efficient solution method. There has been a lot of work already performed in related areas that generally indicates a lack of robustness of the solution methods. The objective here is therefore to identify and develop remedies for numerical/modeling issues through a systematic approach to verification and validation, taking advantage of the open source nature of OpenFOAM. The presentation will discuss major findings, and suggest strategies for robust and consistent modeling (probably, a more consistent treatment of heat transfer models with two-fluid models in the near-wall cells).

9:44AM A12.00009 Numerical Simulation of Compressible Multi-phase flows using HLLC extension of AUSM+-up Scheme¹ , GAURAV DHIR, PEC University of Technology, KOWSIK BODI, IIT Bombay

Bombay — Solving Multi-fluid equations has always required an onerous effort from researchers with regards to implementing an appropriate numerical scheme which could capture the various facets of such type of flows along with the interaction between the various phases present. Additionally, multi-phase flows bring with them peculiar mathematical properties such as non-hyperbolicity and non-conservativeness which further increases the complexity involved. Our presentation shall present an insight into the advantages and limitations of several numerical schemes proposed in the past and propose to use the HLLC extension of AUSM+-up approach to model such type of flows. We use the single pressure based stratified flow concept and by presenting several test cases, we prove that our method robustly computes multi-phase flow involving discontinuities, such as shock waves and fluid interfaces. Additionally, we present a formulation to incorporate phase transition within multi-fluid equations and establish the validity of this method by presenting several two dimensional test cases such as the Shock-Water Column Interaction problem, the Water-Shock/Air Bubble Interaction problem and the 2D Underwater Explosion problem.

¹Industrial Research and Consultancy Centre, IIT Bombay

Sunday, November 20, 2016 8:00AM - 9:31AM –
Session A13 Geophysical Fluid Dynamics: Sediment Transport & Gravity Currents
C124 - Justin Finn, University of Liverpool

8:00AM A13.00001 Coupled large-eddy simulation and morphodynamics of a large-scale river under extreme flood conditions¹, ALI KHOSRONEJAD, FOTIS SOTIROPOULOS, Stony Brook University, STONY BROOK UNIVERSITY TEAM — We present a coupled flow and morphodynamic simulations of extreme flooding in 3 km long and 300 m wide reach of the Mississippi River in Minnesota, which includes three islands and hydraulic structures. We employ the large-eddy simulation (LES) and bed-morphodynamic modules of the VFS-Geophysics model to investigate the flow and bed evolution of the river during a 500 year flood. The coupling of the two modules is carried out via a fluid-structure interaction approach using a nested domain approach to enhance the resolution of bridge scour predictions. The geometrical data of the river, islands and structures are obtained from LiDAR, sub-aqueous sonar and in-situ surveying to construct a digital map of the river bathymetry. Our simulation results for the bed evolution of the river reveal complex sediment dynamics near the hydraulic structures. The numerically captured scour depth near some of the structures reach a maximum of about 10 m. The data-driven simulation strategy we present in this work exemplifies a practical simulation-based-engineering-approach to investigate the resilience of infrastructures to extreme flood events in intricate field-scale riverine systems.

¹This work was funded by a grant from Minnesota Dept. of Transportation

8:13AM A13.00002 Analysis of Reynolds stress budgets in LES of Langmuir supercells under crosswind currents in a coastal ocean¹, ANDRES TEJADA-MARTINEZ, JIE ZHANG, University of South Florida — Langmuir supercells (LSCs) in coastal oceans consist of parallel counter rotating vortices engulfing the water column in unstratified conditions. These cells have been observed in continental shelf regions 15–30 meters deep during the passage of storms. LSCs are aligned roughly in the wind direction and are generated via interaction of the wind-driven shear current and Stokes drift velocity induced by surface gravity waves. LSCs have been determined to be an important contributor to the suspension of sediments and their overall transport across shelves. It has also been shown that tidal forcing distorts and weakens LSCs, inhibiting their potential for sediment suspension. Large-eddy simulations of LSCs in flows driven by a surface wind stress and a constant crosswind pressure gradient (representative of crosswind tidal forcing) have been performed. Although a crosswind tidal current stronger than the wind-driven current is able to break up the LSCs giving rise to smaller scale, weaker Langmuir cells (LCs), analysis of Reynolds shear stress budgets reveals that non-local transport remains significant relative to flow without LCs. This demonstrates the need for a non-local transport term in Reynolds shear stress and turbulent scalar flux closures for coastal flows with LCs.

¹Support from the US National Science Foundation and the Gulf of Mexico Research Initiative is gratefully acknowledged

8:26AM A13.00003 Regimes of sediment-turbulence interaction and guidelines for simulating the multiphase bottom boundary layer¹, JUSTIN FINN, MING LI, University of Liverpool — Characterizing the interaction of mobile sediments with a turbulent boundary layer driven by waves and/or currents represents an important scientific and engineering challenge. To approach this, Balachandars scaling relations for particle Reynolds number and Stokes number (IJMF, vol. 35, pg 801110, 2009) are recast in terms of Shields parameter, Θ , particle Galileo number, Ga , and particle-to-fluid density ratio, s . This allows the modified Shields diagram to be partitioned into at least five regimes, where distinct primary mechanisms of sediment-turbulence interaction can be identified. Additionally, practical guidelines for selecting an appropriate multiphase modeling strategy for direct and large eddy simulation (DNS/LES) of the bottom boundary layer are proposed based on the results.

¹This research received support from the UK Engineering and Physical Science Research Council (EPSRC) through the UK/Dutch SINBAD project (EP/J005541/1).

8:39AM A13.00004 Partial-depth lock-release and related phenomena: a vorticity-based analysis, MOHAMMAD AMIN KHODKAR, Ph.D. student at University of California, Santa Barbara, MOHAMAD NASR-AZADANI, posdoctoral scholar at University of California, Santa Barbara, ECKART MEIBURG, Professor at University of California, Santa Barbara — We present a vorticity-based model for partial-depth Boussinesq lock exchange flows, without empirical closure assumptions. Upon release of the lock fluid, experiments and DNS simulations show that a gravity current front forms, followed by a rarefaction wave. For sufficiently large ratios of lock height to tank height, a left-propagating internal bore will be generated as well. By employing the conservation of mass in each fluid, along with the overall vorticity, we propose a set of ODEs that predicts the velocity distribution within the rarefaction wave, in addition to the velocity and height of the gravity current front. The model furthermore predicts when a bore is produced, and how it influences the dynamics of the flow. The model predictions are compared to DNS results, as well as to theoretical and experimental results of earlier investigations, and good agreement is observed for all flow properties.

8:52AM A13.00005 The dynamics of bottom-boundary gravity currents propagating over a submerged array of cylinders¹, JIAN ZHOU, Colorado State University, TIM WILLIAMS, MEGAN BALL, University of Canterbury, CLAUDIA CENEDESE, Woods Hole Oceanographic Institution, SUBHAS VENAYAGAMOORTHY, Colorado State University, ROGER NOKES, University of Canterbury — The structure and propagation of lock-exchange bottom-boundary gravity currents (BBGC) in a rectangular horizontal channel containing a submerged array of cylindrical obstacles are investigated using experiments and large eddy simulations. Excellent agreement on the front velocity between the experimental and numerical results is found. A broad-range three-dimensional parametric study is performed in which the solid volume fraction ϕ of the array is varied continuously from 0 (flat-bed case) to 1 (solid-slab case), and the submergence ratio is varied from 1 (emergent) to 10 (deeply submerged). Both in-line and staggered cylinder arrangements are considered. The various flow regimes arising from the current-array interaction and their mutual transitions are investigated in detail in terms of front velocity, density, vorticity, turbulent mixing and global energy budget. Our analysis provides a new framework for predicting the front velocity of BBGCs propagating over a submerged array of cylinders under the influence of array inhomogeneity.

¹Funded by the Office of Naval Research and National Science Foundation

9:05AM A13.00006 Sediment morpho-dynamics induced by a swirl-flow: an experimental study¹, ALFREDO GONZALEZ-VERA, MATIAS DURAN-MATUTE, GERTJAN VAN HEIJST, Eindhoven Univ of Tech —

This research focuses on a detailed experimental study of the effect of a swirl-flow over a sediment bed in a cylindrical domain. Experiments were performed in a water-filled cylindrical rotating tank with a bottom layer of translucent polystyrene particles acting as a sediment bed. The experiments started by slowly spinning the tank up until the fluid had reached a solid-body rotation at a selected rotation speed (Ω_i). Once this state was reached, a swirl-flow was generated by spinning-down the system to a lower rotation rate (Ω_f). Under the flow's influence, particles from the bed were displaced, which changed the bed morphology, and under certain conditions, pattern formation was observed. Changes in the bed height distribution were measured by utilizing a Light Attenuation Technique (LAT). For this purpose, the particle layer was illuminated from below. Images of the transmitted light distribution provided quantitative information about the local thickness of the sediment bed. The experiments revealed a few characteristic regimes corresponding to sediment displacement, pattern formation and the occurrence of particle pick-up. Such regimes depend on both the Reynolds (Re) and Rossby (Ro) numbers.

¹This research is funded by CONACYT (Mexico) through the Ph.D. grant (383903) and NWO (the Netherlands) through the VENI grant (863.13.022)

9:18AM A13.00007 A study of saltation process using hydrogel particles¹, STELLA WANG, North Carolina School of Science and Mathematics, Durham ,NC, YUCHEN ZHAO, ROBERT BEHRINGER, Department of Physics, Duke University, Durham, NC —

Saltation is a natural process for sediments transported by flow, and occurs in situations such as wind-driven sand dunes in the dessert, and rivers or streams where fluid motion drives gravel. The onset of grain motions is set by the strength of the shear, and grains exhibit rolling, successive jumping where they are lifted by the turbulence. It is an open issue as to how the grain size affects saltation transport, particle velocities and mass fluxes, etc[1], and also how the inelastic collision between grains affects saltation. Here, we describe a new saltation experiment using hydrogel particles immersed in uniform flow of water. Because the refraction indexes of particles and the fluid are nearly matched, the hydrogel particles can be imaged by a parallel light source, resulting in overlapping dark rings that not only reflect lateral positions, but also depths in one 2D image at one time. Mono-disperse particles are used and their size is adjusted by changing salt concentrations in the fluid. Preliminary results show that the softness of hydrogel particles leads to relatively large collisional losses. This property allows us to explore the phase diagram of saltation transport in the in-elastic collision regime.

¹NSF-DMR-1206351 and the William M. Keck Foundation

Sunday, November 20, 2016 8:00AM - 9:44AM –

Session A14 Waves: Air-water Surface Waves C125-126 - Di Yang, University of Houston

8:00AM A14.00001 Experimental study of temporal evolution of waves under transient wind conditions, ANDREY ZAVADSKY, LEV SHEMER, Tel Aviv Univ — Temporal variation of the waves excited by nearly sudden wind forcing over an initially still water surface is studied in a small wind-wave flume at Tel Aviv University for variety of fetches and wind velocities. Simultaneous measurements of the surface elevation using a conventional capacitance wave-gauge and of the surface slope in along-wind and cross-wind directions by a laser slope gauge were performed. Variation with time of two components of instantaneous surface velocity was measured by particle tracking velocimetry. The size of the experimental facility and thus relatively short characteristic time scales of the phenomena under investigation, as well as an automated experimental procedure controlling the experiments made it possible to record a large amount of independent realizations for each wind-fetch condition. Sufficient data were accumulated to compute reliable ensemble averaged temporal variation of governing wave parameters. The essentially three-dimensional structure of wind-waves at all stages of evolution is demonstrated. The results obtained at each wind-fetch condition allowed to characterize the major stages of the evolution of the wind-wave field and to suggest a plausible scenario for the initial growth of the wind-waves.

8:13AM A14.00002 Experimental observation of gravity-capillary solitary waves generated by a moving air-suction¹, BEOMCHAN PARK, YEUNWOO CHO, Korea Advanced Institute of Science and Technology — Gravity-capillary solitary waves are generated by a moving air-suction forcing instead of a moving air-blowing forcing. The air-suction forcing moves horizontally over the surface of deep water with speeds close to the minimum linear phase speed $c_{\min} = 23\text{cm/s}$. Three different states are observed according to forcing speed below c_{\min} . At relatively low speeds below c_{\min} , small-amplitude linear circular depressions are observed, and they move steadily ahead of and along with the moving forcing. As the forcing speed increases close to c_{\min} , however, nonlinear 3-D gravity-capillary solitary waves are observed, and they move steadily ahead of and along with the moving forcing. Finally, when the forcing speed is very close to c_{\min} , oblique shedding phenomena of 3-D gravity-capillary solitary waves are observed ahead of the moving forcing. We found that all the linear and nonlinear wave patterns generated by the air-suction forcing correspond to those generated by the air-blowing forcing. The main difference is that 3-D gravity-capillary solitary waves are observed ahead of the air-suction forcing, whereas the same waves are observed behind the air-blowing forcing.

¹This work was supported by the Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Science, ICT & Future Planning (NRF-2014R1A1A1002441).

8:26AM A14.00003 Experiments on waves under impulsive wind forcing in view of the Phillips (1957) theory, LEV SHEMER, ANDREY ZAVADSKY, Tel Aviv Univ — Only limited information is currently available on the initial stages of wind-waves growth from rest under sudden wind forcing; the mechanisms leading to the appearance of waves are still not well understood. In the present work, waves emerging in a small-scale laboratory facility under the action of step-like turbulent wind forcing are studied using capacitance and laser slope gauges. Measurements are performed at a number of fetches and for a range of wind velocities. Taking advantage of the fully automated experimental procedure, at least 100 independent realizations are recorded for each wind velocity at every fetch. The accumulated data sets allow calculating ensemble-averaged values of the measured parameters as a function of time elapsed from the blower activation. The accumulated results on the temporal variation of wind-wave field initially at rest allow quantitative comparison with the theory of Phillips (1957). Following Phillips, appearance of the initial detectable ripples was considered first, while the growth of short gravity waves at later times was analyzed separately. Good qualitative and partial quantitative agreement between the Phillips predictions and the measurements was obtained for both those stages of the initial wind-wave field evolution.

8:39AM A14.00004 Gravity-capillary waves in countercurrent air/water turbulent

flow¹, FRANCESCO ZONTA, TU Wien, MIGUEL ONORATO, University of Torino, ALFREDO SOLDATI, University of Udine; TU Wien — Using the Direct Numerical Simulation (DNS) of the Navier-Stokes equations, we 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 constant and varying We and Fr , we show that, in the initial stages of the wave generation process, the amplitude of the interface elevation grows in time as $t^2/5$. Wavenumber spectra, $E(kx)$, 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 is addressed.

¹CINECA supercomputing centre (Bologna, Italy) and ISCRA Computing Initiative are gratefully acknowledged for generous allowance of computer resources. Support from PRIN (under Grant 2006098584 004) is gratefully acknowledged. Support from Regione Autonoma

8:52AM A14.00005 Wind-wave generation using interface tracking , ESPEN AKERVIK, Norwegian Defence Research Establishment (FFI) —

The wind-wave generation process in a periodic open channel is studied by means of Large Eddy Simulation, using the Volume of Fluid method to track the interface. The coupled system is initiated by imposing a turbulent air flow at $Re_\tau = 395$ on top of water at rest. Surface tension effects are excluded and the Froude number is chosen so as to fit equilibrium slow moving waves inside the computational domain. In the initial stage, the surface deformation consists of streamwise elongated narrow structures. These may be seen as footprints of the near wall streaks in the turbulent air flow. This phase is associated with linear growth in amplitude, and the behavior of the air flow is largely unaffected by the surface deformations. In the second stage, localized slow moving ($c/u_\tau < 10$) wave packets appear, and the air flow becomes linked to the waves. This phase is associated with exponential growth of the waves. In the third stage, non-linear interactions occur, resulting in redistribution of energy. The growth rates are compared to previous simulations and theoretical results.

9:05AM A14.00006 Numerical investigation of energy transfer in coupled wind and

wave system , XUANTING HAO, LIAN SHEN, University of Minnesota — Energy transfer in the wind-wave system is one of the key physical processes in air-sea interactions. In open oceans, the wind input, nonlinear wave interaction, and wave dissipation are three key mechanisms of energy transfer that govern the wave field evolution. In this study, we conduct a series of high-fidelity numerical experiments using dynamically coupled large-eddy simulation for the turbulent wind and high-order spectral simulation for the waves. By directly analyzing the wave statistics data obtained from our wave-phase-resolved deterministic simulations, we monitor the spectral evolution of the wave field. We observe the frequency downshift phenomenon and the self-similarity of the wind-forced nonlinear wave field throughout the numerical experiment. Further analysis quantifies the wind input, nonlinear wave interaction, as well as wave dissipation. The nonlinear wave interaction is found to be dominant over the wind input and wave dissipation locally in the spectral domain despite its overall energy-conserving property.

9:18AM A14.00007 Wave-interference Effects in the Presence of a Shear Current

, YAN LI, SIMEN ELLINGSEN, Norwegian University of Science and Technology, FRANCIS NOBLESSE, Shanghai Jiao Tong University — Wave-interference effects, based on a 2-point wavemaker model of monohull ships, are analysed when a shear current of uniform vorticity is present. Indeed, wave interferences in the presence of a shear current, similar to the cases in finite water depth, are considerably more complicated than in deep water without vorticity. The effects of a shear current on far-field waves that are formed by 2-point wavemaker models greatly depend on the shear Froude number VS/g , where V is the speed of the ship, S is the uniform vorticity of the shear, g is the gravitational acceleration, as well as the angle between the ship's motion direction and the shear current. Various circumstances, under which ray angles of the highest waves that are associated with constructive interferences between waves are much narrower than the wake angles of the cusps or the asymptotes of wave patterns formed by Kelvin's classical 1-point wavemaker, are shown. In particular, cusp shear Froude numbers Fr_{scusp} where ray angles of the highest waves are equal to the cusp angles are determined. As for shear Froude numbers VS/g that are larger than Fr_{scusp} , the apparent angles where highest waves are found are significantly smaller than the cusp/asymptote angles. Furthermore, the asymmetry due to the presence of a shear current results in remarkable differences between the cases where a ship moves upstream or downstream.

9:31AM A14.00008 Shear turbulence, Langmuir circulation and scalar transfer at an air-water interface , AMINE HAFSI, ANDRES TEJADA-MARTINEZ, University of South Florida, FABRICE VERON, University of Delaware — DNS of an initially quiescent coupled air-water interface driven by an air-flow with free stream speed of 5 m/s generates gravity-capillary waves and small-scale (centimeter-scale) Langmuir circulation (LC) beneath the interface. In addition to LC, the waterside turbulence is characterized by shear turbulence with structures similar to classical "wall streaks" in wall-bounded flow. These streaks, denoted here as "shear streaks", consist of downwind-elongated vortices alternating in sign in the crosswind direction. The presence of interfacial waves causes interaction between these vortices giving rise to bigger vortices, namely LC. LES with momentum equation augmented with the Craik-Leibovich (C-L) vortex force is used to understand the roles of the shear streaks (i.e. the shear turbulence) and the LC in determining scalar flux from the airside to the waterside and vertical scalar transport beneath. The C-L force consists of the cross product between the Stokes drift velocity (induced by the interface waves) and the flow vorticity. It is observed that Stokes drift shear intensifies the shear streaks (with respect to flow without wave effects) leading to enhanced scalar flux at the air-water interface. LC leads to increased vertical scalar transport at depths below the interface.

Sunday, November 20, 2016 8:00AM - 9:57AM –

Session A15 Bio: Cardiovascular Interventions E143/144 - Alison Marsden, Stanford University

8:00AM A15.00001 Patient-specific modeling of the Assisted Bidirectional Glenn (ABG)¹, JESSICA SHANG, University of Rochester, MAHDI ESMAILY-MOGHAMAD, Stanford University, RICHARD FIGLIOLA, Clemson University, TAIN-YEN HSIA, Great Ormond Street Hospital, ALISON MARSDEN, Stanford University — The Assisted Bidirectional Glenn (ABG) is proposed as an early-stage palliative procedure for single ventricle neonates. The ABG augments the pulmonary flow of the Bidirectional Glenn (BDG) with a secondary high-velocity flow through a nozzle-like shunt between the innominate artery and the superior vena cava (SVC). The ABG would provide a superior cavopulmonary connection than the systemic-pulmonary shunt that is typically employed as a stage-I procedure (e.g., a modified Blalock-Taussig shunt) and would address the low pulmonary flow associated with the BDG. Following simulations in vitro and in silico that show the ABG successfully increased pulmonary flows in idealized models, we implemented the ABG in several patient-specific models coupled to a lumped parameter network tuned to clinical values for each patient. The ABG performed similarly across different patients; compared to the BDG, the pulmonary flow increased ~20% with a similar increase in the SVC pressure. The performance of the ABG was the most sensitive to nozzle outlet area, compared to nozzle inlet area and location of the shunt anastomosis. The study verified that the ABG benefits a range of patients and identified key parameters for further optimization of the ABG.

¹Stanford Cardiovascular Institute NIH T32

8:13AM A15.00002 Shape Optimization of the Assisted Bi-directional Glenn surgery for stage-1 single ventricle palliation¹, AAKAANSH VERMA, Stanford University, JESSICA SHANG, University of Rochester, MAHDI ESMAILY-MOGHAMAD, Stanford University, KWAI WONG, University of Tennessee, Knoxville, ALISON MARSDEN, Stanford University — Babies born with a single functional ventricle typically undergo three open-heart surgeries starting as neonates. The first of these stages (BT shunt or Norwood) has the highest mortality rates of the three, approaching 30%. Proceeding directly to a stage-2 Glenn surgery has historically demonstrated inadequate pulmonary flow (PF) & high mortality. Recently, the Assisted Bi-directional Glenn (ABG) was proposed as a promising means to achieve a stable physiology by assisting the PF via an 'ejector pump' from the systemic circulation. We present preliminary parametrization and optimization results for the ABG geometry, with the goal of increasing PF. To limit excessive pressure increases in the Superior Vena Cava (SVC), the SVC pressure is included as a constraint. We use 3-D finite element flow simulations coupled with a single ventricle lumped parameter network to evaluate PF & the pressure constraint. We employ a derivative free optimization method—the Surrogate Management Framework, in conjunction with the OpenDIEI framework to simulate multiple simultaneous evaluations. Results show that nozzle diameter is the most important design parameter affecting ABG performance. The application of these results to patient specific situations will be discussed.

¹This work was supported by an NSF CAREER award (OCI1150184) and by the XSEDE National Computing Resource.

8:26AM A15.00003 Modeling the Mitral Valve¹, ALEXANDER KAISER, Courant Institute, New York University — The mitral valve is one of four valves in the human heart. The valve opens to allow oxygenated blood from the lungs to fill the left ventricle, and closes when the ventricle contracts to prevent backflow. The valve is composed of two fibrous leaflets which hang from a ring. These leaflets are supported like a parachute by a system of strings called chordæ tendineæ. In this talk, I will describe a new computational model of the mitral valve. To generate geometry, general information comes from classical anatomy texts and the author's dissection of porcine hearts. An MRI image of a human heart is used to locate the tips of the papillary muscles, which anchor the chordæ tendineæ, in relation to the mitral ring. The initial configurations of the valve leaflets and chordæ tendineæ are found by solving an equilibrium elasticity problem. The valve is then simulated in fluid (blood) using the immersed boundary method over multiple heart cycles in a model valve tester. We aim to identify features and mechanisms that influence or control valve function.

¹Support from National Science Foundation, Graduate Research Fellowship Program, grant DGE 1342536.

8:39AM A15.00004 Effect of mitral valve prosthesis design and orientation on intraventricular flow and blood stasis, KAREN MAY-NEWMAN, J CAMPOS, R MONTES, V RAMESH, J MOON, C REIDER, San Diego State University, P MARTINEZ-LEGAZPI, J BERMEJO, Hospital Gregorio Marañón, LORENZO ROSSINI, JUAN C DEL ALAMO, UC San Diego — Abnormal blood flow patterns are linked with thromboembolism (TE), especially in the presence of medical devices such as mitral valve prostheses (MVP). We performed PIV on a customized silicone left ventricle (LV) in a mock circulatory loop. We measured the velocity field in the long-axis midplane for 3 different MVP: a porcine bioprosthesis (BP), a tilting disk valve in two orientations: towards the LV lateral (TD-L) or the anterior wall (TD-A), and a bileaflet valve with anti-anatomical orientation (BL). Diastolic LV vortices were tracked and related to measures of blood stasis based on LV residence time. The BP and the TD-L produced flow patterns similar to those measured in patients. The TD-A showed a complete reversal of diastolic vortices. The BL design had increased apical blood stasis, which may lead to increased TE risk.

8:52AM A15.00005 Fluid-structure interaction in the left ventricle of the human heart coupled with mitral valve, VALENTINA MESCHINI, Gran Sasso Science Institute, MARCO DONATO DE TULLIO, Politecnico di Bari, GIORGIO QUERZOLI, Università di Cagliari, ROBERTO VERZICCO, Università di Roma Tor Vergata and University of Twente — In this paper Direct Numerical Simulations (DNS), implemented using a fully fluid-structure interaction model for the left ventricle, the mitral valve and the flowing blood, and laboratory experiments are performed in order to cross validate the results. Moreover a parameter affecting the flow dynamics is the presence of a mitral valve. We model two cases, one with a natural mitral valve and another with a prosthetic mechanical one. Our aim is to understand their different effects on the flow inside the left ventricle in order to better investigate the process of valve replacement. We simulate two situations, one of a healthy left ventricle and another of a failing one. While in the first case the flow reaches the apex of the left ventricle and washout the stagnant fluid with both mechanical and natural valve, in the second case the disturbance generated by the mechanical leaflets destabilizes the mitral jet, thus further decreasing its capability to penetrate the ventricular region and originating heart attack or cardiac pathologies in general.

9:05AM A15.00006 Platelet activation through a Bi-leaflet mechanical heart valve¹

, MOHAMMADALI HEDAYAT, IMAN BORAZJANI, The State University of New York at Buffalo — Platelet activation is one of the major drawbacks of the Mechanical Heart Valves (MHVs) which can increase the risk of thrombus formation in patients. The platelet activation in MHVs can be due to the abnormal shear stress during the systole, the backward leakage flow during the diastole, and the flow through the hinge region. We investigate the contribution of each of the above mechanism to the activation of platelets in MHVs by performing simulations of the flow through the MHV and in the hinge region. The large scale heart valve simulations are performed in a straight aorta using a sharp interface curvilinear immersed boundary method along with a strong-coupling algorithm under physiological flow conditions. In addition, in order to perform the simulation of hinge region the flow field boundary conditions are obtained from the largescale simulations during a whole cardiac cycle. In order to investigate the role of hinge flow on platelet activation in MHVs, a 23mm St. Jude Medical Regent valve hinge with three different gap sizes is tested along with different platelet activation models to ensure the consistency of our results with different activation models. We compare the platelet activation of the hinge region against the bulk of the flow during one cardiac cycle.

¹This work is supported by the American Heart Association grant 13SDG17220022, and the computational resources were partly provided by Center for Computational Research (CCR) at University at Buffalo.

9:18AM A15.00007 Influence of surgical implantation angle of left ventricular assist device outflow graft and management of aortic valve opening on the risk of stroke in heart failure patients , V. KESHAV CHIVUKULA, PATRICK MCGAH, U. Washington, ANTHONY PRISCO, U. Minnesota,

JENNIFER BECKMAN, Div. Cardiology, NANUSH MOKADAM, Div. Cardiothoracic Surgery, CLAUDIOUS MAHR, Div. Cardiology, ALBERTO ALISEDA, U. Washington — Flow in the aortic vasculature may impact stroke risk in patients with left ventricular assist devices (LVAD) due to severely altered hemodynamics. Patient-specific 3D models of the aortic arch and great vessels were created with an LVAD outflow graft at 45, 60 and 90° from centerline of the ascending aorta, in order to understand the effect of surgical placement on hemodynamics and thrombotic risk. Intermittent aortic valve opening (once every five cardiac cycles) was simulated and the impact of this residual native output investigated for the potential to wash out stagnant flow in the aortic root region. Unsteady CFD simulations with patient-specific boundary conditions were performed. Particle tracking for 10 cardiac cycles was used to determine platelet residence times and shear stress histories. Thrombosis risk was assessed by a combination of Eulerian and Lagrangian metrics and a newly developed thrombogenic potential metric. Results show a strong influence of LVAD outflow graft angle on hemodynamics in the ascending aorta and consequently on stroke risk, with a highly positive impact of aortic valve opening, even at low frequencies. Optimization of LVAD implantation and management strategies based on patient-specific simulations to minimize stroke risk will be presented

9:31AM A15.00008 Three-dimensional Computational Simulation and In-vitro Experiments for Assessing Radiocephalic Wrist Arteriovenous Fistulas , RYUNGEUN SONG,

Sungkyunkwan University, SUN CHEOL PARK, HYUN KYU KIM, Uijeongbu St' Mary's Hospital, JINKEE LEE, Sungkyunkwan University — A radio-cephalic arteriovenous fistula (RC-AVF) is the best choice for achieving vascular access (VA) for hemodialysis, but this AVF has high rates of early failure depending on the vessel condition. The high wall shear stress (WSS) contributes to VA failures due to plaque rupture, thrombosis, etc. Thus, we have used a low-Re k-ε turbulence based CFD model combined with an in-vitro experimental approach to evaluate the WSS distribution and to minimize its effects under several conditions. The properties considered in this study were non-Newtonian flow characteristics, complete cardiac pulse cycle, and distention of blood vessels. The computational domain was designed for arteriovenous end-to-side anastomosis based on anastomosis angles of 45, 90, and 135. For experiment the digital domains were converted into 3D artificial RC-AVF via poly(dimethylsiloxane) (PDMS) and 3D printing technology. The micro-particle image velocimetry (μ -PIV) was used to measure the velocity field within the artificial blood vessel. The results showed that the largest anastomosis angle (135) resulted in lower WSS, which would help reduce AVF failures. This research would provide the future possibility of using the proposed method to reduce in-vivo AVF failure for various conditions in each patient.

9:44AM A15.00009 Wall Shear Stress Restoration in Dialysis Patients Venous Stenosis: Elucidation via 3D CFD and Shape Optimization¹ , S. M. JAVID MAHMOUDZADEH AKHERAT,

PhD, KEVIN CASSEL, MARY HAMMES, Professor, MICHAEL BOGHOSIAN, PhD, ILLINOIS INSTITUTE OF TECHNOLOGY TEAM, UNIVERSITY OF CHICAGO TEAM — Venous stenosis developed after the growth of excessive neointimal hyperplasia (NH) in chronic dialysis treatment is a major cause of mortality in renal failure patients. It has been hypothesized that the low wall shear stress (WSS) triggers an adaptive response in patients' venous system that through the growth of neointimal hyperplastic lesions restores WSS and transmural pressure, which also regulates the blood flow rate back to physiologically acceptable values which is violated by dialysis treatment. A strong coupling of three-dimensional CFD and shape optimization analyses were exploited to elucidate and forecast this adaptive response which correlates very well topographically with patient-specific clinical data. Based on the framework developed, a medical protocol is suggested to predict and prevent dialysis treatment failure in clinical practice.

¹Supported by the National Institute of Diabetes and Digestive and Kidney Diseases of the National Institutes of Health (R01 DK90769).

Sunday, November 20, 2016 8:00AM - 9:57AM –

Session A16 Drops: Condensation and Evaporation D133/134 - Vladimir Ajaev, Southern Methodist University

8:00AM A16.00001 Interaction of levitating microdroplets with moist air flow in the contact line region¹ , OLEG KABOV, DMITRY ZAITSEV, DMITRY KIRICHENKO, Institute of Thermophysics, Russia,

VLADIMIR AJAEV, Southern Methodist University — Self-organization of levitating microdroplets of condensate over a liquid-gas interface has been observed in several recent experiments involving evaporation at high heat fluxes, although the nature of thin phenomenon is still not completely understood. We conduct experimental investigation of behavior of an ordered array of levitating microdroplets as it approaches a region of intense evaporation near the contact line. Interaction of the array with the local highly non-uniform gas flow is shown to result in the break-up of the pattern. Furthermore, our experimental set-up provides a unique tool for investigation of the Stefan flow originating near the contact line by using microdroplets as tracers. Local gas flow velocities near the contact line are obtained based on trajectories of the droplets.

¹This work was supported by the Russian Science Foundation (project No. 14-19-01755)

8:13AM A16.00002 Analysis of water microdroplet condensation on silicon surfaces

, TAKUYA HONDA, KENYA FUJIMOTO, YUTA YOSHIMOTO, Univ of Tokyo, KATSUO MOGI, Tokyo Inst. of Tech., IKUYA KINEFUCHI, Univ of Tokyo, YASUHIKO SUGII, Tokyo Inst. of Tech., SHU TAKAGI, Univ of Tokyo, UNIV. OF TOKYO TEAM, TOKYO INST. OF TECH. TEAM — We observed the condensation process of water microdroplets on flat silicon (100) surfaces by means of the sequential visualization of the droplets using an environmental scanning electron microscope. As previously reported for nanostructured surfaces, the condensation process of water microdroplets on the flat silicon surfaces also exhibits two modes: the constant base (CB) area mode and the constant contact angle (CCA) mode. In the CB mode, the contact angle increases with time while the base diameter is constant. Subsequently, in the CCA mode, the base diameter increases with time while the contact angle remains constant. The dropwise condensation model regulated by subcooling temperature does not reproduce the experimental results. Because the subcooling temperature is not constant in the case of a slow condensation rate, this model is not applicable to the condensation of the long time scale ("several tens of minutes"). The contact angle of water microdroplets (\sim several μm) tended to be smaller than the macro contact angle. Two hypotheses are proposed as the cause of small contact angles: electrowetting and the coalescence of sub- μm water droplets.

8:26AM A16.00003 Condensation on Slippery Asymmetric Bumps

, KYOO-CHUL PARK, Harvard University, PHILSEOK KIM, SLIPS Technologies, Inc., JOANNA AIZENBERG, Harvard University — Controlling dropwise condensation by designing surfaces that enable droplets to grow rapidly and be shed as quickly as possible is fundamental to water harvesting systems, thermal power generation, distillation towers, etc. However, cutting-edge approaches based on micro/nanoscale textures suffer from intrinsic trade-offs that make it difficult to optimize both growth and transport at once. Here we present a conceptually different design approach based on principles derived from Namib desert beetles, cacti, and pitcher plants that synergistically couples both aspects of condensation and outperforms other synthetic surfaces. Inspired by an unconventional interpretation of the role of the beetle's bump geometry in promoting condensation, we show how to maximize vapor diffusion flux at the apex of convex millimetric bumps by optimizing curvature and shape. Integrating this apex geometry with a widening slope analogous to cactus spines couples rapid drop growth with fast directional transport, by creating a free energy profile that drives the drop down the slope. This coupling is further enhanced by a slippery, pitcher plant-inspired coating that facilitates feedback between coalescence-driven growth and capillary-driven motion. We further observe an unprecedented six-fold higher exponent in growth rate and much faster shedding time compared to other surfaces. We envision that our fundamental understanding and rational design strategy can be applied to a wide range of phase change applications.

8:39AM A16.00004 ABSTRACT WITHDRAWN —

8:52AM A16.00005 Nucleation pressure threshold in acoustic droplet vaporization¹

, CHRISTOPHER MILES, CHARLES DOERING, OLIVER KRIPFGANS, University of Michigan - Ann Arbor — We combine classical nucleation theory with superharmonic focusing to predict necessary pressures to induce nucleation in acoustic droplet vaporization. We show that linear acoustics is a valid approximation to leading order when particle displacements in the sound field are small relative the radius of the droplet. This is done by perturbation analysis of an axisymmetric compressible inviscid flow about a droplet with small surface perturbations relative to the mean radius subjected to an incoming ultrasonic wave. The necessary nucleation pressure threshold inside the droplet is calculated to be -9.33 ± 0.30 MPa for typical experimental parameters by employing results from *classical homogeneous nucleation theory*. As a result we are able to predict if a given incident pressure waveform will induce nucleation.

¹This research was supported by the Rackham Merit Fellowship, the University of Michigan Physics department, the University of Michigan's MCubed program, and NSF awards PHY-1205219 and DMS-1515161.

9:05AM A16.00006 Evaporation dynamics of water droplets on inclined surfaces¹

, JIN YOUNG KIM, SKKU Advanced Institute of Nanotechnology (SAINT), Sungkyunkwan University, IN GYU HWANG, School of Advanced Materials Science and Engineering, Sungkyunkwan University, BYUNG MOOK WEON², School of Advanced Materials Science and Engineering, SKKU Advanced Institute of Nanotechnology (SAINT), Sungkyunkwan University — When a water droplet is gently placed on a flat substrate, particularly which is tilted at an inclined angle, usually there are advancing and receding angles inside the droplet formed by inclination under gravitational force. Evaporation dynamics of an nonspherical inclined droplet at inclinations would deviate from that of a spherical droplet. Here we study on evaporation dynamics rates of inclined droplets by measuring mass changes with time and their lifetimes. We find that the lifetime of an evaporating inclined droplets becomes longer as the gravitational influence becomes stronger. The lifetime depends on the pinning-depinning transitions and the depinning onset times, which are changed by the gravitational influence. This The dependence inclination-induced evaporation behavior would be useful important in understanding evaporation dynamics of inclined droplets.

¹This research was supported by Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Education (NRF-2016R1D1A1B01007133).

²Email: bmweon@skku.edu

9:18AM A16.00007 ABSTRACT WITHDRAWN —

9:31AM A16.00008 Surface temperature measurements of a levitated water drop during laser irradiation¹

, CODY BROWNELL, TIMOTHY TRACEY, United States Naval Academy — Simulation of high energy laser propagation and scattering in the maritime environment is problematic, due to the high likelihood of turbulence, fog, and rain or sea spray within the beam path. Laser interactions with large water drops (diameters of approximately 1-mm), such as those found in a light rain, have received relatively less attention. In this regime a high energy laser will rapidly heat and vaporize a water drop as it traverses the beam path, but the exact heating / vaporization rate, its dependence on impurities, and ancillary effects on the drop or surroundings are unclear. In this work we present surface temperature measurements of a water drop obtained using a FLIR IR camera. The drop is acoustically levitated, and subject to a continuous wave laser with a wavelength of 1070-nm and a mean irradiance of approximately 500 W/cm^2 . These measurements show that the steady-state surface temperature of the drop is well below the saturation temperature, yet based on the time history of the drop volume vaporization begins almost immediately upon laser strike. Inferences on the turbulence characteristics within the drop are also made from measurements of the fluctuations in the surface temperature.

¹Supported by ONR, HEL-JTO, and USNA Trident Scholar Program

9:44AM A16.00009 Evaporation of particle-laden droplets on a superhydrophobic surface, MASOUD BIGDELI, PEICHUN AMY TSAI, University of Alberta — We experimentally investigated the evaporation dynamics of water droplets suspended with minute particles of varying concentrations on a superhydrophobic surface. The contact angle, diameter, and height of the droplets decreased during the evaporation process. For pure water, the droplet went through a wetting transition from a partial wetting (Cassie-Baxter), with a large contact angle ($> 140^\circ$), to completely wetting (Wenzel) state, with a small contact angle. Unlike pure water, the nanofluid droplets maintain high contact angles ($> 100^\circ$) during evaporation. We found that the contact line was pinned, and an increase (10%) in the weight fraction of nanoparticles led to a remarkable 40% decrease in the total drying time. The nanofluid droplets left donut-shaped drying patterns. In these final drying structures, a shrinkage of the droplet height and base diameter was observed for nanofluids with lower concentrations. The results show that droplet evaporation rate and deposit pattern depend on the concentration of nanoparticles, implying the crucial influences of water evaporation and particle migration dynamics and time-scales.

Sunday, November 20, 2016 8:00AM - 9:57AM –

Session A17 Reacting Flows: DNS D131 - Jacqueline Chen, Sandia National Laboratories

8:00AM A17.00001 Direct Numerical Simulation of a Cavity-Stabilized Ethylene/Air

Premixed Flame¹, JACQUELINE CHEN, ADITYA KONDURI, HEMANTH KOLLA, Sandia National Laboratories, ANDREAS RAUCH, HARSHA CHELLIAH, University of Virginia — Cavity flame holders have been shown to be important for flame stabilization in scramjet combustors. In the present study the stabilization of a lean premixed ethylene/air flame in a rectangular cavity at thermo-chemical conditions relevant to scramjet combustors is simulated using a compressible reacting multi-block direct numerical simulation solver, S3D, incorporating a 22 species ethylene-air reduced chemical model. The fuel is premixed with air to an equivalence ratio of 0.4 and enters the computational domain at Mach numbers between 0.3 and 0.6. An auxiliary inert channel flow simulation is used to provide the turbulent velocity profile at the inlet for the reacting flow simulation. The detailed interaction between intense turbulence, nonequilibrium concentrations of radical species formed in the cavity and mixing with the premixed main stream under density variations due to heat release rate and compressibility effects is quantified. The mechanism for flame stabilization is quantified in terms of relevant non-dimensional parameters, and detailed analysis of the flame and turbulence structure will be presented.

¹We acknowledge the sponsorship of the AFOSR-NSF Joint Effort on Turbulent Combustion Model Assumptions and the DOE Office of Basic Energy Sciences, Division of Chemical Sciences, Geosciences, and Biosciences.

8:13AM A17.00002 DNS of a turbulent, self-igniting n-dodecane / air jet , GIULIO

BORGHESI, JACQUELINE CHEN, Sandia National Laboratories, Livermore, CA — A direct numerical simulation of a turbulent, self-igniting temporal jet between n-dodecane and diluted air at $p=25$ bar has been conducted to clarify certain aspects of diesel engine combustion. The thermodynamics conditions were selected to result in a two-stage ignition event, in which low- and high-temperature chemical reactions play an equally important role during the ignition process. Jet parameters were tuned to yield a target ignition Damköhler number of 0.4, a value representative of conditions found in diesel spray flames. Chemical reactions were described by a 35-species reduced mechanism, including both the low- and high-temperature reaction pathways of n-dodecane. The present work focuses on the influence of low-temperature chemistry on the overall ignition transient. We also study the structure of the flames formed at the end of the autoignition transient. Recent studies on diluted dimethyl ether / air flames at pressure and temperature conditions similar to those investigated in this work revealed the existence of tetra- and penta-brachial flames, and it is of interest to determine whether similar flame structures also exist when diesel-like fuels are used.

8:26AM A17.00003 Large scale Direct Numerical Simulation of premixed turbulent jet flames at high Reynolds number, ANTONIO ATTILI, RWTH Aachen University, STEFANO LUCA, King Abdullah Univ of Sci & Tech (KAUST), ERMANNO LO SCHIAVO, Sapienza University of Rome, FABRIZIO BISSETTI, University of Texas at Austin, FRANCESCO CRETA, Sapienza University of Rome — A set of direct numerical simulations of turbulent premixed jet flames at different Reynolds and Karlovitz numbers is presented. The simulations feature finite rate chemistry with 16 species and 73 reactions and up to 22 Billion grid points. The jet consists of a methane/air mixture with equivalence ratio $\phi = 0.7$ and temperature varying between 500 and 800 K. The temperature and species concentrations in the coflow correspond to the equilibrium state of the burnt mixture. All the simulations are performed at 4 atm. The flame length, normalized by the jet width, decreases significantly as the Reynolds number increases. This is consistent with an increase of the turbulent flame speed due to the increased integral scale of turbulence. This behavior is typical of flames in the thin-reaction zone regime, which are affected by turbulent transport in the preheat layer. Fractal dimension and topology of the flame surface, statistics of temperature gradients, and flame structure are investigated and the dependence of these quantities on the Reynolds number is assessed.

8:39AM A17.00004 DNS investigation of differential-diffusion effects on temporally evolving turbulent diffusion flames¹, ANTONIO ALMAGRO, MANUEL GARCIA-VILLALBA, OSCAR FLORES, Universidad Carlos III de Madrid, ANTONIO L SANCHEZ, University of California San Diego — The peak temperature of nonpremixed flames is known to have a profound effect on kinetically controlled processes with a strong temperature dependence, such as strain-induced extinction and NOx production. Here, the influence of differential diffusion on the flame temperature in diffusion-controlled combustion is investigated by direct numerical simulations of a turbulent diffusion flame in a temporally evolving mixing layer for non-unity Lewis numbers of the fuel. The problem is formulated in the limit of infinitely fast combustion in terms of Shvab-Zeldovich conserved scalars, not changed directly by the reactions, obtained through chemistry-free linear combinations of the temperature and reactant mass fractions. A previously developed low-Mach-number code is used in the numerical integrations, which consider values of the thermochemical parameters – characterizing the exothermicity and stoichiometry of diffusion-controlled combustion – and fuel Lewis number typical of hydrogen-air and hydrocarbon-air flames. The results of the simulations are used to assess the effect of turbulence and fuel diffusivity on the flame response.

¹This work was funded by the Spanish MCININ under project CSD2010-00011. The computational resources were provided by the XSEDE program, supported by NSF grant number ACI-1053575.

8:52AM A17.00005 DNS of High Pressure Supercritical Combustion, SHAO TENG CHONG, VENKATRAMAN RAMAN, Univ of Michigan - Ann Arbor — Supercritical flows have always been important to rocket motors, and more recently to aircraft engines and stationary gas turbines. The purpose of the present study is to understand effects of differential diffusion on reacting scalars using supercritical isotropic turbulence. Focus is on fuel and oxidant reacting in the transcritical region where density, heat capacity and transport properties are highly sensitive to variations in temperature and pressure. Reynolds and Damköhler number vary as a result and although it is common to neglect differential diffusion effects if Re is sufficiently large, this large variation in temperature with heat release can accentuate molecular transport differences. Direct numerical simulations (DNS) for one step chemistry reaction between fuel and oxidizer are used to examine the differential diffusion effects. A key issue investigated in this paper is if the flamelet progress variable approach, where the Lewis number is usually assumed to be unity and constant for all species, can be accurately applied to simulate supercritical combustion.

9:05AM A17.00006 LES-inspired forcing technique for DNS of turbulent premixed flames, CHANDRU DHANDAPANI, Graduate Aerospace Laboratories, California Institute of Technology, GUILLAUME BLANQUART, Mechanical Engineering Department, California Institute of Technology — Direct numerical simulations (DNS) of high Karlovitz number (Ka) flames have been performed extensively in an inflow/outflow configuration, but in the absence of mean shear. Without a mean shear to sustain turbulence, the turbulent kinetic energy decays in the domain. Hence, a turbulence forcing has been used in previous simulations to emulate the missing shear effects. Rather than using an arbitrary forcing, the current study uses a source term for this turbulence forcing obtained from the results of previously performed large eddy simulations (LES) of a practical turbulent jet flame. The pseudo-shear term used for this turbulence forcing is linear and takes the form $A_{ij}u_j$ in the momentum equation, such that the source term is proportional to the velocity fluctuations. Different forms of the proportionality matrix A_{ij} are considered, including a scalar matrix δ_{ij} . DNS of high Ka n-heptane air flames are performed with the new forcing and the energy spectrum is calculated. This energy spectrum is combined with that obtained from the LES and compared with results from previously performed experiments of turbulent jet flames.

9:18AM A17.00007 Direct Numerical Simulation of Combustion Using Principal Component Analysis¹, OPEOLUWA OWOYELE, TAREK ECHEKKI, North Carolina State University — We investigate the potential of accelerating chemistry integration during the direct numerical simulation (DNS) of complex fuels based on the transport equations of representative scalars that span the desired composition space using principal component analysis (PCA). The transported principal components (PCs) offer significant potential to reduce the computational cost of DNS through a reduction in the number of transported scalars, as well as the spatial and temporal resolution requirements. The strategy is demonstrated using DNS of a premixed methane-air flame in a 2D vortical flow and is extended to the 3D geometry to further demonstrate the computational efficiency of PC transport. The PCs are derived from *a priori* PCA of a subset of the full thermo-chemical scalars' vector. The PCs' chemical source terms and transport properties are constructed and tabulated in terms of the PCs using artificial neural networks (ANN). Comparison of DNS based on a full thermo-chemical state and DNS based on PC transport based on 6 PCs shows excellent agreement even for species that are not included in the PCA reduction. The transported PCs reproduce some of the salient features of strongly curved and strongly strained flames. The 2D DNS results also show a significant reduction of two orders of magnitude in the computational cost of the simulations, which enables an extension of the PCA approach to 3D DNS under similar computational requirements.

¹This work was supported by the National Science Foundation grant DMS- 1217200.

9:31AM A17.00008 ABSTRACT WITHDRAWN —

9:44AM A17.00009 Conditional budgets of second-order statistics in nonpremixed and premixed turbulent combustion, JONATHAN F. MACART, TEMISTOCLE GRENGA, MICHAEL E. MUELLER, Princeton Univ — Combustion heat release modifies or introduces a number of new terms to the balance equations for second-order turbulence statistics (turbulent kinetic energy, scalar variance, etc.) compared to incompressible flow. A major modification is a significant increase in viscosity and dissipation in the high-temperature combustion products, but new terms also appear due to density variation and gas expansion (dilatation). Previous scaling analyses have hypothesized that dilatation effects are important in turbulent premixed combustion but are unimportant in turbulent nonpremixed combustion. To explore this hypothesis, a series of DNS calculations have been performed in the low Mach number limit for spatially evolving turbulent planar jet flames of hydrogen and air in both premixed and nonpremixed configurations. Unlike other studies exploring the effects of heat release on turbulence, the turbulence is not forced, and detailed chemical kinetics are used to describe hydrogen-air combustion. Budgets for second-order statistics are computed conditioned on progress variable in the premixed flame and on mixture fraction in the nonpremixed flame in order to locate regions with respect to the flame structure where dilatation effects are strongest.

Sunday, November 20, 2016 8:00AM - 9:57AM — Session A18 Flow Instability: Boundary Layers D135 - Joseph Kuehl, Baylor University

8:00AM A18.00001 Implications of a wavepacket formulation for the nonlinear parabolized stability equations to hypersonic boundary layers¹, JOSEPH KUEHL, Baylor University — The parabolized stability equations (PSE) have been developed as an efficient and powerful tool for studying the stability of advection-dominated laminar flows. In this work, a new "wavepacket" formulation of the PSE is presented. This method accounts for the influence of finite-bandwidth-frequency distributions on nonlinear stability calculations. The methodology is motivated by convolution integrals and is found to appropriately represent nonlinear energy transfer between primary modes and harmonics, in particular nonlinear feedback, via a "nonlinear coupling coefficient." It is found that traditional discrete mode formulations overestimate nonlinear feedback by approximately 70%. This results in smaller maximum disturbance amplitudes than those observed experimentally. The new formulation corrects this overestimation, accounts for the generation of side lobes responsible for spectral broadening and results in disturbance saturation amplitudes consistent with experiment. A Mach 6 flared-cone example is presented.

¹Support from the AFOSR Young Investigator Program via Grant FA9550-15-1-0129 is gratefully acknowledged.

8:13AM A18.00002 Optimal Disturbances in Spatially Developing Turbulent Boundary Layers¹, TIMOTHY DAVIS, FARRUKH ALVI, Florida State University — Perturbations leading to optimal energy growth in zero pressure gradient turbulent boundary layers are computed. A spatial formulation is adopted to account for the slow development of the turbulent mean flow. Optimals are computed using both an eddy viscosity and quasi-laminar assumption with initial focus given towards steady, streamwise elongated streaks. Results using the eddy viscosity qualitatively agree well with previous temporal analyses, identifying both inner and outer scaled peaks in the energy amplification. Significant differences, however, are noted in the large scale outer structures with spanwise wavelengths $\sim 3\delta$. The eddy viscosity is further shown to have a substantial effect on the optimal structures and, in general, better agreement with experimental observation is found using the quasi-laminar approach. In this case, the optimal structures are found to scale with the geometric mean of the logarithmic layer in the mean flow. Propagating modes are also considered, achieving large energy amplifications when the disturbance phase speed approaches the local mean. The most energetic streamwise scales and optimal structures are found to agree well with natural structures observed in turbulent boundary layers.

¹Partial support by FCAAP, ARO and NSF

8:26AM A18.00003 Real gas effects on receptivity to kinetic fluctuations¹ , ANATOLI

TUMIN, LUKE EDWARDS, The University of Arizona — Receptivity of high-speed boundary layers is considered within the framework of fluctuating hydrodynamics where stochastic forcing is introduced through fluctuating shear stress and heat flux stemming from kinetic fluctuations (thermal noise). The forcing generates unstable modes whose amplification downstream and may lead to transition. An example of high-enthalpy (16.53 MJ/kg) boundary layer at relatively low wall temperatures ($T_w = 1000 \text{ K} - 3000 \text{ K}$), free stream temperature ($T_e = 834 \text{ K}$), and low pressure (0.0433 atm) is considered. Dissociation at the chosen flow parameters is still insignificant. The stability and receptivity analyses are carried out using a solver for calorically perfect gas with effective Prandtl number and specific heats ratio. The receptivity phenomenon is unchanged by the inclusion of real gas effects in the mean flow profiles. This is attributed to the fact that the mechanism for receptivity to kinetic fluctuations is localized near the upper edge of the boundary layer. Amplitudes of the generated wave packets are larger downstream in the case including real gas effects. It was found that spectra in both cases include supersonic second Mack unstable modes despite the temperature ratio $T_w/T_e > 1$.

¹Supported by AFOSR

8:39AM A18.00004 Modeling boundary-layer transition in DNS and LES using Parabolized Stability Equations¹ , ADRIAN LOZANO-DURAN, M. J. PHILIPP HACK, PARVIZ MOIN, Center for Turbulence Research, Stanford University

— The modeling of the laminar region and the prediction of the point of transition remain key challenges in the numerical simulation of boundary layers. The issue is of particular relevance for wall-modeled large eddy simulations which require 10 to 100 times higher grid resolution in the thin laminar region than in the turbulent regime (Slotnick *et al.*, NASA/CR-2014-218178, 2014). Our study examines the potential of the nonlinear parabolized stability equations (PSE) to provide an accurate, yet computationally efficient treatment of the growth of disturbances in the pre-transitional flow regime. The PSE captures the nonlinear interactions that eventually induce breakdown to turbulence, and can as such identify the onset of transition without relying on empirical correlations. Since the local PSE solution at the point of transition is the solution of the Navier-Stokes equations, it provides a natural inflow condition for large eddy and direct simulations by avoiding unphysical transients. We show that in a classical H-type transition scenario, a combined PSE/DNS approach can reproduce the skin-friction distribution obtained in reference direct numerical simulations. The computational cost in the laminar region is reduced by several orders of magnitude.

¹Funded by the Air Force Office of Scientific Research

8:52AM A18.00005 Convective and global stability analysis of a Mach 5.8 boundary layer grazing a compliant surface¹ , FABIAN DETTENRIEDER, DANIEL BODONY, University of Illinois at Urbana-Champaign

— Boundary layer transition on high-speed vehicles is expected to be affected by unsteady surface compliance. The stability properties of a Mach 5.8 zero-pressure-gradient laminar boundary layer grazing a nominally-flat thermo-mechanically compliant panel is considered. The linearized compressible Navier-Stokes equations describe small amplitude disturbances in the fluid while the panel deformations are described by the Kirchhoff-Love plate equation and its thermal state by the transient heat equation. Compatibility conditions that couple disturbances in the fluid to those in the solid yield simple algebraic and robin boundary conditions for the velocity and thermal states, respectively. A local convective stability analysis shows that the panel can modify both the first and second Mack modes when, for metallic-like panels, the panel thickness exceeds the lengthscale $\delta_{99} Re_x^{-0.5}$. A global stability analysis, which permits finite panel lengths with clamped-clamped boundary conditions, shows a rich eigenvalue spectrum with several branches. Unstable modes are found with streamwise-growing panel deformations leading to Mach wave-type radiation. Stable global modes are also found and have distinctly different panel modes but similar radiation patterns.

¹Air Force Office of Scientific Research

9:05AM A18.00006 Some Insights on Roughness Induced from DNS and Experiments¹ , SAIKISHAN SURYANARAYANAN², IFEOLUWA IE

Texas, Austin, GARRY BROWN, Princeton University — We study the receptivity and subsequent evolution of the boundary layer on a flat plate to single and multiple discrete roughness elements (DRE) using a combination of direct numerical simulation (DNS) and wind tunnel channel flow visualization experiments. We examine the transition caused by a single DRE and demonstrate the effect of an appropriately designed second DRE in both DNS and experiments. The different phases of transition and transition numbers based on roughness height and boundary layer thickness are investigated. The underlying mechanisms of transition control are understood by examining detailed vorticity flux balances. Connections are also made to recent work on streak instability. A unified picture is sought from a parametric study of different DRE dimensions and locations. The relevance of the observations and understanding derived from this study to controlling transition caused by discrete roughness on aircraft wings is discussed.

¹supported by AFOSR FA9550-15-1-0345

²Postdoctoral fellow, Aerospace Engineering

³Undergraduate student, Aerospace Engineering

⁴Professor, Aerospace Engineering

Fellow of the American Physical Society

9:18AM A18.00007 On The Stability Of Model Flows For Chemical Vapour Deposition , ROBERT MILLER, Univ of Leicester

— The flow in a chemical vapour deposition (CVD) reactor is assessed. The reactor is modelled as a flow over an infinite-radius rotating disk, where the mean flow and convective instability of the disk boundary layer are measured. Temperature-dependent viscosity and enforced axial flow are used to model the steep temperature gradients present in CVD reactors and the pumping of the gas towards the disk, respectively. Increasing the temperature-dependence parameter of the fluid viscosity (ε) results in an overall narrowing of the fluid boundary layer. Increasing the axial flow strength parameter (T_s) accelerates the fluid both radially and axially, while also narrowing the thermal boundary layer. It is seen that when both effects are imposed, the effects of axial flow generally dominate those of the viscosity temperature dependence. A local stability analysis is performed and the linearized stability equations are solved using a Galerkin projection in terms of Chebyshev polynomials. The neutral stability curves are then plotted for a range of ε and T_s values. Preliminary results suggest that increasing T_s has a stabilising effect on both type I and type II stationary instabilities, while small increases in ε results in a significant reduction to the critical Reynolds number.

9:31AM A18.00008 Exact coherent structures for the turbulent cascade, BRUNO ECKHARDT, Philipps-Universität Marburg, STEFAN ZAMMERT, TU Delft — The exact coherent structures that are connected with the transition to turbulence in interior flows usually extend across the full height of the domain. Using exact coherent states that are localized in the shear direction together with scaling ideas for the Navier-Stokes equation that combine length and Reynolds number, we show how such large scale structures can be morphed into smaller scale coherent structures. As the Reynolds number increases, more of these states with ever smaller scales appear, all the way down to the Kolmogorov scale. We present the structure and dynamical properties of several families of exact coherent solution in plane Couette flow, with different degrees of spatial localization: Some of them remain localized in the center and help to built the turbulence cascade, others are localized near the walls and contribute to shaping the boundary layer profile.

9:44AM A18.00009 Helical mode breakdown in transitional boundary layers¹, RIKHI BOSE, PAUL DURBIN, Iowa State Univ — Results of direct numerical simulation of transition to turbulence in adverse pressure gradient boundary layers beneath free-stream turbulence will be presented. Instability waves are excited spontaneously and may be identified when intensity of free-stream turbulence (Tu) is sufficiently low. At very low $Tu \sim 0.1\%$, secondary instability of the TS waves and at high $Tu > 2\%$, conventional bypass mechanisms trigger turbulent spot formation. At low $Tu \sim 1\%$ transition proceeds through formation of helical modes. Helical structures as in $n = 1$ instability modes of axisymmetric wakes and jets are clearly identifiable in visualizations of isosurfaces of streamwise perturbation velocity. Helical modes also trigger transition at same level of Tu in zero pressure gradient boundary layers as well, provided that the inlet disturbances include a low amplitude time-periodic unstable TS wave. This indicates that these secondary instability modes might arise due to interaction of Klebanoff streaks and instability waves. Characteristically, the helical modes are inner instability modes.

¹This work was supported by NSF grant CBET-1228195. Computer time was provided by the Extreme Science and Engineering Discovery Environment (XSEDE).

Sunday, November 20, 2016 8:00AM - 9:57AM – Session A19 Bio: Fluid-Structure Interaction D136 - Jeffrey Guasto, Tufts University

8:00AM A19.00001 Non-invasive estimation of coral tentacle material properties using underwater PIV data¹, ANNE STAPLES, Virginia Tech, SHAI ASHER, URI SHAVIT, Technion - Israel Institute of Technology — With corals worldwide currently undergoing a third global bleaching event, understanding a detailed picture of local coral colony flow transport processes is more crucial than ever. Many coral species invest energy in extending flexible organs such as tentacles, that extrude from the coral's soft tissue surface and are used in either a passive or active manner for feeding, competitor sensing and even egg release. The significant role of these organs in transport and mixing processes is just beginning to be understood. For example, *Xeniidea*'s rhythmic pulsation of its tentacles has recently been shown to intensify mixing and enhance photosynthesis (Kremien et al., 2013). A critical part of modeling these tentacle-induced flows is obtaining measurements of the tentacles' material properties. Obtaining such measurements, however, is challenging, since the tentacle is expected to have significantly different material properties than a harvested specimen. Here, we demonstrate a non-invasive, *in situ* approach for estimating these material properties for *Favia favus* tentacles using underwater particle image velocimetry (PIV) data and tentacle-tracking data, along with structural dynamics models of the tentacles. In this data, 2.7×2 [cm 2] 1392x1024 pixel images were collected at a rate of 5 Hz 7mm above the crest of two separate *Favia Favus* colonies in Eilat, Israel. Using the data and models, we are able to estimate the Young's modulus for the tentacles, which is found to be a function of the wave frequency.

¹Partial funding by the Fulbright and Israel Science Foundations

8:13AM A19.00002 Passive scalar transport to and from the surface of a *Pocillopora* coral colony, MD MONIR HOSSAIN, ANNE STAPLES, Virginia Tech — Three-dimensional simulations of flow through a single *Pocillopora* coral colony were performed to examine the interaction between the flow conditions and scalar transport near a coral colony. With corals currently undergoing a third global bleaching event, a fuller understanding of the transport of nutrients, weak temperature gradients, and other passive scalars to and from the coral polyp tissue is more important than ever. The complex geometry of a coral colony poses a significant challenge for numerical simulation. To simplify grid generation and minimize computational cost, the immersed boundary method was implemented. Large eddy simulation was chosen as the framework to capture the turbulent flow field in the range of realistic Reynolds numbers of 5,000 to 30,000 and turbulent Schmidt numbers of up to 1,000. Both uniform and oscillatory flows through the colony were investigated. Significant differences were found between the cases when the scalar originated at the edge of the flow domain and was transported into the colony, versus when the scalar originated on the surface of the colony and was transported away from the coral. The domain-to-colony transport rates were found to be orders of magnitude higher than the colony-to-domain rates.

8:26AM A19.00003 Utilizing Surface Sensors to Identify Wake Regimes, MENGYING WANG, MAZIAR S. HEMATI, Univ of Minnesota - Twin Cities — Marine swimmers often exploit external flow structures to reduce locomotive effort. To achieve this advantage, these swimmers utilize mechanosensory organs on the surface of their bodies to detect hydrodynamic signals from the surrounding fluid, which can then be used to inform the control task. Recently, there has been a growing interest in developing similar flow sensing systems to achieve enhanced propulsive efficiency and maneuverability in human-engineered underwater vehicles. In particular, much attention has been given to the problem of wake sensing; however, these investigations have concentrated on a restricted class of wakes—i.e., Kármán-type vortex streets—whereas more complicated wake structures can arise in practice. In this talk, we will explore the possibility of identifying wake regimes through the use of surface sensors. Potential flow theory is adopted to simulate the interactions of various wakes with a fish-like body. Wakes in different dynamical regimes impart distinct hydrodynamic signatures on the body, which permits these regimes to be distinguished from one another in an automated fashion. Our results can provide guidance for improving flow sensing capabilities in human-engineered systems and hint at how marine swimmers may sense their hydrodynamic surroundings.

8:39AM A19.00004 Be together, not the same: Spatiotemporal organization of different cilia types generates distinct transport functions, JANNA NAWROTH, Harvard Univ, HANLIANG GUO, USC, EDWARD RUBY, University of Hawaii at Manoa, JOHN DABIRI, Stanford Univ, MARGARET MCFALL-NGAI, University of Hawaii at Manoa, EVA KANSO, USC — Motile cilia are microscopic, hair-like structures on the cell surface that can sense and propel the extracellular fluid environment. Cilia are often thought to be limited to stereotypic morphologies, beat kinematics and non-discriminatory clearance functions, but we find that the spatiotemporal organization of different cilia types and beat behaviors can generate complex flow patterns and transport functions. Here, we present a case study in the Hawaiian bobtail squid where collective ciliary activity and resulting flow fields help recruit symbiont bacteria to the animal host. In particular, we demonstrate empirically and computationally how the squid's internal cilia act like a microfluidic device that actively filters the water for potential bacterial candidates and also provides a sheltered zone allowing for accumulation of mucus and bacteria into a biofilm. Moreover, in this sheltered zone, different cilia-driven flows enhance diffusion of biochemical signals, which could accelerate specific bacteria-host recognition. These results suggest that studying cilia activity on the population level might reveal a diverse range of biological transport and sensing functions. Moreover, understanding cilia as functional building blocks could inspire the design of ciliated robots and devices.

8:52AM A19.00005 Cilia induced cerebrospinal fluid flow in the third ventricle of brain, YONG WANG, CHRISTIAN WESTENDORF, MPI for Dynamics and Self-Organization, REGINA FAUBEL, GREGOR EICHELE, MPI for Biophysical Chemistry, EBERHARD BODENSCHATZ, MPI for Dynamics and Self-Organization — Cerebrospinal fluid (CSF) conveys many physiologically important signaling factors through the ventricles of the mammalian brain. The walls of the ventricles are covered with motile cilia that were thought to generate a laminar flow purely following the curvature of walls. However, we recently discovered that cilia of the ventral third ventricle (v3V) generate a complex flow network along the wall, leading to subdivision of the v3V. The contribution of such cilia induced flow to the overall three dimensional volume flow remains to be investigated by using numerical simulation, arguably the best approach for such investigations. The lattice Boltzmann method is used to study the CSF flow in a reconstructed geometry of the v3V. Simulation of CSF flow neglecting cilia in this geometry confirmed that the previous idea about pure confined flow does not reflect the reality observed in experiment. The experimentally recorded ciliary flow network along the wall was refined with the smoothed particle hydrodynamics and then adapted as boundary condition in simulation. We study the contribution of the ciliary network to overall CSF flow and identify site-specific delivery of CSF constituents with respect to the temporal changes.

9:05AM A19.00006 Laboratory model of inner ear mechano-transduction¹, IBRAHIM MOHAMMAD, Student, University of Rochester, SRDJAN PRODANOVIC, PhD Student, University of Rochester, DANIELLE LAIACONA, Binghamton University, JONG-HOON NAM, Assistant Prof.of Mechanical Engineering and Biomedical Engineering, University of Rochester, DOUGLAS KELLEY, Assistant Professor of Mechanical Engineering, University of Rochester — A sound wave entering the mammalian ear displaces cochlear fluid, which in turn displaces hair-like organelles called stereocilia that act as acoustic sensors. Their incredible sensitivity is poorly understood, and probably depends on pre-amplification via fluid-structure interaction. In this talk, I will show how our lab uses a laboratory model to simulate this biological system to study the viscous coupling between the vibrating structures, cochlear fluid, and stereocilia. I will present measurements of modeled stereocilia gain and phase difference over a range of frequencies. Recent numerical simulations show that the sensor behaves as a high-pass filter with a gain plateau. However, our results show a peak in the gain. Further, I will show how the length of stereocilia affects gain.

¹This project was supported by NIH NIDCD R01 DC014685

9:18AM A19.00007 Predator localization by sensory hairs in free-swimming arthropods, DAISUKE TAKAGI, DANIEL K. HARTLINE, University of Hawaii at Manoa — Free-swimming arthropods such as copepods rely on minute deflections of cuticular hairs (or "setae") for local flow sensing that is needed to detect food and escape from predators. We present a simple hydrodynamic model to analyze how the location, speed, and size of an approaching distant predator can be inferred from local flow deformation alone. The model informs suitable strategies of escape from an imminent predatory attack. The sensory capabilities of aquatic arthropods could inspire the design of flow sensors in technological applications.

9:31AM A19.00008 Bioinspired sensory systems for local flow characterization, BRENDAN COLVERT, KEVIN CHEN, EVA KANSO, Univ of Southern California — Empirical evidence suggests that many aquatic organisms sense differential hydrodynamic signals. This sensory information is decoded to extract relevant flow properties. This task is challenging because it relies on local and partial measurements, whereas classical flow characterization methods depend on an external observer to reconstruct global flow fields. Here, we introduce a mathematical model in which a bioinspired sensory array measuring differences in local flow velocities characterizes the flow type and intensity. We linearize the flow field around the sensory array and express the velocity gradient tensor in terms of frame-independent parameters. We develop decoding algorithms that allow the sensory system to characterize the local flow and discuss the conditions under which this is possible. We apply this framework to the canonical problem of a circular cylinder in uniform flow, finding excellent agreement between sensed and actual properties. Our results imply that combining suitable velocity sensors with physics-based methods for decoding sensory measurements leads to a powerful approach for understanding and developing underwater sensory systems.

9:44AM A19.00009 Pulsating Soft Corals¹, SHILPA KHATRI, University of California, Merced, ROI HOLZMAN, Tel Aviv University and Inter-University Institute for Marine Sciences, LAURA MILLER, JULIA SAMSON, University of North Carolina at Chapel Hill, URI SHAVIT, Technion (Israel Institute of Technology) — Soft corals of the family Xeniidae have a pulsating motion, a behavior not observed in many other sessile organisms. We are studying how this behavior may give these corals a competitive advantage. We will present experimental data and computational simulations of the pulsations of the coral. Video data and kinematic analysis will be shown from the lab and the field. We will present direct numerical simulations of the pulsations of the coral and the resulting fluid flow by solving the Navier-Stokes equations coupled with the immersed boundary method. Furthermore, parameter sweeps studying the resulting fluid flow will be discussed.

¹This work is supported by NSF PoLS 1505061 (to S. Khatri) and 1504777 (to L. Miller)

Sunday, November 20, 2016 8:00AM - 9:57AM –
Session A20 Bio: Proteins and Cells D137-138 - Jin Liu, Washington State University

8:00AM A20.00001 Coarse-grained Simulations of Sugar Transport and Conformational Changes of Lactose Permease¹, JIN LIU, S M YEAD JEWEL, PRASHANTA DUTTA, Washington State University

— *Escherichia coli* lactose permease (LacY) actively transports lactose and other galactosides across cell membranes through lactose/H⁺ symport process. Lactose/H⁺ symport is a highly complex process that involves sugar translocation, H⁺ transfer, as well as large-scale protein conformational changes. The complete picture of lactose/H⁺ symport is largely unclear due to the complexity and multiscale nature of the process. In this work, we develop the force field for sugar molecules compatible with PACE, a hybrid and coarse-grained force field that couples the united-atom protein models with the coarse-grained MARTINI water/lipid. After validation, we implement the new force field to investigate the transport of a β -D-galactopyranosyl-1-thio- β -D-galactopyranoside (TDG) molecule across a wild-type LacY during lactose/H⁺ symport process. Results show that the local interactions between TDG and LacY at the binding pocket are consistent with the X-ray experiment. Protonation of Glu325 stabilizes the TDG and inward-facing conformation of LacY. Protonation of Glu269 induces a dramatic protein structural reorganization and causes the expulsion of TDG from LacY to both sides of the membrane. The structural changes occur primarily in the N-terminal domain of LacY.

¹This work is supported by NSF grants: CBET-1250107 and CBET -1604211

8:13AM A20.00002 Computational Modeling and Simulations of Bioparticle Internalization Through Clathrin-mediated Endocytosis¹, HUA DENG, PRASHANTA DUTTA, JIN LIU, Washington State University

— Clathrin-mediated endocytosis (CME) is one of the most important endocytic pathways for the internalization of bioparticles at lipid membrane of cells, which plays crucial roles in fundamental understanding of viral infections and interacellular/transcellular targeted drug delivery. During CME, highly dynamic clathrin-coated pit (CCP), formed by the growth of ordered clathrin lattices, is the key scaffolding component that drives the deformation of plasma membrane. Experimental studies have shown that CCP alone can provide sufficient membrane curvature for facilitating membrane invagination. However, currently there is no computational model that could couple cargo receptor binding with membrane invagination process, nor simulations of the dynamic growing process of CCP. We develop a stochastic computational model for the clathrin-mediated endocytosis based on Metropolis Monte Carlo simulations. In our model, the energetic costs of bending membrane and CCP are linked with antigen-antibody interactions. The assembly of clathrin lattices is a dynamic process that correlates with antigen-antibody bond formation. This model helps study the membrane deformation and the effects of CCP during functionalized bioparticles internalization through CME.

¹This work is supported by NSF grants: CBET-1250107 and CBET -1604211

8:26AM A20.00003 Modeling of Nutrient Transport and the Onset of Hypoxia in a Microfluidic Cell Culture Environment¹, ADNAN MORSHED, PRASHANTA DUTTA, Washington State University

— Transport of essential nutrients such as oxygen and ascorbate plays a critical role in dictating tumor growth. For example, hypoxia, the depletion of intracellular oxygen levels below 6%, initiates major changes in cellular dynamics causing tumor cell survival. The intercapillary distance (distance between blood vessels) across a colony of growing tumor cells and the flow around the colony are important factors for the initiation of hypoxia. In this study, the dynamics of intracellular species inside a colony of tumor cells are investigated by varying the flow and unsteady permeation in a microfluidic cell culture device. The oxygen transport across the cell membrane is modeled through diffusion, while ascorbate transport from plasma is addressed by a concentration dependent uptake model. Our model shows that the onset of hypoxia is possible in HeLa cell within the first minute of total extracellular oxygen deprivation. This eventually leads to anoxia inside the cell block representing the development of a necrotic core that maintains a dynamic balance with growing cells and scarce supply. Results also indicate that the intercapillary distance and flow rate of nutrients can alter this balance, which has implications in the progression of hypoxic response.

¹This work was supported in part by the U.S. National Science Foundation under grant no. DMS 1317671.

8:39AM A20.00004 Computational Studies of Drug Release, Transport and Absorption in the Human Intestines , FARHAD BEHAFARID, J. G. BRASSEUR, U Colorado, G. VIJAYAKUMAR, NREL, B. JAYARAMAN, Oklahoma State U, Y. WANG, Georgia Tech

— Following disintegration of a drug tablet, a cloud of particles 10-200 μm in diameter enters the small intestine where drug molecules are absorbed into the blood. Drug release rate depends on particle size, solubility and hydrodynamic enhancements driven by gut motility. To quantify the interrelationships among dissolution, transport and wall permeability, we apply lattice Boltzmann method to simulate the drug concentration field in the 3D gut released from polydisperse distributions of drug particles in the "fasting" vs. "fed" motility states. Generalized boundary conditions allow for both solubility and gut wall permeability to be systematically varied. We apply a local 'quasi-steady state' approximation for drug dissolution using a mathematical model generalized for hydrodynamic enhancements and heterogeneity in drug release rate. We observe fundamental differences resulting from the interplay among release, transport and absorption in relationship to particle size distribution, luminal volume, motility, solubility and permeability. For example, whereas smaller volume encourages higher bulk concentrations and reduced release rate, it also encourages higher absorption rate, making it difficult to generalize predictions. *Supported by FDA.*

8:52AM A20.00005 Transport of Brownian spheroidal nanoparticles in near-wall vascular flows for cancer therapy¹, TIRAS Y. LIN, PREYAS N. SHAH, BRYAN R. SMITH, ERIC S.G. SHAQFEH, Stanford University

— The microenvironment local to a tumor is characterized by a leaky vasculature induced by angiogenesis from tumor growth. Small pores form in the blood vessel walls, and these pores provide a pathway for cancer-ameliorating nanoparticle drug carriers. Using both simulations and microfluidics experiments, we investigate the extravasation of nanoparticles through pores. Using Brownian dynamics simulations, we evolve the stochastic equations for both point particles and finite-size spheroids of varying aspect ratio. We investigate the effect of wall shear flow and pore suction flow (Sampson flow) on the extravasation process. We consider pores of two types: physiologically relevant short pores with a length equal to the particle size and long pores which are relevant to diffusion through membranes. Additionally, we perform microfluidics experiments in which the extravasation rates of various nanoparticles tagged with fluorescent dye through pores are measured. In particular, using fluorometry we measure the flux of nanoparticles across a track-etched membrane, which separates two chambers. Our preliminary results indicate that the flux measured from experiment agrees reasonably with the simulations done with long pores, and we discuss the effect of pore length on extravasation.

¹T.Y.L. is supported by the Department of Defense (DoD) through the National Defense Science & Engineering Graduate Fellowship (NDSEG) Program

9:05AM A20.00006 Numerical analysis of cell adhesion in capillary flow¹ , NAOKI TAKEISHI,

Kyoto University, YOHSUKE IMAI, SHUNICHI ISHIDA, TOSHIHIRO OMORI, Tohoku University, ROGER KAMM, Massachusetts Institute of Technology, TAKUJI ISHIKAWA, Tohoku University — Numerical simulation of cell adhesion was performed for capillaries whose diameter is comparable to or smaller than that of the cell. Despite a lot of works about leukocyte and tumor cell rolling, cell motion in capillaries has remained unclear. The solid and fluid mechanics of a cell in flow was coupled with a slip bond model of ligand-receptor interactions. When the size of a capillary was reduced, the cell always transitioned to bullet-like motion, with a consequent decrease in the velocity of the cell. A state diagram is obtained for various values of capillary diameter and receptor density. According to our numerical results, bullet motion enables firm adhesion of a cell to the capillary wall even for a weak ligand-receptor binding. We also quantified effects of various parameters, including the dissociation rate constant, the spring constant, and the reactive compliance on the characteristics of cell motion. Our results suggest that even under the interaction between PSGL-1 and P-selectin, which is mainly responsible for leukocyte rolling, a cell is able to show firm adhesion in a small capillary. These findings may help in understanding such phenomena as leukocyte plugging and cancer metastasis.

¹This research was supported by JSPS KAKENHI Grant Numbers 25000008, 26107703, 14J03967. We also acknowledge support from the Tohoku University Division for International Advanced Research and Education Organization.

9:18AM A20.00007 Exercise, Insulin Absorption Rates, and Artificial Pancreas Control , SPENCER FRANK, Univ of California - Berkeley, LING HINSHAW, RITA BASU, ANANDA BASU, Division of Endocrinology, Mayo Clinic, ANDREW J. SZERI, Univ of California - Berkeley — Type 1 Diabetes is characterized by an inability of a person to endogenously produce the hormone insulin. Because of this, insulin must be injected – usually subcutaneously. The size of the injected dose and the rate at which the dose reaches the circulatory system have a profound effect on the ability to control glucose excursions, and therefore control of diabetes. However, insulin absorption rates via subcutaneous injection are variable and depend on a number of factors including tissue perfusion, physical activity (vasodilation, increased capillary throughput), and other tissue geometric and physical properties. Exercise may also have a sizeable effect on the rate of insulin absorption, which can potentially lead to dangerous glucose levels. Insulin-dosing algorithms, as implemented in an artificial pancreas controller, should account accurately for absorption rate variability and exercise effects on insulin absorption. The aforementioned factors affecting insulin absorption will be discussed within the context of both fluid mechanics and data driven modeling approaches.

9:31AM A20.00008 Study of Microfluidic System for Mechanical Property Measurement of Fluid-cell Interface¹ , JI YOUNG MOON, Yonsei University, The University of Sydney, JUNG SHIN LEE, SE BIN CHOI, HONG MIN YOON, Yonsei Univ, ROGER I. TANNER, The University of Sydney, JOON SANG LEE, Yonsei Univ — The system for measuring the mechanical properties of active cell is studied through an integrated microfluidic system for cell separation, alignment and measurement of mechanical properties. A highly efficient lattice Boltzmann method (LBM) was employed to optimize the micro-fluidic system to investigate the interrelations between mechanical properties and various surrounding fluid ingredients which are difficult to observe using current experimental techniques. A combination model of the three dimensional LBM and the immersed boundary method (IBM) were used to simulate these systems. The LBM was used to determine incompressible fluid flow with a regular Eulerian grid. The IBM was used to solve the deformation of cells and matrix fluid interaction with a Lagrangian grid. Highly non-linear results such as cell-cell interactions, fluid-cell interactions, and optical force-cell interactions is studied.

¹National Research Foundation of Korea (NRF) (grant number: NRF-2015R1A2A1A15056182, NRF-2015R1A5A1037668)

9:44AM A20.00009 Human endothelial cell responses to cardiovascular inspired pulsatile shear stress , MATTHEW WATSON, LAUREN BAUGH , LAUREN BLACK III , ERICA KEMMERLING, Tufts University — It is well established that hemodynamic shear stress regulates blood vessel structure and the development of vascular pathology. This process can be studied via *in vitro* models of endothelial cell responses to pulsatile shear stress. In this study, a macro-scale cone and plate viscometer was designed to mimic various shear stress waveforms found in the body and apply these stresses to human endothelial cells. The device was actuated by a PID-controlled DC gear-motor. Cells were exposed to 24 hours of pulsatile shear and then imaged and stained to track their morphology and secretions. These measurements were compared with control groups of cells exposed to constant shear and no shear. The results showed that flow pulsatility influenced levels of secreted proteins such as VE-cadherin and neuroregulin IHC. Cell morphology was also influenced by flow pulsatility; in general cells exposed to pulsatile shear stress developed a higher aspect ratio than cells exposed to no flow but a lower aspect ratio than cells exposed to steady flow.

Sunday, November 20, 2016 8:00AM - 9:57AM –

Session A21 Bubbles: Dynamics D139-140 - Randy Ewoldt, University of Illinois at Urbana Champaign

@ArticleLamstaes2016,

author="Lamstaes, Catherine

and Eggers, Jens",

title="Arrested Bubble Rise in a Narrow Tube",

journal="Journal of Statistical Physics",

year="2016",

8:00AM A21.00001 Arrested bubble 'rise' in a narrow tube¹ , CATHERINE LAMSTAES, JENS EGGRERS, University of Bristol — A long air bubble placed inside a vertical tube closed at the top rises by displacing the fluid above it. Bretherton, however, found that if the tube radius, R , is smaller than a critical value $R_c = 0.918 \ell_c$, where $\ell_c = \sqrt{\gamma/\rho g}$ is the capillary length, there is no solution corresponding to steady rise. We explain this finding by studying the unsteady bubble motion for $R < R_c$. We show the minimum spacing between the bubble and the tube goes to zero like $t \propto t^{-4/5}$ in limit of large time t . This leads to a rapid slow-down of the bubble's mean speed $U \propto -t^{-2}$, giving the appearance of arrested motion. What may seem surprising is that U is negative: the bubble moves down rather than up. We explain this observation by the bubble's expansion to the walls of the tube, pushing fluid in the direction opposite to gravity.

¹EPSRC

8:13AM A21.00002 Sinking Bubbles , JEREMY KOCH, RANDY EWOLDT, University of Illinois at Urbana-Champaign — Intuition tells us that bubbles will rise and steel objects will sink in liquids, though here we describe the opposite. With experimental demonstration and theoretical rationale, we describe how the motion of containers of liquid with immersed solid objects and air bubbles can cause curious behaviors: sinking bubbles and rising high-density particles. Bubbles and solid spheres of diameter on the order of a few millimeters are introduced into fluids with different rheological constitutive behaviors. Imposed motion of the rigid container allows for control of the trajectories of the immersed particles – without the container imparting direct shearing motion on the fluid. Results demonstrate the necessary conditions to prevent or produce net motion of the bubbles and heavy particles, both with and against gravitational expectations.

8:26AM A21.00003 Numerical simulations of the rise and stability of Taylor bubbles in vertical tubes using the diffuse-interface method¹ , HABIB ABUBAKAR, Imperial College London, ARNOLDO BADILLO, Paul Scherrer Institute, Switzerland, OMAR MATAR, Imperial College London — Taylor bubbles are characteristic features of the slug flow regime in gas-liquid flows in vertical pipes. Experimental observations have shown that at sufficiently large pipe diameters (>0.1 m), the slug flow regime, and hence Taylor bubbles, are no longer observed. Numerical simulations of a Taylor bubble rising in a quiescent liquid have also shown that the turbulent bubble wake in such large-diameter tubes has great impact on the stability of the subsequent trailing bubbles. In view of these observations, large-scale numerical simulations of Taylor bubbles are carried out using the diffuse-interface method over a range of experimentally relevant conditions. The results of these simulations (including benchmark cases) are discussed with a view to providing insight into the mechanisms underlying Taylor bubble instability.

¹EPSRC UK Programme Grant MEMPHIS (EP/K003976/1)

8:39AM A21.00004 Armoring confined bubbles in concentrated colloidal suspensions

, YINGXIAN YU, SEPIDEH KHODAPARAST, HOWARD STONE, Princeton University — Encapsulation of a bubble with microparticles is known to significantly improve the stability of the bubble. This phenomenon has recently gained increasing attention due to its application in a variety of technologies such as foam stabilization, drug encapsulation and colloidosomes. Nevertheless, the production of such colloidal armored bubble with controlled size and particle coverage ratio is still a great challenge industrially. We study the coating process of a long air bubble by microparticles in a circular tube filled with a concentrated microparticles colloidal suspension. As the bubble proceeds in the suspension of particles, a monolayer of micro-particles forms on the interface of the bubble, which eventually results in a fully armored bubble. We investigate the phenomenon that triggers and controls the evolution of the particle accumulation on the bubble interface. Moreover, we examine the effects of the mean flow velocity, the size of the colloids and concentration of the suspension on the dynamics of the armored bubble. The results of this study can potentially be applied to production of particle-encapsulated bubbles, surface-cleaning techniques, and gas-assisted injection molding.

8:52AM A21.00005 Tightrope walking bubbles , HELENE DE MALEPRADE, CHRISTOPHE CLANET,

DAVID QUERE, PMMH - ESPCI ; Ladhyx - Ecole Polytechnique — A fiber can hold a certain amount of liquid, which allows us to capture flying drops and control their motion. Immersed in water, a fiber can efficiently capture air bubbles only if it is hydrophobic. Using a superhydrophobic coating on an inclined wire, we experimentally control the rising velocity of air bubbles walking along the tightrope. We discuss the nature of the friction around the walker, and the resulting speed of bubbles.

9:05AM A21.00006 Effect of gravity on the liquid film surrounding a bubble translating in a tube.¹ , OMER ATASI, Universite Libre de Bruxelles and Princeton University, SEPIDEH KHODAPARAST, Ecole polytechnique federale de Lausanne and Princeton University, BENOIT SCHEID, Universite Libre de Bruxelles, HOWARD A. STONE, Princeton University — The motion of confined elongated bubbles in small diameter tubes filled with viscous liquid is a ubiquitous problem relevant to many industrial and medical applications such as lubrication, oil extraction and the treatment of pulmonary disorders. As a confined bubble proceeds into a liquid-filled tube a thin film of liquid is formed on the tube wall. For negligible inertia and buoyancy ($Bo, Re \approx 0$), the thickness of this film depends only on the capillary number Ca . However, gravitational effects are not negligible for horizontal tubes of millimeter-scale diameter, corresponding to a finite Bond number Bo . We perform experiments and theoretical analysis to investigate the effect of Bo on the thin film thickness. Several values of Bo are tested experimentally by changing the tube diameter. Due to gravity, the film deposited on the upper wall of the channel is thinner than the film at the bottom wall, and the bubble is inclined toward the bottom of the tube as it translates along the tube. The inclination angle increases with increasing Bo and Ca . Our theoretical analysis shows that this effect is caused by the bubble being off-center in the tube at finite values of Bo .

¹F.R.S. - FNRS

9:18AM A21.00007 Mixing of a passive scalar in a turbulent bubbly flow , ELISE ALMRAS,

VARGHESE MATHAI, DETLEF LOHSE, Physics of Fluids, University of Twente, CHAO SUN, Center for Combustion Energy & Department of Thermal Engineering, Tsinghua University — In this work, we investigate the mixing of a passive scalar at high Schmidt number by a homogenous bubble swarm in the presence of external turbulence. The experiments are conducted in the Twente Water Tunnel, in which nearly homogeneous and isotropic turbulence is produced using an active grid. The level of the external turbulence is varied for Taylor-Reynolds number ranging from 180 to 360 and the global gas volume fraction is varied from 0 to 3%. We continuously inject a passive fluorescent dye at a fixed position, and measure the horizontal concentration profile of the dye at different heights by recording the fluorescence levels and applying an image processing. A horizontal effective diffusivity is then calculated from the spatial evolution of the variance of the horizontal concentration profile. The diffusion coefficient is found to increase with the strength of the external turbulence and the gas volume fraction, indicating an enhancement of mixing. Finally, we connect the effective diffusion coefficient to the hydrodynamic properties of the flow in order to have a better understanding on the underlying mixing mechanisms.

9:31AM A21.00008 Bubble Size Distribution in a Vibrating Bubble Column , SHAHROUZ

MOHAGHEGHIAN, TREVOR WILSON, BRET VALENZUELA, TYLER HINDS, KEVIN MOSEN, BRIAN ELBING, Oklahoma State Univ — While vibrating bubble columns have increased the mass transfer between phases, a universal scaling law remains elusive. Attempts to predict mass transfer rates in large industrial scale applications by extrapolating laboratory scale models have failed. In a stationary bubble column, mass transfer is a function of phase interfacial area (PIA), while PIA is determined based on the bubble size distribution (BSD). On the other hand, BSD is influenced by the injection characteristics and liquid phase dynamics and properties. Vibration modifies the BSD by impacting the gas and gas-liquid dynamics. This work uses a vibrating cylindrical bubble column to investigate the effect of gas injection and vibration characteristics on the BSD. The bubble column has a 10 cm diameter and was filled with water to a depth of 90 cm above the tip of the orifice tube injector. BSD was measured using high-speed imaging to determine the projected area of individual bubbles, which the nominal bubble diameter was then calculated assuming spherical bubbles. The BSD dependence on the distance from the injector, injector design (1.6 and 0.8 mm ID), air flow rates (0.5 to 5 lit/min), and vibration conditions (stationary and vibration conditions varying amplitude and frequency) will be presented. In addition to mean data, higher order statistics will also be provided.

9:44AM A21.00009 Modeling the initial contact line dynamics of dewetting bubbles¹

, MARK MENESSES, MATTHIEU LAURENT, JAMES BIRD, Boston University — When a rising bubble comes to rest beneath a solid horizontal surface, the resulting liquid film dewets to minimize the total free energy of the three phase system. For partially wetting surfaces, the presence of the contact angle yields dynamics which are assumed to be governed by viscous effects. In contrast, the early-time dynamics for drops spreading on partially wetting surfaces are dominated by inertial effects. Motivated by the discrepancy between these two systems, we conduct experiments on dewetting bubbles and find that the short-time dynamics fail to obey purely viscous or inertial scalings. We draw from previously proposed dewetting and spreading models to develop a new model that can rationalize the anomalous scalings that we observe. Our results suggest that the speed that a bubble adheres to a partially wetting surface is set by an interplay of capillary waves and contact line motion.

¹We acknowledge support from ONR, Saint-Gobain, and NSF GRFP.

Sunday, November 20, 2016 8:00AM - 9:57AM –

Session A22 Nano Flows: General E141/142 - Jonathan Freund, University of Illinois at Urbana-Champaign

8:00AM A22.00001 Atomic-scale thermocapillary flow in focused ion beam milling

, KALLOL DAS, HARLEY JOHNSON, JONATHAN FREUND, University of Illinois at Urbana-Champaign — Focused ion beams (FIB) offer an attractive tool for nanometer-scale manufacturing and material processing, particularly because they can be focused to a few nanometer diameter spot. This motivates their use for many applications, such as sample preparation for transmission electron microscopy (TEM), forming nanometer scale pores in thin films for DNA sequencing. Despite its widespread use, the specific mechanisms of FIB milling, especially at high ion fluxes for which significant phase change might occur, remains incompletely understood. Here we investigate the process of nanopore fabrication in thin Si films using molecular dynamics simulation where Ga⁺ ions are used as the focused ions. For a range of ion intensities in a realistic configuration, a recirculating melt region develops, which is seen to flow with a symmetrical pattern, counter to how it would flow were it is driven by the ion momentum flux. Such flow is potentially important for the shape and composition of the formed structures. Relevant stress scales and estimated physical properties of silicon under these extreme conditions support the importance thermocapillary effects. A continuum flow model with Marangoni forcing reproduces the flow.

8:13AM A22.00002 Origin of dynamic contact angle at the nanoscale. , ALEX LUKYANOV,

ALEXEI LIKHTMAN¹, University of Reading — Generation of a dynamic contact angle in the course of wetting is a fundamental phenomenon of nature. Dynamic wetting processes have a direct impact on flows at the nanoscale, and therefore, understanding them is exceptionally important to emerging technologies. Here, we reveal the microscopic mechanism of dynamic contact angle generation, which is demonstrated using large-scale molecular dynamics simulations of bead-spring model fluids. It has been shown that the main cause of local contact angle variations is the distribution of microscopic force acting at the contact line region. We were able to retrieve this force with high accuracy to understand its nature and its characteristic physical parameters. It has been directly established that the force distribution can be solely predicted on the basis of a general friction law for liquid flow at solid surfaces first formulated by Thompson & Troian on the basis of molecular dynamics simulations of Lennard-Jones liquids. The relationship with the friction law provides both an explanation of the phenomenon of dynamic contact angle and a methodology for future predictions. The mechanism is intrinsically microscopic, universal, and irreducible and is applicable to a wide range of problems associated with wetting phenomena.

¹Deceased

8:26AM A22.00003 Flow rate and slip length measurements of water in single nanopipes , DAVID MALLIN, PETER TABOREK, ANGEL VELASCO, University of California Irvine — Measurements of pressure

driven water flows in hydrophobic and hydrophilic fused quartz capillaries of 200 nm diameter are compared. Typical flow rates on the order of 100 femtoliters and pressure drops up to 50 Atm were used. Water exited the capillaries into an oil reservoir where the volume of the pendant drop was monitored using time lapse photography. The typical growth rate for the drop diameter was ~50 um per day. Flow through a single nanotube can be continued for several weeks. For the untreated quartz capillary, the results are consistent with a no-slip boundary condition. The hydrophilic capillaries are chemically treated with polydimethylsiloxane (PDMS) to form hydrophobic surfaces. Successful surface preparation is confirmed with pressure threshold behavior of the water flow. Our technique can detect slip lengths above 3 nm.

8:39AM A22.00004 Massive radius-dependent flow slippage in carbon nanotubes¹

, ALESSANDRO SIRIA, ELEONORA SECCHI, CNRS - ENS, SOPHIE MARBACH, ENS, ANTOINE NIGUS, CNRS - ENS, DEREK STEIN, Brown University, LYDRIC BOCQUET, CNRS - ENS — Nanofluidics is the frontier where the continuum picture of fluid mechanics confronts the atomic nature of matter. Recent reports indicate that carbon nanotubes exhibit exceptional water transport properties due to nearly frictionless interfaces and this has stimulated interest in nanotube-based membranes for desalination, nano-filtration, and energy harvesting. However, the fundamental mechanisms of water transport inside nanotubes and at water-carbon interfaces remain controversial, as existing theories fail to provide a satisfying explanation for the limited experimental results. We report a study of water jets emerging from single nanotubes made of carbon and boron-nitride materials. Our experiments reveal extensive and radius-dependent surface slippage in carbon nanotubes (CNT). In stark contrast, boron-nitride nanotubes (BNNT), which are crystallographically similar to CNTs but electronically different, exhibit no slippage. This shows that slippage originates in subtle atomic-scale details of the solid-liquid interface.

¹ERC StG - NanoSOFT

8:52AM A22.00005 Macroscopic nanoporous graphene membranes for molecular-sieving-based gas separation , MICHAEL BOUTILIER, ROHIT KARNIK, NICOLAS HADJICONSTANTINOU, MIT —

Nanoporous graphene membranes have the potential to exceed permeance and selectivity limits of existing gas separation membranes. This is made possible by the atomic thickness of the material, which can support sub-nanometer pores that enable molecular sieving while presenting low resistance to permeate flow. The feasibility of gas separation by graphene nanopores has been demonstrated experimentally on micron-scale areas of graphene. However, scaling up to macroscopic membrane areas presents significant challenges, including graphene imperfections and control of the selective nanopore size distribution across large areas. Towards this goal, gas permeance experiments are conducted on single and few layer graphene membranes to understand leakage pathways and a model is developed to predict conditions under which molecular sieving can occur in macroscopic membranes. Approaches to seal or mitigate the effects of micron and nanometer scale defects in graphene are investigated and methods of creating a high density of selectively permeable nanopores are explored. Experimental results demonstrating separation ratios exceeding the Knudsen effusion limit, indicating molecular sieving in agreement with the model predictions, are presented and discussed.

9:05AM A22.00006 Quantifying pore size and density for membranes in the Knudsen and transitional-flow regimes¹, RICHARD CASTELLANO, MATTHEW PURRI, ERICK HERNANDEZ, JERRY SHAN, Rutgers Univ, NGOC BUI, CHIATI CHEN, ERIC MESHOT, FRANCESCO FORNASIERO, LLNL — Membranes with well-controlled nanoscale pores have interest for applications as diverse as chemical separations, water purification, and “green” power generation. For instance, membranes incorporating carbon nanotubes (CNTs) as through-pores have been shown to pass fluids orders-of-magnitude faster than predicted by theory.¹ However, the efficient characterization of the pore size and density of membranes is an important area of focus, particularly for membranes fabricated from bulk nanotubes.² Here, we report on a new technique to identify the pore size (d) and number of open pores (N) in membranes. A nanoporous membrane is characterized with a combination of pressure-driven gas flow, and electrical-conductance measurements in aqueous solution. For the conductance measurements, the electrical current passing through the membrane scales as $d^2 N$. For pressurized gas flow, the scaling with molecular weight (M) and gas viscosity (μ) identifies the flow as either Poiseuille or Knudsen, scaling as either $d^4 N/\mu$ or $d^3 N/M^{1/2}$, respectively. With this combination of measurements, the pore size and number of pores in the membrane can be calculated. We validate this technique using track-etched polycarbonate membranes and CNT membranes with known pores, and show that it can be used to count open pores and identify defects in CNT membranes.

¹N. Biu, et al., Adv. Mat. (2016) 2) R. J. Castellano, et al., J. Applied Physics. (2015)

¹We would like to acknowledge DTRA for their funding and support of our research.

9:18AM A22.00007 Effect of meniscus contact angle during early regimes of spontaneous capillarity in nanochannels¹, N.K. KARNA, ELTON OYARZUA, University of Concepcion, J.H. WALTHER, Technical University of Denmark, HARVEY ZAMBRANO, University of Concepcion — In capillary imbibition, the classical Lucas-Washburn equation predicts a singularity as the fluid enters the channel consisting in an anomalous infinite velocity of the capillary meniscus. The Bosanquet's equation overcomes this problem by taking into account fluid inertia predicting an initial imbibition regime with constant velocity. Nevertheless, the initial constant velocity predicted by Bosanquet's equation is much greater than experimentally observed. In the present study, we conduct atomistic simulations to investigate capillary imbibition of water in silica nanochannels with heights between 4 and 18 nm. We find that the meniscus contact angle remains constant during the inertial regime and its value depends upon the height of the channel. We also find that the meniscus velocity computed at the channel entrance is related to the particular value of the meniscus contact angle. Moreover, after the inertial regime, the meniscus contact angle is found to be time dependent for all the channels under study. We propose an expression for the time evolution of the dynamic contact angle in nanochannels which, when incorporated in Bosanquet's equation, satisfactorily explains the initial capillary rise.

¹We acknowledge financial support from Conicyt project

9:31AM A22.00008 Carbon nanotube-based coatings to induce flow enhancement in hydrophilic nanopores.¹, ENRIQUE WAGEMANN, Universidad de Concepcion, J. H. WALTHER, Technical University of Denmark, HARVEY A. ZAMBRANO, Universidad de Concepcion — With the emergence of the field of nanofluidics, the transport of water in hydrophilic nanopores has attracted intensive research due to its many promising applications. Experiments and simulations have found that flow resistance in hydrophilic nanochannels is much higher than those in macrochannels. Indeed, this might be attributed to significant fluid adsorption on the channel walls and to the effect of the increased surface to volume ratio inherent to the nanoconfinement. Therefore, it is desirable to explore strategies for drag reduction in nanopores. Recently, studies have found that carbon nanotubes (CNTs) feature ultrafast water flow rates which result in flow enhancements of 1 to 5 orders of magnitude compared to Hagen-Poiseuille predictions. In the present study, CNT-based coatings are considered to induce water flow enhancement in silica nanopores with different radius. We conduct atomistic simulations of pressurized water flow inside tubular silica nanopores with and without inner coaxial carbon nanotubes. In particular, we compute water density and velocity profiles, flow enhancement and slip lengths to understand the drag reduction capabilities of single- and multi-walled carbon nanotubes implemented as coating material in silica nanopores.

¹We wish to thank partial funding from CRHIAM and FONDECYT project 11130559, computational support from DTU and NLHPC (Chile).

9:44AM A22.00009 CNT based thermal Brownian motor to pump water in nanodevices¹, ELTON OYARZUA, HARVEY ZAMBRANO, Universidad de Concepcion, J. H. WALTHER, Technical University of Denmark — Brownian molecular motors are nanoscale machines that exploit thermal fluctuations for directional motion by employing mechanisms such as the Feynman-Smoluchowski ratchet. In this study, using Non Equilibrium Molecular Dynamics, we propose a novel thermal Brownian motor for pumping water through Carbon Nanotubes (CNTs). To achieve this we impose a thermal gradient along the axis of a CNT filled with water and impose, in addition, a spatial asymmetry by fixing specific zones on the CNT in order to modify the vibrational modes of the CNT. We find that the temperature gradient and imposed spatial asymmetry drive the water flow in a preferential direction. We systematically modified the magnitude of the applied thermal gradient and the axial position of the fixed points. The analysis involves measurement of the vibrational modes in the CNTs using a Fast Fourier Transform (FFT) algorithm. We observed water flow in CNTs of 0.94, 1.4 and 2.0 nm in diameter, reaching a maximum velocity of 5 m/s for a thermal gradient of 3.3 K/nm. The proposed thermal motor is capable of delivering a continuous flow throughout a CNT, providing a useful tool for driving liquids in nanofluidic devices by exploiting thermal gradients.

¹We acknowledge partial support from Fondecyt project 11130559

Sunday, November 20, 2016 8:00AM - 9:57AM –
Session A25 Microscale Flows: Flow in Microchannels E145 - Omar Matar, Imperial College London

8:00AM A25.00001 Experimental and numerical study of a complex cross-junction microchannel¹, EMILIA NOWAK, MARK SIMMONS, University of Birmingham, LYES KAHOUADJI, RICHARD CRASTER, OMAR MATAR, Imperial College London, DAMIR JURIC, JALEL CHERGUI, LIMSI, CNRS, France, SEUNGWON SHIN, Hongik University, South Korea — Microfluidic devices occur in various fields such as inkjet printing, DNA chips, lab-on-a-chip technology, micro-propulsion and droplet-based microfluidics. Here, we examine drop and plug formation of immiscible liquids in a cross-shaped microchannel via high-speed imaging, shadowgraphy and PIV that allows interface topology and flow field tracking. We also present comparisons with direct numerical simulations using the new solver, BLUE, for massively parallel simulations of fully three-dimensional multiphase flows in complex solid geometries.

¹EPSRC UK Programme Grant MEMPHIS (EP/K003976/1)

8:13AM A25.00002 Flow control mechanism of capillary driven flow in microchannel using non-mechanical forces.

, BHARATH BABU NUNNA, SHIQUANG ZHUANG, EON SOO LEE¹, New Jersey Inst of Tech

— The capillary driven flow in microchannel is a self-driven flow by the natural phenomenon called surface tension of the fluid. The gradients in surface tension force which influence the flow field in microchannel is generated by the modulation of contact angle through a defined hydrophilization of the PDMS (Polydimethylsiloxane) microchannel surface. PDMS which is hydrophobic in nature is treated with various surface treatments in order to convert it to hydrophilic. The contact angle made by the fluid with the PDMS microchannel surface is altered when the surface is converted from hydrophobic to hydrophilic. The flow rate of fluid in the microchannel is directly proportional to the hydrophilicity of that microchannel since the capillary force which is the driving force of the flow is dependent on the contact angle. Flow control mechanism of capillary driven flow in microchannel using non-mechanical forces is developed by treating the microchannel surfaces with various surface treatments. The precise control of the surface characteristics like hydrophilicity and roughness of the microchannel helps to control the capillary flow in microchannel. The flow rate variation with respect to the various surface treated channels are studied.

¹Principal Investigator

8:26AM A25.00003 Effect of shear-thinning behaviour on liquid-liquid plug flow in microchannels¹

, EVANGELIA ROUMPEA, MAXIME CHINAUD, WEHELIYE HASHI WEHELIYE, PANAGIOTA ANGELI², Department of Chemical Engineering, University College London, Torrington Place, London WC1E 7JE, UK, LYES KAHOUADJI, OMAR K. MATAR, Department of Chemical Engineering, Imperial College London, South Kensington Campus, London SW7 2AZ, UK — The present work investigates the dynamics of plug formation of shear-thinning solutions in a 200 μm microchannel using a two-colour micro-PIV system. Measurements, including phase-averaged velocity fields, have been conducted both at the T-junction inlet and the main channel to enhance understanding of non-Newtonian liquid-liquid flows. Two aqueous glycerol solutions containing xanthan gum are used as the non-Newtonian fluids while 5 cSt silicone oil is the Newtonian phase. The current experimental results revealed a pronounced impact of the xanthan gum (shear-thinning behaviour) on the flow pattern transition boundaries, and enhance the fluid flowrates where plug flow occurred. The addition of polymer resulted also in different hydrodynamic characteristics such as a bullet-shaped plug and an increased film thickness between the plug and the wall. In the present work, the technique allows to capture the velocity field of both phases simultaneously. Experimental results are compared with the numerical simulations provided by the code BLUE.

¹Project funded under the UK Engineering and Physical Sciences Research Council (EPSRC) Programme Grant MEMPHIS

²corresponding author

8:39AM A25.00004 Effect of meniscus curvature on thermal transport in microchannels with ridged walls

, TOBY KIRK, SIMON GAME, Imperial College London, MARC HODES, Tufts University, ERIC KEAVENY, DEMETRIOS PAPAGEORGIOU, Imperial College London — It is well known that textured surfaces can reduce flow resistance in microchannels, but their effect on thermal transport in, e.g., direct liquid cooling of microprocessors, has only recently been considered. We investigate thermal transport in Poiseuille flow through a channel textured with periodic longitudinal ridges that are held at constant heat flux. We assume the liquid only makes contact with the tips of the ridges, reducing drag but also the area for heat transfer. Accounting for curvature of the interfaces (menisci) that bridge each cavity, we consider two asymptotic limits: (i) small meniscus deflection from flat, using boundary perturbation; (ii) channel height large compared to ridge period, using matched asymptotics. In limit (i), the problem is reduced to dual series equations. If limit (ii) is also taken, we find explicit expressions for the effective slip length and Nusselt number. A remarkable finding is that the simple slip length expressions have exponentially small errors and so are accurate even for channel heights as low as half a ridge period. Finally, limit (i) is compared against direct numerical computations using Chebyshev collocation, and the effect of arbitrary curvature on the Nusselt number is presented for the full range of channel geometries.

8:52AM A25.00005 Comparison of Hydrodynamic and Thermal Performance of Micro Heat Sinks with Inline and Staggered Arrangements of Cylindrical Micro Pin Fins

, ALI MOHAMMADI, PhD student, ALI KOSAR, Professor, FACULTY OF ENGINEERING AND NATURAL SCIENCES (FENS) COLLABORATION, NANOTECHNOLOGY RESEARCH AND APPLICATION CENTER (SUNUM) COLLABORATION — This computational study compares the hydrodynamic and thermal characteristics of flow inside a rectangular microchannel with different in-line and staggered arrangements of cylindrical micro pin fins (MPF). The channel dimensions are 5000 \times 1500 \times 100 m^3 ($\text{l} \times \text{w} \times \text{h}$) while the height and diameter of MPFs are both 100 m which results in the H/D ratio of 1. Two different values of 1.5 and 3 are considered for the horizontal and vertical pitch ratios (S_L/D and S_T/D) among MPFs in each of the in-line and staggered arrangements which results in eight configurations. A constant heat flux of 30 W/cm^2 is applied through the bottom section of microchannel as well as the liquid interacting surfaces of MPFs. The flow field is simulated at five different Reynolds numbers of 20, 40, 80, 120 and 160 using ANSYS FLUENT v.14.5. Four parameters of pressure drop, friction factor, Nusselt number and Thermal Performance Index (TPI) are used to analyze the hydrodynamic and thermal performance of micro heat sinks. Results show a great dependency of evaluating parameters on the vertical pitch ratios while minor dependencies are seen on the horizontal pitch ratio.

9:05AM A25.00006 Inertial migration of spherical particles in submillimeter-sized square channel flows

, HIROYUKI SHICHI, HIROSHI YAMASHITA, JUNJI SEKI, TOMOAKI ITANO, MASAKO SUGIHARA-SEKI, Kansai University — The distributions of neutrally buoyant spherical particles were measured at downstream cross-sections of submillimeter-sized square channels for the Reynolds number from 1 to 800. Polystyrene particles of diameter $d = 30 - 70 \mu\text{m}$ were suspended in water-glycerol mixture at the volume concentration of $2.5 - 11 \times 10^2 \text{ cm}^{-3}$, and this suspension was made to flow through square channels of width $D = 400 - 800 \mu\text{m}$ and length $L = 50 - 600 \text{ mm}$. The Reynolds number (Re) was defined in terms of the average flow velocity and the channel width. For the size ratio $d/D = 0.075 - 0.125$, we found that for $Re < 260$, particles were focused on four equilibrium positions placed at the center of channel faces, which was in accord with previous experimental and numerical studies. For $Re > 450$, four additional equilibrium positions were observed near the channel corners. Between these two Reynolds numbers (i.e., $260 < Re < 450$), we observed new equilibrium positions located on a heteroclinic orbit connecting the channel face and corner equilibrium positions. These new equilibrium positions were shifted towards the corner equilibrium positions with increasing Re .

9:18AM A25.00007 Prediction and validation of concentration gradient generation in a paper-based microfluidic channel¹, ILHOON JANG, GANG-JUNE KIM, SIMON SONG, Hanyang Univ — A paper-based microfluidic channel has obtained attention as a diagnosis device that can implement various chemical or biological reactions. With benefits of thin, flexible, and strong features of paper devices, for example, it is often utilized for cell culture where controlling oxygen, nutrients, metabolism, and signaling molecules gradient affects the growth and movement of the cells. Among various features of paper-based microfluidic devices, we focus on establishment of concentration gradient in a paper channel. The flow is subject to dispersion and capillary effects because a paper is a porous media. In this presentation, we describe facile, fast and accurate method of generating a concentration gradient by using flow mixing of different concentrations. Both theoretical prediction and experimental validation are discussed along with inter-diffusion characteristics of porous flows.

¹This work was supported by the National Research Foundation of Korea(NRF) grant funded by the Korea government(MSIP) (No. 2016R1A2B3009541)

9:31AM A25.00008 Internal convection of two interacting liquid slugs of aqueous solution placed inside a microchannel¹, TAPAN KUMAR PRADHAN, PRADIPTA KUMAR PANIGRAHI, Indian Institute of Technology Kanpur — We experimentally investigated the internal convection of two neighboring liquid slugs of aqueous NaCl solution present inside a microchannel having cross-sectional area of $1\text{ mm} \times 1\text{ mm}$. Micro-PIV technique is used to measure the velocity field inside the slugs. The two slugs have different solute concentration (1 M and 2 M) and the volume of each slug is equal to $2\text{ }\mu\text{L}$. There is no physical contact between the two slugs and the slugs are separated by a distance of $680\text{ }\mu\text{m}$. Concentration difference between the two slugs lead to different vapor pressure at the liquid-air interface of the two slugs. Slug having lower solute concentration has higher vapor pressure at the interface as compared to the slug having higher solute concentration. Hence, water evaporates from the slug having lower solute concentration and condenses on the slug having higher solute concentration. Evaporation and condensation lead to buoyancy driven Rayleigh convection inside both the slugs. Single recirculating loop is observed in both the slugs. The flow strength in both the slugs decreases with time as evaporation and condensation decreases due to reduce in concentration difference between the two slugs.

¹Supported by Department of Science and Technology, Government of India.

9:44AM A25.00009 Tailoring tails in Taylor dispersion: how boundaries shape chemical delivery in microfluidics: experiments¹, DANIEL M. HARRIS, MANUCHEHR AMINIAN, FRANCESCA BERNARDI, ROBERTO CAMASSA, RICHARD M. MC LAUGHLIN, UNC Chapel Hill, UNC JOINT FLUIDS LAB TEAM — We present the results of an experimental investigation of the spreading of an initial dye concentration in laminar shear flow through rectangular ducts. In particular, we demonstrate the critical role that the cross-sectional aspect ratio plays in defining the longitudinal asymmetry of the resulting tracer distribution. Thin ducts (aspect ratio $\ll 1$) generate distributions with sharp fronts and tapering tails, whereas thick ducts (aspect ratio ~ 1) produce the opposite effect. The experimental results are shown to be in strong agreement with recent theoretical predictions. Our findings could potentially be useful in a number of microfluidic applications, some of which will be discussed.

¹Funding from NSF grant Nos.: RTG DMS-0943851, CMG ARC-1025523, DMS-1009750, and DMS-1517879.

Sunday, November 20, 2016 8:00AM - 9:31AM – Session A26 Reacting Flows: Emissions and Soot E146 - Yuan Xuan, Pennsylvania State University

8:00AM A26.00001 Effects of nucleating species on soot formation in turbulent non-premixed sooting jet flames, ABHISHEK JAIN¹, YUAN XUAN², Pennsylvania State Univ — Soot nucleation is one of the most unknown processes in the soot life cycle, and it is believed to occur from Polycyclic Aromatic Hydrocarbons (PAH) generated from the combustion of various fuel sources under locally fuel-rich conditions. Current soot nucleation models may include as few as one (typically naphthalene) or as many as a dozen of nucleating species. In this study, the effects of PAH inclusion in the soot nucleation model on soot yield and distribution are studied by means of Large-Eddy Simulations (LES) of two piloted turbulent non-premixed sooting jet flames, using ethylene and a jet fuel surrogate, respectively. Two sets of simulations are performed for each flame, one considering only a single nucleating PAH (naphthalene) and the other one considering a range of nucleating PAH from naphthalene to cyclopenta[cd]pyrene. Flamelet-based chemistry tabulation is used for the major thermochemical quantities, and a recently developed relaxation model is used for PAH species to account for the interactions between turbulence and their chemistry. The effects of nucleating PAH species on soot are highlighted by comparing the mean soot volume fraction distributions and statistical characteristics of soot obtained from both sets of simulations against experimental measurements.

¹Graduate Student, MNE

²Assistant Professor, Mechanical and Nuclear Engineering

8:13AM A26.00002 Radiation and Turbulence-Chemistry-Soot-Radiation Interactions in a High-Pressure Turbulent Spray Flame, S. FERREYRO, C. PAUL, A. SIRCAR, A. IMREN, D. C. HAWORTH, The Pennsylvania State University, S. ROY, M. F. MODEST, University of California, Merced — Simulations are performed of a transient high-pressure turbulent n-dodecane spray flame under engine-relevant conditions. An unsteady RANS formulation is used, with detailed chemistry, a two-equation soot model, various radiation heat transfer models, and a particle-based transported composition probability density function (PDF) method to account for composition and temperature. The PDF model results are compared with those from a locally well-stirred reactor (WSR) model to quantify the effects of turbulence-chemistry-soot-radiation interactions. Computed liquid and vapor penetration versus time, ignition delay, and flame lift-off are in good agreement with experiment, and relatively small differences are seen between the WSR and PDF models for these global quantities. Computed soot levels and spatial distributions from the WSR and PDF models show large differences, with PDF results being in better agreement with experimental measurements. A photon Monte Carlo method with line-by-line spectral resolution is used to compute the spectral intensity distribution of the radiation reaching the wall. This provides new insight into the relative importance of molecular gas radiation versus soot radiation, and the importance of unresolved turbulent fluctuations on radiative heat transfer.

8:26AM A26.00003 A priori analysis of a LES subfilter model for soot-turbulence-chemistry interactions, JEFFRY K. LEW, MICHAEL E. MUELLER, Princeton Univ — In a turbulent flame, soot interacts with turbulence and combustion chemistry at the smallest scales. An existing LES subfilter model [Mueller et al. Phys. Fluids 23 (2011)] proposes that soot-turbulence interactions are independent of chemistry due to the time scale separation between slow soot formation and rapid heat-releasing reactions. However, interactions between soot, turbulence, and chemistry occur even after the nucleation of soot from polycyclic aromatic hydrocarbon (PAH) dimers. In fact, the interplay of soot and gas-phase chemistry may be intensified during oxidation and surface growth. To capture these effects, a dependence on the local mixture fraction has been introduced into the subfilter model. This modified model is evaluated a priori using a direct numerical simulation (DNS) database of soot evolution in a turbulent non-premixed *n*-heptane/air jet flame.

8:39AM A26.00004 Direct numerical simulations of temporally developing hydrocarbon shear flames at elevated pressure: effects of the equation of state and the unity Lewis number assumption, AYSE KORUCU, RICHARD MILLER, Clemson Univ — Direct numerical simulations (DNS) of temporally developing shear flames are used to investigate both equation of state (EOS) and unity-Lewis (Le) number assumption effects in hydrocarbon flames at elevated pressure. A reduced *Kerosene/Air* mechanism including a semi-global soot formation/oxidation model is used to study soot formation/oxidation processes in a temporally developing hydrocarbon shear flame operating at both atmospheric and elevated pressures for the cubic Peng-Robinson real fluid EOS. Results are compared to simulations using the ideal gas law (IGL). The results show that while the unity- Le number assumption with the IGL EOS under-predicts the flame temperature for all pressures, with the real fluid EOS it under-predicts the flame temperature for 1 and 35 atm and over-predicts the rest. The soot mass fraction, Y_s , is only under-predicted for the 1 atm flame for both IGL and real gas fluid EOS models. While Y_s is over-predicted for elevated pressures with IGL EOS, for the real gas EOS Y_s 's predictions are similar to results using a non-unity Le model derived from non-equilibrium thermodynamics and real diffusivities. Adopting the unity Le assumption is shown to cause misprediction of Y_s , the flame temperature, and the mass fractions of CO, H and OH.

8:52AM A26.00005 Effects of Flamelet Generated Manifolds on Turbulent Flame Structure and Pollutant Emissions, A. CODY NUNNO, TEMISTOCLE GRENGA, MICHAEL E. MUELLER, Princeton University — Heat losses substantially modify turbulent combustion processes, especially the formation of pollutant emissions such as nitrogen oxides, which are highly sensitive to temperature. To account for heat loss effects in Large Eddy Simulation (LES) with flamelet models, *a priori* flamelet solutions are computed at reduced enthalpy. In this work, two methods for generating flamelets of lower enthalpy are compared to determine under what conditions the different methods produce different flame structure and different pollutant emissions in order to determine their validity limits. In the first method, a variable heat loss is introduced into the flamelet solutions that mimics a real heat loss, reducing the enthalpy primarily in the post-flame region of the flamelet. In the second method, fuel and oxidizer are converted to products in the unburned gases while retaining a constant unburned temperature, reducing the enthalpy over the entire flamelet. The two methods are compared in methane-air piloted turbulent premixed planar jet flames with increasing levels of dilution with both water and carbon dioxide that maintain a constant adiabatic flame temperature. The "product conversion" method is expected to mirror some of the same effects as physical dilution.

9:05AM A26.00006 A novel scaling approach for sooting laminar coflow flames at elevated pressures¹, AHMED ABDELGADIR, SCOTT A. STEINMETZ, ANTONIO ATTILI, King Abdullah University of Science and Technology (KAUST), FABRIZIO BISSETTI, University of Texas at Austin, USA, WILLIAM L. ROBERTS, King Abdullah University of Science and Technology (KAUST) — Laminar coflow diffusion flames are often used to study soot formation at elevated pressures due to their well-characterized configuration. In these experiments, these flames are operated at constant mass flow rate (constant Reynolds number) at increasing pressures. Due to the effect of gravity, the flame shape changes and as a result, the mixing field changes, which in turn has a great effect on soot formation. In this study, a novel scaling approach of the flame at different pressures is proposed. In this approach, both the Reynolds and Grashof's numbers are kept constant so that the effect of gravity is the same at all pressures. In order to keep the Grashof number constant, the diameter of the nozzle is modified as pressure varies. We report both numerical and experimental data proving that this approach guarantees the same nondimensional flow fields over a broad range of pressures. In the range of conditions studied, the Damkoehler number, which varies when both Reynolds and Grashof numbers are kept constant, is shown to play a minor role. Hence, a set of suitable flames for investigating soot formation at pressure is identified.

¹This research made use of the resources of IT Research Computing at King Abdullah University of Science & Technology (KAUST), Saudi Arabia

9:18AM A26.00007 DNS of turbulent premixed slot flames with mixture inhomogeneity: a study of NO_x formation, STEFANO LUCA, KAUST, Saudi Arabia, ANTONIO ATTILI, RWTH, Aachen University, Germany, FABRIZIO BISSETTI, University of Texas at Austin, USA — A set of Direct Numerical Simulations of three-dimensional methane/air lean flames in a spatially developing turbulent slot burner are performed. The flames are in the thin-reaction zone regimes and the jet Reynolds number is 5600. This configuration is of interest since it displays turbulent production by mean shear as in real devices. The gas phase hydrodynamics are modeled with the reactive, unsteady Navier-Stokes equations in the low Mach number limit. Combustion is treated with finite-rate chemistry. The jet is characterized by a non-uniform equivalence ratio at the inlet and varying levels of incomplete premixing for the methane/air mixture are considered. The global equivalence ratio is 0.7 and temperature is 800 K. All simulations are performed at 4 atm. The instantaneous profiles of the mass fractions of methane and air at the inlet are sampled from a set of turbulent channel simulations that provide realistic, fully turbulent fields. The data are analyzed to study the influence of partial premixing on the flame structure. Particular focus is devoted to the assessment of heat release rate fluctuations and NO_x formation. In particular, the effects of partial premixing on the production rates for the various pathways to NO_x formation are investigated.

Sunday, November 20, 2016 8:00AM - 9:57AM –
Session A27 Surface Tension: Analytical and Numerical Advances E147-148 - Boris Khusid,
New Jersey Institute of Technology

8:00AM A27.00001 Pressure-dependent surface viscosity and its surprising consequences in interfacial flows , HARISHANKAR MANIKANTAN, TODD SQUIRES, University of California, Santa Barbara

The surface shear viscosity of a surfactant monolayer almost always depends strongly on surface pressure, and this oft-ignored rheological feature significantly alters fluid flow and dynamics of particles on the interface. In order to illustrate the qualitatively new phenomena that arise out of pressure-dependent rheology, we focus here on a series of analytically tractable yet paradigmatic examples of lubrication geometries. Thin-gap flows naturally amplify pressure changes, and thus exemplify the effects of pressure-dependent viscosity. We show that much of the mathematical machinery from Newtonian lubrication analyses can be modified in a relatively straightforward manner in such systems. Our analysis reveals novel features such as a self-limiting flux when a surfactant is pumped through a narrow channel, a maximum approach velocity in squeeze flows due to divergent inter-particle forces, and forces perpendicular to the direction of motion that breaks symmetries associated with Newtonian analogs. We discuss the broader implications of these phenomena, especially with regard to interfacial suspension mechanics for which these lubrication geometries provide a convenient limit.

8:13AM A27.00002 Thermo/Soluto-capillary instabilities in evaporating bi-component liquid layers using DNS¹ , ADAM WILLIAMS, University of Edinburgh, PEDRO SENZ, Massachusetts

Institute of Technology, KHELLIL SEFIANE, PRASHANT VALLURI, University of Edinburgh — We investigate the stability, flow dynamics and evaporation kinetics of bi-component miscible liquid layers subject to a horizontal temperature gradient by means of two-phase direct numerical simulation. Flow is dominated by surface tension and driven by both thermal and solutal Marangoni effects, in which thermophoresis and mixture thermodynamics play a role. We employ a 3D model based on the Volume-of-Fluid method to account for the deformable liquid-gas interface. We note that the addition of a second species to the liquid phase affects the stability of the laterally heated layer over the single component case. We focus on systems with low Prandtl numbers ($\text{Pr} < 1$) and find that the layer is unstable for a certain critical Marangoni number, exhibiting the so-called hydrothermal waves. The structure of internal flow is a strong function of the instability and this we are also able to determine. Phase-separation/segregation within the liquid is more pronounced in evaporating systems, as is its effects on the hydrothermal waves. Our results show that solutal Marangoni convection is not only stronger in evaporating systems but also has a destabilising effect on the layer.

¹EPSRC - EP/K00963X/1

8:26AM A27.00003 Spreading and mixing of drops on a miscible liquid of different surface tension , S. AFKHAMI, I. SERIC, L. KONDIC, Department of Mathematical Sciences, New Jersey Institute of Technology, H. KIM, O. SHARDT, H. A. STONE, Department of Mechanical and Aerospace Engineering, Princeton University — We carry out Volume-of-Fluid based numerical simulations of a Marangoni-driven spreading of isopropyl alcohol (IPA) drops placed on water-air interface. The two fully miscible liquids create a spatially varying surface tension, leading to the spreading of the IPA drop on the water surface. We study the spreading of drops as IPA concentration is varied. In particular, we compute the spreading velocity and show that the scaling of the front position, L , with time, t , is given by $L \sim t^{0.7}$. We observe that while the surface tension difference between the two liquids controls the spreading velocity, it only slightly alters the power-law behavior for the range of considered IPA concentrations. We also provide detailed insight of the mixing of the IPA and water, and show the time evolution of liquid-air surface tension distribution. We show that the mixing results in a volume flux in a thin region on the surface, generating a vortical flow underneath the spreading front; we investigate the details of these flow patterns and show the time evolution of the circulation within the water. The numerical results are supported by new experimental observations reported separately.

8:39AM A27.00004 Draining Capillary Liquids from Containers with Interior Corners¹ , JOSH MCCRANEY, Student, MARK WEISLOGEL, Portland State University — A new solution is found for the late stage draining of a wetting capillary fluid in an interior corner. A formulation for slender flow along the interior base-corner of a right circular cylinder is presented, where a separation of variables solution offers a method to predict drain rates for this and related double-drain geometries. It is shown the maximum volumetric liquid removal rate is $Q \sim t^{-3}$, volume removed is $V \sim t^{-2}$, and nominal liquid depth is $h \sim t^{-1}$. Representative experimental results are presented to assess the quantitative value of the approach.

¹NASA NNX12A047A

8:52AM A27.00005 Dynamic Wetting Failure and Hydrodynamic Assist in Curtain Coating , CHEN-YU LIU, ERIC VANDRE, University of Minnesota, MARCIO CARVALHO, PUC-Rio, SATISH KUMAR, University of Minnesota — Dynamic wetting failure in curtain coating of Newtonian liquids is studied in this work. A hydrodynamic model accounting for air flow near the dynamic contact line (DCL) is developed to describe two-dimensional (2D) steady wetting and to predict the onset of wetting failure. A hybrid approach is used where air is described by a one-dimensional model and liquid by a 2D model, and the resulting hybrid formulation is solved with the Galerkin finite element method. The results reveal that the delay of wetting failure in curtain coating—often termed hydrodynamic assist—mainly arises from the hydrodynamic pressure generated by the inertia of the impinging curtain. This pressure leads to a strong capillary-stress gradient that pumps air away from the DCL and thus increases the critical substrate speed for wetting failure. Although the parameter values used in the model are different from those in experiments due to computational limitations, the model is able to capture the experimentally observed non-monotonic behavior of the critical substrate speed as the feed flow rate increases [T. D. Blake et al., Phys. Fluids, 11(8), 1995 (1999)].

9:05AM A27.00006 ABSTRACT WITHDRAWN —

9:18AM A27.00007 Marangoni flows induced by A + B → C reaction fronts with arbitrary diffusion coefficients¹ , REDA TIANI, LAURENCE RONGY, Université libre de Bruxelles — We consider horizontal aqueous solutions in contact with air where three reacting species A, B, and C can affect the surface tension of the solution, thereby driving Marangoni flows. When the two reactants A and B, that are initially separated, are brought into contact, a reaction front producing species C is formed and evolves in time due to diffusion, convection and reaction processes. The resulting dynamics is studied by numerically integrating the incompressible Navier-Stokes equations coupled to reaction-diffusion-convection equations for the three chemical species. For equal initial concentrations of reactants and equal diffusion coefficients, we have explained how chemically-driven Marangoni flows can lead to complex dynamics of the front propagation. Here we extend such results for arbitrary values of the diffusion coefficients and initial concentrations of reactants. We give the general classification of the surface tension profiles as a function of the Marangoni numbers quantifying the effect of each species on the surface tension, the ratio of initial concentrations of reactants and the ratios of diffusion coefficients. Such a classification allows us then to study the resulting structure of the convective rolls as well as the nonlinear dynamics of the reaction front.

¹F.R.S.- FNRS, ARC

9:31AM A27.00008 Dynamics of two-phase interfaces and surface tensions: A density-functional theory perspective¹, PETR YATSYSHIN, Department of Chemical Engineering, Imperial College London, London, UK, DAVID N. SIBLEY, Department of Mathematical Sciences, Loughborough University, Loughborough, UK; Department of Chemical Engineering, Imperial College London, London, UK, MIGUEL A. DURAN-OLIVENCIA, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, London, UK — Classical density functional theory (DFT) is a statistical mechanical framework for the description of fluids at the nanoscale, where the inhomogeneity of the fluid structure needs to be carefully accounted for. By expressing the grand free-energy of the fluid as a functional of the one-body density, DFT offers a theoretically consistent and computationally accessible way to obtain two-phase interfaces and respective interfacial tensions in a ternary solid-liquid-gas system. The dynamic version of DFT (DDFT) can be rigorously derived from the Smoluchowsky picture of the dynamics of colloidal particles in a solvent. It is generally agreed that DDFT can capture the diffusion-driven evolution of many soft-matter systems. In this context, we use DDFT to investigate the dynamic behaviour of two-phase interfaces in both equilibrium and dynamic wetting and discuss the possibility of defining a time-dependent surface tension, which still remains in debate.

¹We acknowledge financial support from the European Research Council via Advanced Grant No. 247031 and from the Engineering and Physical Sciences Research Council of the UK via Grants No. EP/L027186 and EP/L020564

9:44AM A27.00009 Variational Methods For Sloshing Problems With Surface Tension.

CHEE HAN TAN, MAX CARLSON, CHRISTEL HOHENEGGER, BRAXTON OSTING, Univ of Utah — We consider the sloshing problem for an incompressible, inviscid, irrotational fluid in a container, including effects due to surface tension on the free surface. We restrict ourselves to a constant contact angle and we seek time-harmonic solutions of the linearized problem, which describes the time-evolution of the fluid due to a small initial disturbance of the surface at rest. As opposed to the zero surface tension case, where the problem reduces to a partial differential equation for the velocity potential, we obtain a coupled system for the velocity potential and the free surface displacement. We derive a new variational formulation of the coupled problem and establish the existence of solutions using the direct method from the Calculus of Variations. In the limit of zero surface tension, we recover the variational formulation of the classical Steklov eigenvalue problem, as derived by B. A. Troesch. For the particular case of an axially symmetric container, we propose a finite element numerical method for computing the sloshing modes of the coupled system. The scheme is implemented in FEniCS and we obtain a qualitative description of the effect of surface tension on the sloshing modes.

Sunday, November 20, 2016 8:00AM - 9:57AM –

Session A28 Particle-laden Flows: General I F149 - Sarma Rani, University of Alabama in Huntsville

8:00AM A28.00001 Effects of Deterministic and Stochastic Forcing Schemes on Inertial Particle Statistics in DNS of Isotropic Turbulence, ROHIT DHARIWAL, KIRUTHIKA SUNDARARAJAN, SARMA L. RANI, Univ of Alabama - Huntsville — In DNS of isotropic turbulence, statistical stationarity is achieved through a forcing scheme that supplies energy to the large scales. These schemes may be broadly classified into deterministic and stochastic forcing schemes. In the deterministic scheme, the turbulent kinetic energy dissipated during a time step is resupplied, whereas in stochastic schemes, forcing is determined based on the evolution of Ornstein-Uhlenbeck processes. Both approaches add the forcing within a band of wavenumbers at the low-wavenumber end of the energy spectrum. The goal of this study is to investigate the effects of the two forcing schemes on the flow, and on the relative motion statistics of inertial particles in forced isotropic turbulence. An important parameter in stochastic forcing is the forcing time scale T_F . DNS was performed using both forcing schemes for $T_F = T_E/4, T_E/2, T_E, 2T_E, 4T_E$. Here T_E is the large eddy turnover time obtained from the DNS with deterministic forcing. Three Taylor micro-scale Reynolds numbers $Re_\lambda = 76, 131, 195$, and twelve particle Stokes numbers based on the Kolmogorov time-scale, $St_\eta = 0.05$ to 40 are considered. Detailed analysis of the effects of forcing time scales on both fluid and particle statistics is undertaken.

8:13AM A28.00002 Comparison of Stochastic Theory and DNS for the Relative Motion of High-Inertia Particle Pairs in Isotropic Turbulence, SARMA RANI, ROHIT DHARIWAL, Univ of Alabama - Huntsville, DONALD KOCH, Cornell University — In an earlier work, we derived closures in the limit of high Stokes number for the diffusivity tensor in the PDF equation for particle pairs. The diffusivity contained the time integral of the Eulerian two-time correlation of fluid relative velocities seen by pairs that are nearly stationary. The two-time correlation was analytically resolved through the approximation that the temporal change in the fluid relative velocities seen by a pair occurs principally due to the advection of smaller eddies past the pair by large scale eddies. Two diffusivity expressions were obtained based on whether the pair center of mass remained fixed during flow time scales, or moved in response to integral-scale eddies. A quantitative analysis of the stochastic theory is performed through a comparison of the pair statistics obtained using Langevin simulations with those from DNS. Langevin simulations of particle pair dispersion were performed using the diffusivity closures for four particle Stokes numbers based on the Kolmogorov time-scale, $St_\eta = 10, 20, 40, 80$ and at two Taylor micro-scale Reynolds numbers $Re_\lambda = 76, 131$. Statistics such as RDF, PDF, variance and kurtosis of particle-pair relative velocities were computed using both Langevin and DNS runs, and compared.

8:26AM A28.00003 Effects of Reynolds Number and Stokes Number on Particle-pair Relative Velocity in Isotropic Turbulence: An Experimental Study¹, ZHONGWANG DOU, University at Buffalo - SUNY, ANDREW BRAGG, Duke University, ADAM HAMMOND, ZACH LIANG, University at Buffalo - SUNY, LANCE COLLINS, Cornell University, HUI MENG, University at Buffalo - SUNY — Effects of Reynolds number (R_λ) and Stokes number (St) on particle-pair relative velocity (RV) were studied using four-frame particle tracking in an enclosed turbulence chamber. Two tests were performed: varying R_λ between 246 and 357 at six St values, and varying St between 0.02 and 4.63 at five R_λ values. By comparing experimental and DNS results of mean inward particle-pair RV, $\langle w_r^- \rangle$, we observed excellent agreement for all test conditions across a large range of particle separation distance (r); however at $r \leq 10\eta$ (η : Kolmogorov length scale), experimental $\langle w_r^- \rangle$ values were higher than simulation. At fixed St , $\langle w_r^- \rangle$ was found to be independent of R_λ in the observable St , r , and R_λ ranges. At fixed R_λ , $\langle w_r^- \rangle$ increased with St at small r and decreased with St at large r . We further compared $\langle w_r^- \rangle$ and variance of RV, $\langle w_r^2 \rangle$, between experiments, DNS and theoretical predictions by Pan and Padoan (2010). At $0 < St \leq 1$, theory-predicted $\langle w_r^- \rangle$ and $\langle w_r^2 \rangle$ matched with DNS and experiment in the range of $r = 1 - 60\eta$. As St increased, theoretical predictions were lower than experiment and DNS results. The potential causes of these trends are explored. Additionally, we discuss the observed electrostatic charge effect on particle relative motion in isotropic turbulence and our plans of studying this effect using an integrated experimental, numerical and theoretical approach.

¹This work was supported by NSF CBET-0967407 and CBET-0967349

8:39AM A28.00004 Effects of elevated line sources on turbulent mixing in channel

flow , QUOC NGUYEN, DIMITRIOS PAPAVASSILIOU, Univ of Oklahoma — Fluids mixing in turbulent flows has been studied extensively, due to the importance of this phenomena in nature and engineering. Convection effects along with motion of three-dimensional coherent structures in turbulent flow disperse a substance more efficiently than molecular diffusion does on its own. We present here, however, a study that explores the conditions under which turbulent mixing does not happen, when different substances are released into the flow field from different vertical locations. The study uses a method which combines Direct Numerical Simulation (DNS) with Lagrangian Scalar Tracking (LST) to simulate a turbulent channel flow and track the motion of passive scalars with different Schmidt numbers (Sc). The particles are released from several instantaneous line sources, ranging from the wall to the center region of the channel. The combined effects of mean velocity difference, molecular diffusion and near-wall coherent structures lead to the observation of different concentrations of particles downstream from the source. We then explore in details the conditions under which particles mixing would not happen. Results from numerical simulation at friction Reynolds number of 300 and 600 will be discussed and for Sc ranging from 0.1 to 2,400.

8:52AM A28.00005 Rolling and sliding motion of spheres propagating down inclined planes in still water¹ , YI HUI TEE, ELLEN LONGMIRE, Aerospace Engineering and Mechanics Department, University of Minnesota — In modelling the motion of spheres submerged in liquid, gravity, drag, lift, and added mass forces have to be taken into account.

For spheres contacting bounding surfaces, friction coefficients due to rolling and sliding increase the complexity of the model. In this study, experiments are conducted to investigate the effects of particle density and diameter on the rolling and sliding motion of spheres. Spherical particles with marked surfaces are released from rest on an inclined glass plate in still water at various inclination angles and allowed to accelerate. A 45° mirror mounted beneath the plate allows simultaneous capture of both longitudinal and spanwise motions of the sphere. Based on sequences obtained by high speed imaging, the translational and rotational velocities are determined. Particle Reynolds numbers at terminal velocity range from 400 to 2500 corresponding with Galileo numbers of 800 to 2800. By comparing the translational and rotational velocities, the occurrence of sliding motion can be identified. The onset of sliding motion is then determined as a function of inclination angle and Galileo number for multiple particle materials. The experimental results are also compared against the existing models from the literature.

¹Supported by NSF (CBET-1510154).

9:05AM A28.00006 Shear resuspension and rheology of dense particles at moderate and large Re , ESPERANZA LINARES-GUERRERO, MELANY HUNT, California Institute of Technology, ROBERTO ZENIT, Universidad Nacional Autonoma de Mexico — We experimentally investigate the resuspension behavior of dense particles subjected to lateral shear in a Couette device for particle Reynolds numbers from 15 to 500. Before the shear is applied, the particles sediment and form a compact bed that sits in the bottom of an annular ring of the apparatus. At sufficiently high shear rate, applied by rotating the inner cylinder, the particles re-suspend and the bed expands reaching a steady state. The mean volume fraction of the bed is determined from the initial bed height, the Archimedes number, and the Stokes number. The measurements are compared with a model that predicts the bed expansion by considering a balance between the rate of settling and a shear induced Fickian flux for moderate to high Reynolds numbers. Good agreement was found between experiments and predictions, considering values of the diffusion coefficient found in the literature. Once the column has resuspended, a measurement of the effective viscosity is performed. We discuss the implications of the sedimenting granular phase on the rheology of the suspension.

9:18AM A28.00007 Fingering instability of a suspension film spreading on a spinning disk , MAYURESH KULKARNI, SUBHADARSHINEE SAHOO, Chemical Engineering Division, CSIR-National Chemical Laboratory, Pune, India 411008, PANKAJ DOSHI, Pfizer, Inc., Groton, Connecticut, USA, ASHISH ORPE, Chemical Engineering Division, CSIR-National Chemical Laboratory, Pune, India 411008 — We have experimentally investigated the spreading of a suspension drop when rotated atop a spinning disk using flow visualization techniques. The suspension is made of 50 ± 10 micron glass beads suspended in a low viscosity, partially wetting Newtonian liquid having same density as the glass beads. The suspension drop is placed centrally on a horizontal disc and the disc is then rotated at a desired speed. The spreading behavior is captured using a high speed camera and the acquired images are analysed to find the edges of the spreading film. For all the particle volume fractions (ϕ_p) studied, the suspension drops spread radially until they reach a critical radius, following which the contact line develops instabilities which further grow into fingers. The critical radius for the onset of instability shows an increase with increase in the particle fraction (ϕ_p) before decreasing slightly at the highest value of ϕ_p studied, while the instability wavelength (λ) exhibits a non-monotonic dependence. The value of λ is close to that for a partially wetting liquid at lower ϕ_p , it decreases with increasing ϕ_p to a minimum before increasing again at the largest ϕ_p .

9:31AM A28.00008 A collision model for simulating dense suspensions , EDWARD BIEGERT, BERNHARD VOWINCKEL, ECKART MEIBURG, University 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 will present the collision modeling strategy for our code PARTIES (PARTicle-laden flows via immersed boundarLES), which includes lubrication, normal contact, and tangential contact forces. While our strategy is based on other collision models, we will highlight several improvements we have made and then demonstrate the effectiveness of the collision model in reproducing experimental results for binary particle-wall collisions as well as bulk transport rates for a laminar shear flow over a bed of thousands of spheres.

9:44AM A28.00009 A settling-driven instability in two-component, stably stratified fluids , AHMAD ALSINAN, ECKART MEIBURG, UC Santa Barbara, PASCALE GARAUD, UC Santa Cruz — We analyze the stability of stably stratified fluids whose density depends on two scalar fields, for situations where one of the scalar fields is unsteadily stratified and involves a settling velocity. Such conditions may be found, for example, in flows involving the transport of sediment and heat or salt. A linear stability analysis for constant-gradient base states demonstrates that the settling velocity generates a phase shift between the perturbation fields of the two scalars, which gives rise to a novel instability mode. This instability mechanism favors the growth of waves that are inclined with respect to the horizontal direction. It is active for all density and diffusivity ratios, including for cases in which the two scalars diffuse at identical rates. If the scalars have different diffusivities, the new instability mechanism competes with the dominant elevator mode of the classical double-diffusive instability. We present linear stability results as a function of the governing dimensionless parameters, including for lateral gradients of the base state density fields that result in predominantly horizontal intrusion instabilities. Highly resolved direct numerical simulation results serve to illustrate the nonlinear competition of the various instabilities for such flows in different parameter regimes.

Sunday, November 20, 2016 8:00AM - 9:31AM –
Session A29 CFD: Applications | F150 - Sourabh Apte, Oregon State University

8:00AM A29.00001 Numerical study on the interaction between supercavitation and turbulence, HAN LIU, University of Minnesota and Peking University, ZUOLI XIAO, Peking University, LIAN SHEN, University of Minnesota — SUpercavitation uses a bubble of gas inside a liquid large enough to encompass an object travelling through the liquid so that the skin friction on the object can be greatly reduced and high speed can be obtained. In this study, computational fluid dynamics is used to investigate the interaction between supercavitation and turbulence. The study builds on an in-house simulation code that uses the coupled level set and volume of fluid method to accurately capture the interface between the water and gas phases. A ventilated disk cavitator is used for the bubble generation, and it is modelled by a sharp interface immersed boundary method. Turbulence in the incoming flow is generated by a grid of small spheres upstream. Based on the simulation data, the influence of turbulence on the supercavitation and the underlying mechanisms are analyzed.

8:13AM A29.00002 Sprocket- Chain Simulation: Modelling and Simulation of a Multi Physics problem by sequentially coupling MotionSolve and nanoFluidX, ADITYA JAYANTHI, CHRISTOPHER COKER¹, Altair Engineering Inc — In the last decade, CFD simulations have transitioned from the stage where they are used to validate the final designs to the main stream development of products driven by the simulation. However, there are still niche areas of applications like oiling simulations, where the traditional CFD simulation times are prohibitive to use them in product development and have to rely on experimental methods, which are expensive. In this paper a unique example of Sprocket- Chain simulation will be presented using nanoFluidx a commercial SPH code developed by FluiDyna GmbH and Altair Engineering. The grid less nature of the SPH method has inherent advantages in the areas of application with complex geometry which pose severe challenge to classical finite volume CFD methods due to complex moving geometries, moving meshes and high resolution requirements leading to long simulation times. The simulations times using nanoFluidx can be reduced from weeks to days allowing the flexibility to run more simulation and can be used in main stream product development. The example problem under consideration is a classical Multiphysics problem and a sequentially coupled solution of Motion Solve and nanoFluidX will be presented.

¹This abstract is replacing DFD16-2016-000045

8:26AM A29.00003 Multi-fidelity modelling for flow over a cylinder, PRERNA PATIL, Brown University, HESSAM BABAEE, Massachusetts Institute of Technology, GEORGE KARNIADAKIS, Brown University — We tackle the classical problem of predicting the relation between C_L , C_D and C_P vs Reynolds number for flow over cylinder using the multi-fidelity framework. The stochastic response surface is obtained by implementing the auto-regressive stochastic modelling (Kennedy and O'Hagan, 2000) and Gaussian process regression to combine data from variable levels of fidelity. In particular, we predict the lift, drag and pressure coefficients where codes with multiple levels of fidelity are available. We correlate data at each of these levels and build the surrogate model using multi-level recursive co-kriging. The deficient physics of the low-fidelity model is explored by examining the cross-correlation between the low-fidelity and high-fidelity models. The proposed framework ultimately intends to meld computational accuracy of the expensive high fidelity with the computational cost of the inexpensive low-fidelity.

8:39AM A29.00004 Electrohydrodynamic deformation of capsules in electric field, SUDIP DAS, ROCHISH THAOKAR, Indian Institute of Technology Bombay — Micron size capsules are abundant in natural, technological and biological processes but they still require extensive investigation for better understanding of their mechanical behavior. A spherical capsule containing a Newtonian fluid bounded by a viscoelastic membrane and immersed in another Newtonian fluid, and subject to electric field is considered. Discontinuity of electrical properties such as conductivity and permittivity leads to a net Maxwell stress at the capsule interface. In response the capsule undergoes elastic deformation, leading to strain fields and elastic stresses that can balance the applied forces. We investigate this problem with fully resolved hydrodynamics in the Stokes flow limit and electrostatics using the capacitance model. Effect of AC, DC and pulsed DC fields is investigated. Our results show that membrane electrical properties have a huge impact on the equilibrium deformation as well as on the break up of capsules. Our results match with the literature results in the limit of high conductance of the membrane. Analytical theory is employed using spherical harmonics and numerical investigations are conducted using the Boundary integral method.

8:52AM A29.00005 ABSTRACT WITHDRAWN —

9:05AM A29.00006 Convective heat transfer by oscillating flow in an enclosure with non-uniform spatial bottom wall temperature profile, SAEID RAHEIMPOUR ANGENEH, MURAT KADRI AKTAS, TOBB University of Economics and Technology — Effects of the acoustic streaming motion on convective heat transfer in a rectangular shallow enclosure with sinusoidal spatial bottom wall temperature distribution are investigated numerically. Acoustic excitation is generated by the periodic vibration of left wall. The top wall of the enclosure is isothermal while the side walls are adiabatic. A FORTRAN code is developed to predict the oscillatory and mean flow fields by considering the compressible form of the Navier -Stokes equation and solved by flux-corrected transport algorithm. In order to validate the results of the simulations, a case with an unheated bottom wall is considered and compared with the existing literature. Applying the sinusoidal temperature profile to the bottom wall provides axial and transverse temperature gradients. In return these gradients strongly affect the flow pattern in the enclosure. Heat transfer depends on the flow structure considerably. This is the first time that the effect of nonzero mean vibrational flow on thermal convection from a surface with sinusoidal temperature profile investigated. Results of this study may lead up to design of new heat removal applications.

9:18AM A29.00007 Computing the numerical solution to functional differential equations: some recent progresses towards E. Hopf's 1952 dream, DANIELE VENTURI, University of California Santa Cruz — The fundamental importance of functional differential equations has been recognized in many areas of mathematical physics, such as fluid dynamics, quantum field theory and statistical physics. For example, in the context of fluid dynamics, the Hopf characteristic functional equation was deemed by Monin and Yaglom to be "the most compact formulation of the turbulence problem", which is the problem of determining the statistical properties of the velocity and pressure fields of Navier-Stokes equations given statistical information on the initial state. However, no effective numerical method has yet been developed to compute the solution to functional differential equations. In this talk I will provide a new perspective on this general problem, and discuss recent progresses in approximation theory for nonlinear functionals and functional equations. The proposed methods will be demonstrated through various examples.

Sunday, November 20, 2016 8:00AM - 9:57AM —

Session A30 Granular Flows: Jamming and Drag F151 - Dong Wang, Duke University

8:00AM A30.00001 Origins of Shear Jamming for Frictional Grains¹, DONG WANG, Duke University, HU ZHENG, Hohai University, JIE REN, Merck & Co., JOSHUA DIJKSMAN, Wageningen University, JONATHAN BARES, Laboratoire de Mecanique et de Genie Civil, Universite de Montpellier, ROBERT BEHRINGER, Duke University — Granular systems have been shown to be able to behave like solids, under shear, even when their densities are below the critical packing fraction for frictionless isotropic jamming. To understand such a phenomena, called shear jamming, the question we address here is: how does shear bring a system from a unjammed state to a jammed state, where the coordination number, Z , is no less than 3, the isotropic jamming point for frictional grains? Since Z can be used to distinguish jammed states from unjammed ones, it is vital to understand how shear increases Z . We here propose a set of three particles in contact, denoted as a trimer, as the basic unit to characterize the deformation of the system. Trimers, stabilized by inter-grain friction, fail under a certain amount of shear and bend to make extra contacts to regain stability. By defining a projection operator of the opening angle of the trimer to the compression direction in the shear, O , we see a systematically linear decrease of this quantity with respect to shear strain, demonstrating the bending of trimers as expected. In addition, the average change of O from one shear step to the next shows a good collapse when plotted against Z , indicating a universal behavior in the process of shear jamming.

¹We acknowledge support from NSF DMR1206351, NASA NNX15AD38G, the William M. Keck Foundation and a RT-MRSEC Fellowship

8:13AM A30.00002 Shear jamming in highly strained granular system without shear banding¹, YIQIU ZHAO, Duke University, JONATHAN BARS, Universit de Montpellier, HU ZHENG, ROBERT 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 $\phi_S < \phi < \phi_J$ 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 shear band. We present a novel 2D periodic shear apparatus made of 21 independent, aligned and mirrored glass rings. Each 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. By performing different shear profiles for different packing fractions we explored the details of jamming diagram including the location of the yield surface.

¹This work is supported by NSF No.DMR1206351, NASA No.NNX15AD38G and W. M. Keck Foundation.

8:26AM A30.00003 Jamming transition of angular shaped particles under compression¹, CACEY STEVENS BESTER, YIQIU ZHAO, YUANYUAN XU, MEREDITH COX, ROBERT BEHRINGER, Department of Physics, Duke University — A fundamental challenge of understanding the global behavior of granular assemblies is to determine the effect of local particle properties, such as particle shape. Here we investigate how particle shape influences the jamming transition of granular packings by comparing the response of systems of angular shaped particles to that of disks under isotropic compression. These experiments are performed using two-dimensional arrangements of photoelastic particles, allowing us to visualize the change in force propagation during the jamming transition. We find qualitative and quantitative differences in the macroscopic responses of the systems with changing particle shape. We compare the packing fraction and the contact number evolution of compression experiments as we vary particle shape. The pair correlation function also shows a different geometric feature with particle shape. Using cyclic compression, we additionally explore the stress relaxation and dynamical heterogeneity of the particles.

¹Duke University Provost's Postdoctoral Program, NASA grant NNX15AD38G, NSF-DMR1206351, DMS1248071, and the W.M. Keck Foundation

8:39AM A30.00004 Clogging and Intermittent Flow in a 2D Hopper, KERSTIN NORDSTROM, EMMA THACKRAY, Mount Holyoke College — We have constructed a quasi-two-dimensional system of bidisperse, millimeter-scale disks with photoelastic properties that make force networks within the material visible. The system is contained in an acrylic box with an adjustable bottom opening. We can approach the clogging transition by adjusting this opening and by adding external forcing to the top of the flowing pile. By placing the system between cross-polarizers, we can obtain high-speed video of this system during flow, and extract intensity signals that can be used to identify and quantify localized, otherwise indeterminate forces. We simultaneously track individual particle motions, which can be used to identify shear transformation zones in the system. We are therefore able to correlate local forces with rearrangements within the system, and characterize the evolution of this interplay on the approach to the clogging transition.

8:52AM A30.00005 Characterizing Three Dimensional Granular Materials¹, DAVID CHEN, Duke University, JONATHAN BARES, University of Montpellier, HU ZHENG, CASEY BESTER, ROBERT BEHRINGER, Duke University — We use systems of hydrogel particles to determine the microscopic response of 3D granular systems to deformations near jamming. We visualize the particles using a laser scan technique, and we determine the motion of the particles along with their inter-particle forces and contacts from the reconstructed scans. We focus on their response to shear with low friction.

¹NSF-DMF-1206351, NASA NNX15AD38G

9:05AM A30.00006 Couette shear of an ideal 2D photo-elastic granular system¹, ROBERT BEHRINGER, HU ZHENG, Duke University, JONATHAN BARS, Duke University; Universit de Montpellier, DONG WANG, Duke University — In this study, Couette shear experiments are conducted using 2D photoelastic granular particles, which allows us to apply infinite shear strain to the granular system. We obtain force information at the granular scale using the calibrated photo-elastic grain force response. The whole granular system is density matched in salt solution, which guarantees an ideal 2D system without basal friction between the particles and the table. The viscosity is negligible at the very small shear strain rate (0.017 rpm). This talk will address two main points: i) how does the system reach a jammed state; ii) how does system reach a long term stable state and what are the properties of that state.

¹We acknowledge support from NSF Grant No. DMR1206351, NASA Grant No. NNX15AD38G and the W.M. Keck Foundation.

9:18AM A30.00007 Slip-stick excitation and travelling waves excite silo honking, NATHALIE VRIEND, KASIA WARBURTON, University of Cambridge, ELZE PORTE, Imperial College London — Industrial storage silos filled with PET-particles can create a sound upon discharge. The sound forms a nuisance for the environment when the structure starts to act as a loudspeaker and may ultimately result in structural failure. This work investigates the phenomenon experimentally—the deployment of a microphone, an accelerometer and high-speed imaging on a laboratory set-up reveal the driving mechanism for the structural resonance: stick-slip at the wall. Particle image velocimetry shows an asymmetric, upwards travelling wave (at 50 m/s) which contains the dynamic slip-region. The frequency of the mechanical motion of the grains is successfully correlated to the frequency of the emitted sound. Friction models are explored to describe and quantify the frictional interaction between the grains and the wall.

9:31AM A30.00008 A Fluidic Hourglass¹ , ALVARO MARIN, Bundeswehr University Munich, HENRI LHUISSIER,

IUSTI, Aix-Marseille Université, MASSIMILIANO ROSSI, ANDREAS VOLK, CHRISTIAN J. KÄHLER, Bundeswehr University Munich — A group of objects passing through a constriction might get eventually stuck. It occurs no matter what type of object is considered: sand in an hourglass, particles in a fluid through a porous medium or people leaving a room in panic. The case of particles in a fluid affects porous mediums, filters and membranes, which become unusable when clogged. Certainly the adherence of the particles to the walls and to each other is an important parameter in such systems (Wyss et al., Phys. Rev. E, 2006), but even without adherence the clogging probability is far from negligible. Focusing in these low-adherence regimes, we use microfluidic devices with a bottleneck of squared cross-section through which we force dilute polystyrene particle solutions with diameters comparable to the bottleneck size and down to one tenth its size. In such low friction conditions we show experimental evidence of a strong transition at a critical particle-to-neck ratio, just as it occurs in dry granular systems (Zuriguel et al., Phys. Rev. E, 2003). We describe analytically such a transition by modeling the arch formation as a purely stochastic process, which yields a good agreement with the experimental data.

¹Deutsche Forschungsgemeinschaft KA1808/22-1

9:44AM A30.00009 Drag reduction by rotation in granular media. , WONJONG JUNG, SUNG MOK CHOI, Seoul National Univertisy, WONJUNG KIM, Sogang University, HO-YOUNG KIM, Seoul National Univertisy — We present

quantitative measurements and mathematical analysis of the granular drag reduction by rotation inspired by some self-burrowing seeds whose morphologies respond to environmental changes in humidity. The seeds create a motion to dig into soil using their moisture-responsive awns, which are basically helical shaped in a dry environment but reversibly deform to a linear shape in a humid environment. When the tip of the awn is fixed by an external support, the hygroscopic deformation of the awn gives the seed a thrust with rotation against the soil. By measuring the granular drag of vertically penetrating intruders with rotation, we find the drag to decrease with its rotation speed. Noting that the relative motions of the grains in contact with the intruder induce the collapse of the force chains in the granular bulk, we develop a general correlation for the drag reduction by rotation in terms of the relative slip velocity of the grains, which successfully explains the drag reduction of the rotating intruders including self-burrowing rotary seeds.

Sunday, November 20, 2016 8:00AM - 9:57AM –

**Session A31 Experimental Techniques - 2D Particle Velocimetry F152 - Ronald Adrian,
Arizona State University**

8:00AM A31.00001 On the rms errors and dynamic ranges of triple- and quadruple-pulse particle tracking velocimetry (PTV)¹ , LIUYANG DING, RONALD ADRIAN, Arizona State University, SIVARAM GOGINENI, Spectral Energies LLC — Multi-pulse PTV extends conventional dual-pulse PTV by fitting a polynomial to particle locations measured from three or four pulses in a burst, aiming at more accurately resolving a particle short-period trajectory. Particle velocity and acceleration are then evaluated at an optimal time minimizing rms errors. Numerical simulations were performed to completely study the behaviors of position, velocity, and acceleration rms errors of triple- and quadruple-pulse PTV in a 4-D space spanned by four dimensionless variables – normalized time, normalized displacement, normalized particle locating noise, and acceleration factor. We compared three analysis methods – 3-pulse with quadratic fitting, 4-pulse with cubic fitting and 4-pulse with quadratic least-square fitting. In addition, generalized definitions of dynamic spatial range (DSR) and dynamic velocity range (DVR) are proposed for multi-pulse analyses. We calculated DSR ratios and DVR ratios between the multi-pulse and 2-pulse under various flow conditions and noise levels. It is found that the DSR and DVR could be improved by up to 100 times and 10 times, respectively, when the particle trajectory is strongly curved, deceleration is pronounced, and particle locations are accurately determined.

¹This work is supported by ONR N00014-14-C-0095.

8:13AM A31.00002 Uncertainty Quantification and Statistical Convergence Guidelines for PIV Data , MATTHEW STEGMER, DAN KASSEN, TSI Inc — As Particle Image Velocimetry has continued to mature, it has developed into a robust and flexible technique for velocimetry used by expert and non-expert users. While historical estimates of PIV accuracy have typically relied heavily on “rules of thumb” and analysis of idealized synthetic images, recently increased emphasis has been placed on better quantifying real-world PIV measurement uncertainty. Multiple techniques have been developed to provide per-vector instantaneous uncertainty estimates for PIV measurements. Often real-world experimental conditions introduce complications in collecting “optimal” data, and the effect of these conditions is important to consider when planning an experimental campaign. The current work utilizes the results of PIV Uncertainty Quantification techniques to develop a framework for PIV users to utilize estimated PIV confidence intervals to compute reliable data convergence criteria for optimal sampling of flow statistics. Results are compared using experimental and synthetic data, and recommended guidelines and procedures leveraging estimated PIV confidence intervals for efficient sampling for converged statistics are provided.

8:26AM A31.00003 Uncertainty Estimation for 2D PIV: An In-Depth Comparative Analysis , AARON BOOMSMA, TSI Inc, SYANTAN BHATTACHARYA, Purdue University, DAN TROOLIN, TSI Inc, PAVLOS VLACHOS, Purdue University, STAMATIOS POTROS, TSI Inc — Uncertainty quantification methods have recently made great strides in accurately predicting uncertainties for planar PIV, and several different approaches are now documented. In the present study, we provide an analysis of these methods across different experiments and different PIV processing codes. To assess the performance of said methods, we follow the approach of Sciacchitano et al. (2015) and utilize two PIV measurement systems with overlapping fields of view—one acting as a reference (which is validated using simultaneous LDV measurements) and the other as a measurement system, paying close attention to the effects of interrogation window overlap and bias errors on the analysis. A total of three experiments were performed: a jet flow and a cylinder in cross flow at two Reynolds numbers. In brief, the standard coverages (68% confidence interval) ranged from approximately 65%-77% for PPR and MI methods, 40%-50% for image matching methods. We present an in-depth survey of both global (e.g., coverage and error histograms) and local (e.g., spatially varying statistics) parameters to examine the strengths and weaknesses of each method in monitor their responses to different regions of the experimental flows.

8:39AM A31.00004 Adaptive spectral filtering of PIV cross correlations, MATTHEW GIARRA, Virginia Tech , PAVLOS VLACHOS, Purdue University, AETHER LAB TEAM — Using cross correlations (CCs) in particle image velocimetry (PIV) assumes that tracer particles in interrogation regions (IRs) move with the same velocity. But this assumption is nearly always violated because real flows exhibit velocity gradients, which degrade the signal-to-noise ratio (SNR) of the CC and are a major driver of error in PIV. Iterative methods help reduce these errors, but even they can fail when gradients are large within individual IRs. We present an algorithm to mitigate the effects of velocity gradients on PIV measurements. Our algorithm is based on a model of the CC, which predicts a relationship between the PDF of particle displacements and the variation of the correlation's SNR across the Fourier spectrum. We give an algorithm to measure this SNR from the CC, and use this insight to create a filter that suppresses the low-SNR portions of the spectrum. Our algorithm extends to the ensemble correlation, where it accelerates the convergence of the measurement and also reveals the PDF of displacements of the ensemble (and therefore of statistical metrics like diffusion coefficient). Finally, our model provides theoretical foundations for a number of "rules of thumb" in PIV, like the quarter-window rule.

8:52AM A31.00005 Application of photogrammetry to transforming PIV-acquired velocity fields to a moving-body coordinate system¹, POURYA NIKOUEEYAN, JONATHAN NAUGHTON, University of Wyoming — Particle Image Velocimetry is a common choice for qualitative and quantitative characterization of unsteady flows associated with moving bodies (e.g. pitching and plunging airfoils). Characterizing the separated flow behavior is of great importance in understanding the flow physics and developing predictive reduced-order models. In most studies, the model under investigation moves within a fixed camera field-of-view, and vector fields are calculated based on this fixed coordinate system. To better characterize the genesis and evolution of vortical structures in these unsteady flows, the velocity fields need to be transformed into the moving-body frame of reference. Data converted to this coordinate system allow for a more detailed analysis of the flow field using advanced statistical tools. In this work, a pitching NACA0015 airfoil has been used to demonstrate the capability of photogrammetry for such an analysis. Photogrammetry has been used first to locate the airfoil within the image and then to determine an appropriate mask for processing the PIV data. The photogrammetry results are then further used to determine the rotation matrix that transforms the velocity fields to airfoil coordinates. Examples of the important capabilities such a process enables are discussed.

¹P. Nikoueeyan is supported by a fellowship from the University of Wyoming's Engineering Initiative.

9:05AM A31.00006 Postage-Stamp PIV: Small Velocity Fields at 400 kHz for Turbulence Spectra Measurements, STEVEN BERESH, JOHN HENFLING, RUSSELL SPILLERS, Sandia National Laboratories — Time-resolved particle image velocimetry (TR-PIV) recently has been demonstrated in high-speed flows using a pulse-burst laser at repetition rates reaching 50 kHz. However, the turbulent behavior at all but the largest scales occurs at still higher frequencies. Pulse-burst PIV can be achieved at higher frequencies if the field of view is greatly reduced and lower laser pulse energy is accepted. Current technology allows image acquisition at 400 kHz for sequences exceeding 4,000 frames, but for a small array of only 128 × 120 pixels, giving rise to the moniker of postage-stamp PIV. Despite the limited spatial extent, this approach is well-suited to measuring turbulent velocity spectra in high-speed flows because it is not subject to frozen turbulence assumptions and it can employ advanced algorithms using the closely-spaced laser pulses and local spatial information. The resulting spectra reveal two regions exhibiting power-law dependence describing the turbulent decay. One is the well-known inertial subrange with a slope of -5/3 at high frequencies. The other displays a -1 power-law dependence for a decade of mid-range frequencies corresponding to the energetic eddies measured by PIV, which appears to have been previously unrecognized for high-speed free shear flows.

9:18AM A31.00007 Validation of Multi-plane Particle Shadow Velocimetry to Quantify Turbulence Scales, CHRISTINE TRUONG, STEVEN HINKLE, KYLE SINDING, JEFF HARRIS, MICHAEL KRANE, RHETT JEFFERIES, Pennsylvania State University — Estimates of radial integral length scales using multi-plane Particle Shadow Velocimetry (PSV) are presented using measurements from the 11.2-inch diameter glycerin tunnel in the Applied Research Lab Garfield Thomas Water Tunnel. 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. Thus, velocity fields in two imaging planes separated radially along the optical axis can be simultaneously measured. These multi-plane velocity fields are correlated over a range of separations to obtain integral length scales. Integral time scales are also calculated and converted into a streamwise length scale using Taylor's hypothesis for further confirmation. The inter-plane radial length scales, the in-plane length scales, the converted time scale in the inter-plane radial direction, and multi-plane turbulent statistics are compared with published studies, which used proven measurement methods. An additional, independent check is provided from PSV measurements in a single radial-axial plane.

9:31AM A31.00008 Application of Multi-Plane Particle Shadow Velocimetry to Obtain Velocity Fields Through an Optically Clear Object¹, STEVEN HINKLE, CHRISTINE TRUONG, KYLE SINDING, RHETT JEFFERIES, JEFF HARRIS, MICHAEL KRANE, Pennsylvania State Univ — Particle Shadow Velocimetry (PSV) is performed using an LED array to illuminate a volume of fluid rather than individual two-dimensional laser sheets as is done in Particle Image Velocimetry (PIV). Multi-plane PSV is a technique that is able to take advantage of the volumetric illumination of PSV to simultaneously take velocity field measurements in two different planes along the same optical axis within the fluid flow. This technique can be further extended to resolve flow fields around and through clear objects to obtain measurements along the optical axis both in front of and behind the object. A proof of concept application of taking images both in front of and behind cylindrical rods is presented. The rods, one made of clear acrylic and the other borosilicate glass, were chosen to have an index of refraction close to that of the surrounding fluid. Two different calibration targets are arranged on either side of the cylinder and simultaneous images are taken to show that velocity measurements in front of and behind the rod are possible to obtain utilizing multi-plane PSV. This methodology will be implemented in future measurements to obtain velocity fields for an airfoil on both the suction and pressure sides simultaneously in fully developed turbulent flow.

¹Penn State ARL

9:44AM A31.00009 Recovering mean flow quantities from limited time-resolved PIV measurements through a data assimilation framework, SEAN SYMON, California Institute of Technology, PETER SCHMID, Imperial College, DENIS SIPP, ONERA, BEVERLEY MCKEON, California Institute of Technology — Data assimilation combines experimental and numerical realizations of the same flow to produce hybrid flow fields. These have the advantages of less noise contamination and higher resolution while simultaneously reproducing the main physical features of the measured flow. This study investigates data assimilation of the mean flow around an idealized airfoil ($Re = 13500$) obtained from time-averaged PIV data. The experimental data, which represents a low-dimensional representation of the full flow field due to resolution and field of view limitations, is incorporated into a simulation governed by the incompressible RANS equations with an unknown momentum forcing. This forcing, which corresponds to the divergence of the Reynolds stress tensor, is calculated from a direct-adjoint optimization procedure to match the experimental and numerical mean velocity fields. The simulation is projected to the low-dimensional subspace of the experiment to calculate the discrepancy and a smoothing procedure is used to recover adjoint solutions on the higher-dimensional subspace of the simulation. The study quantifies how well data assimilation can reconstruct the mean flow and the minimum experimental measurements needed by altering the resolution and domain size of the time-averaged PIV.

Sunday, November 20, 2016 8:00AM - 9:44AM –

Session A32 Rough Wall Turbulent Boundary Layers - I Oregon Ballroom 201 - Mostafa Toloui,

University of Minnesota

8:00AM A32.00001 Comparison of turbulent flows over surfaces of rigid and flexible roughness, MOSTAFA TOLOUI, JIARONG HONG, University of Minnesota — The work aims at examining the influence of flexible surface roughness on wall-bounded turbulent flows. The experiments are conducted in a refractive-index-matched turbulent channel (using NaI solution) with a test section of 1.2 m in length and 50 mm square cross section. The rough samples consist of tapered cylinders of 0.35 mm in base diameter with 3 mm in height and 4 mm spacing in a 25 cm stretch. Two types of transparent polydimethylsiloxane (Sylgard 184 and Solaris) are used to generate roughness with about an order of magnitude difference in compliance (i.e. bulk elastic modulus of 1.8 MPa vs. 0.2 MPa). The dimension and the elastic modulus of roughness elements are designed such that the rough surface with higher modulus shows no deformation (namely rigid roughness) while the one with lower value deforms appreciably under the present flow conditions. Flow measurements are conducted using digital inline holographic PTV to obtain 3D velocity fields in a volume of 10 mm (streamwise) \times 50 mm (wall-normal) \times 10 mm (spanwise) with a temporal resolution of 0.16 ms and a spatial resolution of 1.1 mm/vector, above both rigid and deformable rough surfaces under two Reynolds numbers. A selection of instantaneous samples, mean velocity profile, turbulent fluctuation, and energy spectra as well as conditionally-sampled flow fields are analyzed to quantify the effect of roughness compliance on general turbulent statistics and coherent flow structures.

8:13AM A32.00002 Effects of spanwise topographic heterogeneity on amplitude and frequency modulation of streamwise velocity fluctuations.¹, ANKIT AWASTHI, UT Dallas, MATTHEW SUBDBERG, UT Dallas, WILLIAM ANDERSON, UT Dallas — We present results on the effects of topographic height, and spanwise heterogeneity, on amplitude and frequency modulation of small-scale structures in the roughness sublayer due to large-scale structures in the logarithmic region of turbulent channel flows. This work follows preceding contributions on amplitude and frequency modulation in smooth wall turbulent boundary layers (Mathis et al., J. Fluid Mech., **628**, 2009a, 311–337 and Ganapathisubramani et al., J. Fluid Mech., **712**, 2012, 61–91). We have considered three topographic cases with different characteristics from homogeneous (sandpaper), to two spanwise heterogeneous cases where the height amplitude is increased (this topographic configuration induces turbulent secondary flows, which are known to alter the outer-layer flow characteristics). Indeed, pre-multiplied energy spectra across wavelength and elevation (so-called spectrograms) illustrate how turbulent energy is redistributed with systematic modification to the underlying topography. We have determined how the large-scale (low-pass filtered) streamwise velocity modulates the amplitude and frequency response of small-scale (high-pass filtered) signal. We find that outer-layer topographic-induced perturbations completely alter the intensity of amplitude and frequency modulation. This highlights the passive-actuator-like role of roughness heterogeneity, and underpins the need to incorporate such functional dependence in the development of wall models for LES.

¹Air Force Office of Scientific Research, Grant FA9550-14-1-0101

8:26AM A32.00003 Effects of bio-inspired microscale roughness on macroscale flow structures, HUMBERTO BOCANEGRAS EVANS, Texas Tech University, ALI M. HAMED, University of Illinois at Urbana-Champaign, SERDAR GORUMLU, ALI DOOSTTALAB, BURAK AKSAK, Texas Tech University, LEONARDO P. CHAMORRO, University of Illinois at Urbana-Champaign, LUCIANO CASTILLO, Texas Tech University — The interaction between rough surfaces and flows is a complex physical situation that produces rich flow phenomena. While random roughness typically increases drag, properly engineered roughness patterns may produce positive results, e.g. dimples in a golf ball. Here we present a set of PIV measurements in an index matched facility of the effect of a bio-inspired surface that consists of an array of mushroom-shaped micro-pillars. The experiments are carried out—under fully wetted conditions—in a flow with adverse pressure gradient, triggering flow separation. The introduction of the micro-pillars dramatically decreases the size of the recirculation bubble; the area with backflow is reduced by approximately 60%. This suggests a positive impact on the form drag generated by the fluid. Furthermore, a negligible effect is seen on the turbulence production terms. The micro-pillars affect the flow by generating low and high pressure perturbations at the interface between the bulk and roughness layer, in a fashion comparable to that of synthetic jets. The passive approach, however, facilitates the implementation of this coating. As the mechanism does not rely on surface hydrophobicity, it is well suited for underwater applications and its functionality should not degrade over time.

8:39AM A32.00004 Turbulent boundary layer measurements over high-porosity surfaces, CHRISTOPH EFSTATHIOU, MITUL LUHAR, Univ of Southern California — Porous surfaces are ubiquitous across a variety of turbulent boundary layer flows of scientific and engineering interest. While turbulent flows over smooth and rough walls have been studied extensively, experimental measurements over porous walls have thus far focused on packed beds, which are limited in porosity ($\Phi = 0.3 - 0.5$) by their geometry. The current project seeks to address this limitation. A two-component laser doppler velocimeter (LDV) is used to generate velocity measurements in turbulent boundary layer flows over commercially available reticulated foams and 3D-printed porous media at Reynolds number $Re_\theta \approx 3000 - 4000$. Smooth wall profiles for mean and turbulent quantities are compared to data over substrates with porosity $\Phi > 0.8$ and average pore sizes in the range 0.4–2.5mm (corresponding to $\approx 8 - 50$ viscous units). Previous analytical and simulation efforts indicate that the effects of porous substrates on boundary layer flows depend on a modified Reynolds number defined using the length scale $\sqrt{\kappa}$, where κ is substrate permeability. A custom permeameter is currently being developed to estimate κ for the substrates tested in the boundary layer experiments.

8:52AM A32.00005 Turbulent flow past three-dimensional patches of roughness, MANUEL FERREIRA, BHARATHRAM GANAPATHISUBRAMANI, Univ of Southampton — Generally, investigations of flows over rough surfaces tend to focus on regular arrays of uniform obstacles (such as cubes or cylinders) or irregular distributions. This approach has led to significant progress in this field of research but so far has been unable to provide an accurate representation of flows past more complex topologies that are of a finite size. In this context, wind tunnel experiments are conducted of flows over three-dimensional patches of roughness. Randomly generated rough patches with large relative height ($h/\delta \approx 0.1$) are placed within a turbulent boundary layer. The characteristics of the finite patch of roughness are systematically varied by altering both frontal solidity (λ_F) and plan solidity (λ_P) over a large range ($\lambda_F \approx 0.05-0.25$ and $\lambda_P = 0.10-0.38$) from sparse to dense. Measurements are made using a floating-element force balance for all cases to study the behaviour of the drag with varying λ_F and λ_P . Additionally, high-resolution planar Particle Image Velocimetry (PIV) are carried out for selected cases in two different planes, streamwise wall-normal plane at the spanwise centerline of the patch as well as wall-parallel plane at $y/h \approx 3$.

9:05AM A32.00006 Roughness induced flow separation in adverse pressure gradient¹, JONGWOOK JOO, United Technologies Research Center , MIKE EMORY, SANJEEB BOSE, Cascade Technologies , GORAZD MEDIC, OM SHARMA, United Technologies Research Center — Surface roughness does not only increase turbulent mixing, but also thickens boundary-layers, making flows more susceptible to separation. Detailed flow physics related to the separation is not understood well. Bammert and Milsch (1972) demonstrates a clear example of surface roughness induced separation under adverse pressure gradient. In the study, compressor cascades with NACA 65 airfoils are systematically roughened and the flow over suction surface gradually separates early as roughness increases. A set of Large-Eddy Simulations (LES) over the Bammert's case is investigated, since RANS simulations using roughness models suffer from capturing the separation. In the current study, surface roughness is represented in two different approaches; 1) Realistic rough surface represented by stochastically distributed hills and valleys are gridded and solved with unstructured finite volume method, 2) Using block-structured grid, surface roughness is gridded as a staggered array of 3D rectangles, in a similar way of the previous study for roughened low pressure turbine (GT2016-57912). The current LES's capture rich features of the flow phenomena, which will bring comprehensive understanding of the roughness induced separation.

¹This collaboration is made through 2016 CTR Summer Program.

9:18AM A32.00007 ABSTRACT WITHDRAWN —

9:31AM A32.00008 Investigation of inner-outer interactions in a rough-wall turbulent boundary layer using time-resolved PIV¹, GOKUL PATHIKONDA, University of Illinois, Urbana-Champaign, KENNETH T. CHRISTENSEN, University of Notre Dame, Notre Dame — A turbulent boundary layer over hexagonally packed hemispherical roughness is investigated using time resolved PIV in a Refractive Index Matched (RIM) facility. Two cameras, with different fields of view and spatial resolutions, have been used to view the streamwise – wall normal plane. Matching the refractive index of the working fluid with that of the roughness model eliminates near-wall reflections of the light sheet, and enables measurements very close to the wall. The high-sampling rate ($dt \approx 3t^*$) and near-wall spatial resolution of first camera ($dx \times dy \approx 10^2 y^*$) capture small scales within the roughness sublayer. The large field of view ($X \times Y \approx 2\delta \times 1.2\delta$) of the second camera simultaneously captures the large scales away from the wall. The modulation influences of the outer large scales on the small scales within the roughness sublayer is explored, and compared with the literature investigating the same using hot-wire measurements. The rich spatio-temporal data available provides a new perspective of the inner-outer interactions, and provides a more direct way of observing the existing hypotheses (for e.g., Baars et al., 2015, Exp. Fluids, 56 : p188).

¹Air Force Office of Scientific Research

Sunday, November 20, 2016 8:00AM - 9:57AM —

Session A33 Turbulence: Coherent Structures and Taylors Hypothesis Oregon Ballroom
202 - Beverley McKeon, California Institute of Technology

8:00AM A33.00001 Roll/streak Structure Instability Induced by Free-stream Turbulence in Couette Flow, BRIAN FARRELL, Harvard University, PETROS IOANNOU, MARIOS NIKOLAIDIS, National and Kapodistrian University of Athens — Statistical state dynamics (SSD) provides a new perspective for studying mechanisms underlying turbulence in shear flow including instabilities which arise intrinsically from interaction between coherent and incoherent components of the turbulence. Implementations of SSD in the form of a closure at second order is used in this work to analyze the instability emergent from the statistical interaction between coherent perturbations of roll/streak form and the incoherent free-stream turbulence in a minimal channel configuration of Couette flow. By perturbing the nonlinear SSD dynamics a new manifold of stable modes with roll/streak structure is shown to exist in the presence of small amplitude free-stream turbulence. With increase in a parameter controlling the free-stream turbulence energy, a member of this set of stable roll/streak structures is destabilized at a bifurcation and the associated roll/streak eigenmode is found to equilibrate at finite amplitude. The bifurcation structure predicted by the SSD roll/streak instability is reflected in both a closely related quasi-linear dynamical system, referred to as the restricted non-linear (RNL) system, and in DNS. This correspondence is further verified using ensemble implementations of the RNL and DNS systems.

8:13AM A33.00002 Structure and mechanism of turbulence under dynamical restriction in plane Poiseuille flow, NAVID CONSTANTINO, Univ of California - San Diego, BRIAN FARRELL, Harvard University, PETROS IOANNOU, National and Kapodistrian University of Athens, JAVIER JIMENEZ, Universidad Politecnica de Madrid, ADRIAN LOZANO-DURAN, Stanford University, MARIOS-ANDREAS NIKOLAIDIS, National and Kapodistrian University of Athens — The perspective of Statistical State Dynamics (SSD) is used to investigate plane Poiseuille turbulence at moderately high Reynolds numbers ($Re_\tau \approx 940$). Simulations of a quasi-linear restricted nonlinear dynamics (RNL), which is an approximation to the full SSD, provide insight into the mechanism and structure of turbulent flow. RNL dynamics spontaneously limits the support of its turbulence to a small set of streamwise Fourier components giving rise to a natural minimal representation of its turbulence dynamics. Although greatly simplified, this RNL turbulence exhibits natural-looking structures and turbulent statistics. RNL turbulence at the Reynolds numbers studied is dominated by the roll/streak structure in the buffer layer and similar very-large-scale structure (VLSM) in the outer layer. Diagnostics of the structure, spectrum and energetics of RNL and DNS turbulence are used to demonstrate that the roll/streak dynamics supporting the turbulence in the buffer and logarithmic layer is essentially similar in RNL and DNS. This mechanism, which has analytical expression in the SSD, comprises a cooperative interaction between the coherent streamwise mean flow and the incoherent turbulent perturbations.

8:26AM A33.00003 On the structure of pressure fluctuations of self-sustaining attached eddies¹, MINJEONG CHO, HAECHON CHOI, Seoul National University, YONGYUN HWANG, Imperial College London — A numerical experiment, which isolates the energy-containing motions at a prescribed spanwise length scale, was recently performed by Hwang (2015, J. Fluid Mech., 767:254–289). In his study, the velocity structures of these motions were shown to emerge in the form of Townsends attached eddies. In the present study, pressure fluctuations of the self-sustaining attached eddies are analyzed for turbulent channel flow at $Re_T = 2000$. The second-order moment and the spectra of the pressure field of each attached eddy are found to be self-similar with respect to the given spanwise size, which is consistent with the velocity statistics. Depending on the nature of the source terms in the pressure Poisson equation, the pressure field of each attached eddy is also decomposed into the rapid and slow parts: the former describes linear interaction of the velocity fluctuation with mean shear, while the latter represents nonlinear interactions between the velocity fluctuations. In this talk, a detailed discussion will be provided with particular emphasis on the role of the rapid and slow parts of the pressure fluctuations in relation to its statistical and dynamical features.

¹Supported by 20152020105600

8:39AM A33.00004 Large-scale structures in turbulent Couette flow¹, JUNG HOON KIM, JAE HWA LEE, Ulsan Natl Inst of Sci & Tech — Direct numerical simulation of fully developed turbulent Couette flow is performed with a large computational domain in the streamwise and spanwise directions ($40\pi h$ and $6\pi l$) to investigate streamwise-scale growth mechanism of the streamwise velocity fluctuating structures in the core region, where h is the channel half height. It is shown that long streamwise-scale structures ($>3h$) are highly energetic and they contribute to more than 80% of the turbulent kinetic energy and Reynolds shear stress, compared to previous studies in canonical Poiseuille flows. Instantaneous and statistical analysis show that negative- u' structures on the bottom wall in the Couette flow continuously grow in the streamwise direction due to mean shear, and they penetrate to the opposite moving wall. The geometric center of the log layer is observed in the centerline with a dominant outer peak in streamwise spectrum, and the maximum streamwise extent for structure is found in the centerline, similar to previous observation in turbulent Poiseuille flows at high Reynolds number. Further inspection of time-evolving instantaneous fields clearly exhibits that adjacent long structures combine to form a longer structure in the centerline.

¹This research was supported by Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Education (NRF-2014R1A1A2057031).

8:52AM A33.00005 An alternative to Reynolds stresses in turbulent channels¹, JAVIER JIMENEZ, Universidad Politecnica Madrid — It is remarked that fluxes in conservation laws, such as the Reynolds stresses in the momentum equation of turbulent shear flows, or the spectral energy flux in isotropic turbulence, are only defined up to an arbitrary solenoidal field. While this is not usually significant for long-time averages, it becomes important when fluxes are modelled locally in large-eddy simulations, or in the analysis of intermittency and cascades. As an example, a numerical procedure is introduced to compute fluxes in scalar conservation equations in such a way that their total integrated magnitude is minimised. The result is an irrotational vector field that derives from a potential, thus minimising sterile flux ‘circuits’. The algorithm is generalised to tensor fluxes and applied to the transfer of momentum in a turbulent channel. The resulting instantaneous Reynolds stresses are compared with their traditional expressions, and found to be substantially different.

¹Funded by the Coturb project of the ERC

9:05AM A33.00006 Effects of Taylor-Grtler vortices on turbulent flows in a spanwise-rotating channel¹, YIJUN DAI, WEIXI HUANG, CHUNXIAO XU, Tsinghua Univ — Fully developed turbulent channel flow with spanwise rotation has been studied by direct numerical simulation at $Re_m = 2800$, 7000 and 20000 with rotation number $0 \leq Ro_m \leq 0.5$. The width of the computational box is adjusted for each case to contain two pairs of Taylor-Grtler (TG) vortices. Under a low rotation rate, the turbulent vortical structures are strongly affected by the TG vortices. A conditional average method is employed to investigate the effects. In the upwash region where the fluid is pumped away from the pressure wall by the TG vortices, turbulence is enhanced, while the reverse is the case in the downwash region. Through budget analysis of the transport equation of vorticity fluctuation, it is revealed that the stretching along the wall-normal direction caused by the TG vortices plays an important role in initiating the difference of turbulence intensity between the two regions, which is further augmented by the Coriolis force in the streamwise direction. The effects of TG vortices is weakened at higher Reynolds number. Meanwhile, the shear stress on the suction wall is observed to fluctuate in a quasi-periodic manner at $Re_m = 7000$ and $Ro_m = 0.3$, which is induced by the TG vortices.

¹The work is supported by National Natural Science Foundation of China (Project No. 11490551, 11472154, 11322221, 11132005)

9:18AM A33.00007 A theory for coupled uniform momentum zones and vortical fissures in turbulent wall flows¹, BRANDON MONTEMURO, JOE KLEWICKI, CHRIS WHITE, GREG CHINI, Univ of New Hampshire — Both field observations and laboratory experiments suggest that at high Reynolds numbers Re the outer region of incompressible turbulent wall flows self-organizes into uniform momentum zones (UMZs) separated by internal shear layers called ‘vortical fissures’ (VFs). In this investigation, a candidate flow configuration is identified that has the potential to generate a self-sustaining interaction between a single VF and adjacent UMZs. Large- Re asymptotic analysis is used to derive coupled, reduced sets of equations that elucidate the dominant physical processes operative in the different regions of the flow. The results indicate that large-scale, streamwise roll modes can act as a homogenizing agent that leads to the formation of the UMZs while simultaneously producing a concentrated region of spanwise vorticity that comprises the VF. The analysis also highlights possible feedback mechanisms between the VF and UMZs that may enable their self-sustenance.

¹Funding for this work from NSF-CBET Award 1437851 is gratefully acknowledged.

9:31AM A33.00008 A Scale-by-Scale Linear Analysis of Convective Velocities and Taylor’s Hypothesis in Turbulent Channel Flows, ISMAIL HAMEDUDDIN, DENNICE GAYME, The Johns Hopkins University — We examine convective velocities in turbulent channel flows using a linear, stochastically forced model approximation of the Navier-Stokes equations. The resulting system is analytically tractable and includes the terms that are typically associated with the breakdown of Taylor’s hypothesis. We show that this approach leads to convective velocity predictions that are consistent with DNS. We demonstrate that our observed differences between the local mean and convective velocities can be attributed to the dependence of the phase velocities on both the streamwise and spanwise wavelengths. The convective velocity in the viscous sublayer is roughly constant and distinct from the mean velocity. Previous work suggests that this viscous sublayer convective velocity arises due to (a) buffer layer rolls and (b) large-scale outer layer structures that influence the near-wall region. We show that there is also a series of structures, self-similar in the cross-stream plane, that modify the convective velocity in the sublayer. The streamwise extent of these structures scales as the square of the cross-stream dimensions, which is similar to previously proposed scalings of near-wall spectra based on DNS. This work is supported by a Johns Hopkins University Catalyst Award.

9:44AM A33.00009 Assessing the role of spanwise roughness heterogeneity and establishing conditions for application of Townsend's hypothesis of outer-layer similarity.¹

, JIANZHI YANG, WILLIAM ANDERSON, UT Dallas — In the present work, turbulent secondary flows in turbulent channel flows are forced by virtue of passive-actuator-like topographic features. The topographies are constructed with streamwise-repeating pyramidal topographic elements which create "rows" with infinite streamwise extent; then, the spacing between such rows is systematically varied. The spanwise spacing is varied from 0.2 to 6.4. For spanwise spacing exceeding double the channel half-height, free domain-scale secondary flows are observed. The domain-scale secondary flows exist in the Reynolds-averaged statistics, and remain permanently positioned such that downwelling and upwelling occurs above the elements and flat region, respectively (i.e., we have introduced low- and high-momentum pathways; Ken Christensen et al.). At the element scale, counter-rotating vortices flank the roughness elements, with rotational sign opposite to the larger-scale circulations. When spanwise spacing is less than double the channel half-height, the domain-scale secondary flows attenuate, while the element-scale secondary flows remain for all spacings. Since the domain-scale circulations alter the outer-layer turbulence statistics, we are working to assess spacing as a control on the efficacy of Townsend's hypothesis of outer-layer similarity in turbulent channel flow.

¹Air Force Office of Scientific Research, Grant FA9550-14-1-0394

Sunday, November 20, 2016 8:00AM - 9:44AM –

Session A34 Turbulence: Analysis of DNS Models Oregon Ballroom 203 - Charles Meneveau, Johns Hopkins University

8:00AM A34.00001 Analysis of Lagrangian stretching in turbulent channel flow using a database task-parallel particle tracking approach¹, CHARLES MENEVEAU, PERRY JOHNSON, STEPHEN HAMILTON, RANDAL BURNS, Johns Hopkins University — An intrinsic property of turbulent flows is the exponential deformation of fluid elements along Lagrangian paths. The production of enstrophy by vorticity stretching follows from a similar mechanism in the Lagrangian view, though the alignment statistics differ and viscosity prevents unbounded growth. In this paper, the stretching properties of fluid elements and vorticity along Lagrangian paths are studied in a channel flow at $Re_\tau = 1000$ and compared with prior, known results from isotropic turbulence. To track Lagrangian paths in a public database containing Direct Numerical Simulation (DNS) results, the task-parallel approach previously employed in the isotropic database is extended to the case of flow in a bounded domain. It is shown that above 100 viscous units from the wall, stretching statistics are equal to their isotropic values, in support of the local isotropy hypothesis. Normalized by dissipation rate, the stretching in the buffer layer and below is less efficient due to less favorable alignment statistics. The Cramér function characterizing cumulative Lagrangian stretching statistics shows that overall the channel flow has about half of the stretching per unit dissipation compared with isotropic turbulence.

¹Supported by a National Science Foundation Graduate Research Fellowship Program under Grant No. DGE-1232825, and by National Science Foundation grants CBET-1507469, ACI-1261715, OCI-1244820 and by JHU IDIES.

8:13AM A34.00002 ABSTRACT WITHDRAWN –

8:26AM A34.00003 Small-scale anisotropy in turbulent boundary layers, ALAIN PUMIR, Ecole Normale Supérieure de Lyon, Lyon, France, HAITAO XU, Tsinghua University, Beijing, China, ERIC SIGGIA, Rockefeller University, New York City, NY. — In a channel flow, the velocity fluctuations are inhomogeneous and anisotropic. Yet, the small-scale properties of the flow are expected to behave in an isotropic manner in the very large Reynolds number limit. We consider the statistical properties of small-scale velocity fluctuations in a turbulent channel flow at moderately high Reynolds number ($Re_\tau \approx 1000$), using the Johns Hopkins University turbulence database. Away from the wall ($y^+ > 200$), the skewness of the normal derivative of the streamwise velocity fluctuation is approximately constant, of order 1, while the Reynolds number based on the Taylor scale is $R_\lambda \approx 150$. This defines a small-scale anisotropy that is stronger than in turbulent homogeneous shear flows at comparable values of R_λ . In contrast the vorticity-strain correlations that characterize homogeneous isotropic turbulence are nearly unchanged in channel flow even though they vary with distance from the wall with an exponent that can be inferred from the local dissipation. The statistical properties of the fluctuating velocity gradient in turbulent channel flow are therefore characterized, on one hand, by observables which are insensitive to the anisotropy, and behave as in homogeneous isotropic flows, and on the other hand by quantities which are sensitive to the anisotropy.

8:39AM A34.00004 Entropy Production and Fluctuation Relation in Turbulent Convection, SERGIO CHIBBARO, Institut d'Alembert Paris 6, FRANCESCO ZONTA, Univ. Udine — We report on a numerical experiment performed to analyze fluctuations of the entropy production in turbulent thermal convection. Using Direct Numerical Simulations (DNS), we estimate the entropy production from instantaneous measurements of the local temperature and velocity fields sampled along the trajectory of a large number of point-wise Lagrangian tracers. Entropy production is related to the work made by buoyancy force. The entropy production is characterized by large fluctuations and becomes often negative. This represents a sort of "finite-time" violation of the second principle of thermodynamics, since the direction of the energy flux is opposite to that prescribed by the external gradient. We provide a physical-sound definition of energy-scale characterizing the system, based upon Kolmogorov theory. Then, we link our results with recent theory of statistical mechanics of nonequilibrium systems, notably the results obtained by Evans, Cohen, Morris and Gallavotti for generic reversible dynamical systems. We show that the fluctuations of entropy production observed in the present system verify neatly the Fluctuation Relation (FR), cornerstone of that theory, even though the system is time-irreversible.

8:52AM A34.00005 On the establishment of fully developed turbulence in direct numerical simulations driven by stochastic forcing, SUALEH KHURSHID, DIEGO DONZIS, Texas AM University, KATEPALLI SREENIVASAN, New York University — Turbulent statistics are commonly described within the classical Richardson-Kolmogorov paradigm which aims at characterizing flows at very high Reynolds numbers. Recent work, however, has shown that certain aspects observed and predicted by high-Reynolds-number theories are present at very low Reynolds numbers. This motivates our investigation of the emergence of turbulent behavior using highly resolved direct numerical simulations (DNS) on a periodic box starting from zero initial conditions and driven by stochastic forcing at the largest scales. In particular, we focus on the evolution of different wavenumber bands in time to study short-time behavior. Detailed analysis of our DNS data shows limitations of classical approaches in explaining the development of energy cascade. We show how energy is transferred to smaller scales and the rate at which this transfer proceeds. A description of the emergence of small scale intermittency and moments of velocity gradients is also presented.

9:05AM A34.00006 The importance of 3D local averaging in turbulence theory: some examples from high-resolution DNS¹, PUI-KUEN YEUNG, X.M ZHAI, Georgia Tech, K.P. IYER, Univ of Rome Tor Vergata and INFN, Italy, K.R. SREENIVASAN, New York Univ — Dissipation fluctuations in turbulence become increasingly intermittent as the Reynolds number increases. Both theoretical and practical reasons then force us to consider the fluctuations averaged locally over three-dimensional (3D) volumes of various sizes. Often, the practice has been to supplant 3D averages by 1D averages, and to replace proper 3D quantities by convenient 1D surrogates. We examine the consequence of these practices using DNS data on a large grid of 8192³ at a Taylor-microscale Reynolds number 1300. We show that these common practices can often lead to erroneous results and significant ambiguities. For instance, both the dissipation and enstrophy turn out to possess the same inertial-range intermittency exponent; moments of locally-averaged dissipation and enstrophy become closer to each other with increasing order (because extreme events in both are spatially co-located); the longitudinal and transverse velocity increments scale similarly—all in contrast to results obtained using the simplifying practices mentioned above.

¹Supported by NSF Grants ACI-1036170 and ACI-1640771

9:18AM A34.00007 Near-wall Behavior of a Scale Self-Recognition Mixed SGS Model, MIZUKI KIHARA, YUKI MINAMOTO, Tokyo Institute of Technology, YOSHITSUGU NAKA, Meiji University, NAOYA FUKUSHIMA, Tokyo University of Science, MASAYASU SHIMURA, MAMORU TANAHASHI, Tokyo Institute of Technology — A Scale Self-Recognition Mixed SGS Model was developed in terms of GS-SGS energy transfer in homogenous isotropic turbulence by Fukushima et al. (2015). In the present research, the near-wall characteristics of the Smagorinsky coefficient, C_S are investigated in terms of GS-SGS energy transfer by analyzing DNS data of turbulent channel flows at $Re_\tau = 400, 800$ and 1270 . C_S is dependent on grid anisotropy, and this cause dependences of C_S on Re_τ . It is revealed that C_S obtained directly from the DNS data is independent of Re_τ and dependent on only dimensionless wall distance, y^+ and filter-width to Kolmogorov scale ratio corrected by f , $f \cdot \Delta/\eta$, when the grid anisotropy is isolated from C_S by using the correction function f proposed by Scotti et al. (1993). The contributions of Leonard, cross and Reynolds terms to total energy transfer are also independent of Re_τ and dependent on only y^+ and $f \cdot \Delta/\eta$ in the near-wall region. These results suggest that C_S can be determined dynamically from $f \cdot \Delta/\eta$ in the wall turbulence if η is sufficiently predicted from the grid scale quantities.

9:31AM A34.00008 Extreme accelerations in turbulent flows, JOHN LAWSON, CRISTIAN LAESCU, MICHAEL WILCZEK, EBERHARD BODENSCHATZ, Max Planck Institute for Dynamics and Self-Organisation — Even in weakly turbulent flows, fluid tracers routinely undergo strong accelerations tens of standard deviations in excess of the mean. These extreme events are thought to influence everyday phenomena such as rain formation in wet clouds or the turbulent combustion of fuel in engines. We report results on high resolution particle tracking experiments in a vigorously stirred turbulent flow between $R_\lambda = 130$ and 450 . These are matched with high resolution direct numerical simulations of isotropic turbulence. By acquiring very large datasets we quantify the distribution of rare, strong acceleration events (as infrequent as one in 10^8 and in excess of 30 standard deviations) and their scaling with Reynolds number. We present back-to-back comparisons between the two to quantify the statistics of accelerations at unprecedented accuracy and discuss their consequences for turbulent flows.

Sunday, November 20, 2016 8:00AM - 9:44AM –

Session A35 Turbulence: General | Oregon Ballroom 204 - Edward Smith, Imperial College London

8:00AM A35.00001 A Molecular Dynamics Simulation of the Turbulent Couette Minimal Flow Unit¹, EDWARD SMITH, Imperial College London — What happens to turbulent motions below the Kolmogorov length scale? In order to explore this question, a 300 million molecule Molecular Dynamics (MD) simulation is presented for the minimal Couette channel in which turbulence can be sustained. The regeneration cycle and turbulent statistics show excellent agreement to continuum based computational fluid dynamics (CFD) at $Re=400$. As MD requires only Newton's laws and a form of inter-molecular potential, it captures a much greater range of phenomena without requiring the assumptions of Newton's law of viscosity, thermodynamic equilibrium, fluid isotropy or the limitation of grid resolution. The fundamental nature of MD means it is uniquely placed to explore the nature of turbulent transport. A number of unique insights from MD are presented, including energy budgets, sub-grid turbulent energy spectra, probability density functions, Lagrangian statistics and fluid wall interactions.

¹EPSRC Post Doctoral Prize Fellowship

8:13AM A35.00002 Response of a turbulent von Karman swirling flow to anisotropy at the molecular scale, TIM GRUENBERG, THOMAS ROESGEN, ETH Zurich — We ask if and how the large-scale structure of a turbulent von Karman swirling flow depends on anisotropies introduced at the smallest scales. We generate such anisotropy at the viscous scale in a paramagnetic colloid whose rheology is modified by an external, uniform magnetic field. We report measurements in a high Reynolds number turbulence experiment ($R_\lambda = 120$). Ultrasound velocimetry provides records of tracer particle velocity. Distinct changes in the velocity statistics can be observed extending from the dissipative scales up to the mean flow topology.

8:26AM A35.00003 Fundamental Entrainment Observations (VSL, etc.) for a SSSL, JOHN FOSS, Michigan State University, KYLE BADE, Spraying Systems Co., DOUGLAS NEAL, RICHARD PREVOST, LaVision Inc. — Fundamental observations of the entrainment process on the low speed side of a high Re self-preserving single stream shear layer have been made using PIV realizations. The Re value was: $U_0\theta_{mid}/\nu = 6.75 \times 10^4$, where $\theta_{mid} = 13.7$ cm is the momentum thickness at the mid-location ($x/\theta(0) = 390$) of the observations. The VSL (Viscous Super Layer), 15-20 η_K thick, is bounded by a well-defined border where the non-vortical/vortical transition occurs. The Kolmogorov microscale (η_K) was determined from the mean-square vorticity adjacent to the VSL. A threshold level to define the border ($\omega_z\theta_{mid}/U_0 = 0.221$) was selected by examination of the data. Quantitative measures of the entrainment process have been obtained, including: *i*) the convoluted length of the border (L_b) made non-dimensional with respect to the length (L_m) of the temporally averaged flow field ($L_b/L_m = 2.8$) and *ii*) $\langle v_b^2 \rangle / v_e^2 = 17$, as a measure of the sink-effect at the border. v_b is the measured velocity at the border; v_e is the well-established entrainment velocity far from the active shear layer whose value: $v_e/U_0 = 0.035$, corresponds to the growth of the self-preserving SSSL ($d\theta/dx$).

8:39AM A35.00004 Modeling the stochastic dynamics of moving turbulent spots over a slender cone at Mach 5 during laminar-turbulent transition¹, BRIAN ROBBINS, RICH FIELD, Sandia National Laboratories, MIRCEA GRIGORIU, Cornell University, RYAN JAMISON, MIKHAIL MESH, KATYA CASPER, LAWRENCE DECHANT, Sandia National Laboratories — During reentry, a hypersonic vehicle undergoes a period in which the flow about the vehicle transitions from laminar to turbulent flow. During this transitional phase, the flow is characterized by intermittent formations of localized turbulent behavior. These localized regions of turbulence are born at the onset of transition and grow as they move to the aft end of the flight vehicle. Throughout laminar-turbulent transition, the moving turbulent spots cause pressure fluctuations on the outer surface of the vehicle, which leads to the random vibration of the structure and its internal components. In light of this, it is of great interest to study the dynamical response of a flight vehicle undergoing transitional flow so that aircraft can be better designed to prevent structural failure. In this talk, we present a statistical model that calculates the birth, evolution, and pressure field of turbulent spots over a generic slender cone structure. We then illustrate that the model appropriately quantifies intermittency behavior and pressure loading by comparing the intermittency and root-mean-square pressure fluctuations produced by the model with theory and experiment. Finally, we present results pertaining to the structural response of a housing panel on the slender cone.

¹Sandia is a multiprogram laboratory operated by Sandia Corporation, a Lockheed Martin Company, for the United States Department of Energy's National Nuclear Security Administration under Contract DE-AC04-94AL85000.

8:52AM A35.00005 Investigation of RANS Model Deficiencies for Flow and Heat Transfer Simulations in a Pin-Fin Array¹, ZENGRONG HAO, CATHERINE GORLE, Stanford University, CIVIL AND ENVIRONMENTAL ENGINEERING DEPARTMENT TEAM — Quantifying turbulence model-form uncertainties of Reynolds-averaged Navier-Stokes (RANS) models is a challenging topic, largely because model inaccuracies can vary greatly across flow regions. The objective of the present study is to gain knowledge on where and how RANS models violate reality in representative heat exchanger geometries, such that a UQ method for use in optimization studies can be developed. To achieve this objective we performed a large-eddy simulation (LES) of the flow and heat transfer for a pin-fin array, and analyzed where RANS fails to predict the key features. The LES results are validated against experimental data available from Ames et al. (J. Turbo., 2005) and Ames and Dvorak (J. Turbo., 2006). The RANS simulations showed significant deviations from the LES for mean velocity profiles downstream of certain pins and for the Nusselt number distribution on the fins. A detailed comparison of the turbulent quantities illustrates a general underestimation of the Reynolds stresses and turbulent heat fluxes in near-fin regions, and incorrect trends in some pin-wake regions near the channel center plane. Based on this analysis, we draw conclusions that will support the development of a turbulence model UQ method for heat exchangers.

¹This work is supported by IWT SBO project EUFORIA.

9:05AM A35.00006 ABSTRACT WITHDRAWN —

9:18AM A35.00007 Analysis of non-stationary turbulent flows using Multivariate EMD and Matching Pursuits, ARVIND MOHAN, LIONEL AGOSTINI, DATTA GAITONDE, The Ohio State University, MIGUEL VISBAL, U.S. Air Force Research Laboratory — Time-series analysis of highly transient non-stationary turbulent flow is challenging. Traditional Fourier based techniques are generally difficult to apply because of the highly aperiodic nature of the data. Another significant obstacle is assimilating multivariate data, such as multiple variables at a location or from different sources in a flow-field. Such an analysis has the potential to identify sensitive events common among these sources. In this work, we explore two techniques to address these challenges - Multivariate Empirical Mode Decomposition and Matching Pursuits, on deep dynamic stall of a plunging airfoil in a mixed laminar-transitional-turbulent regime. Although primarily used for neuroscience applications, we use them in fluid mechanics and highlight their significant potential to overcome limitations of more traditional techniques. Application of these methods highlight different stages in the development of stall. A first stage shows development of 2-D boundary layer oscillations at frequencies similar to those associated with trailing edge vortices. Subsequently, new instabilities arise due to imminent separation. The separation bubble itself is characterized by relatively higher frequency content, and further analysis indicates its 3-D collapse.

9:31AM A35.00008 Investigation of coherent structures in a superheated jet using decomposition methods¹, AVICK SINHA, SHIVASUBRAMANIAN GOPALAKRISHNAN, SRIDHAR BALASUBRAMANIAN, Indian Institute of Technology Bombay — A superheated turbulent jet, commonly encountered in many engineering flows, is complex two phase mixture of liquid and vapor. The superposition of temporally and spatially evolving coherent vortical motions, known as coherent structures(CS), govern the dynamics of such a jet. Both POD and DMD are employed to analyze such vortical motions. PIV data is used in conjunction with the decomposition methods to analyze the CS in the flow. The experiments were conducted using water emanating into a tank containing homogeneous fluid at ambient condition. Three inlet pressure were employed in the study, all at a fixed inlet temperature. 90% of the total kinetic energy in the mean flow is contained within the first five modes. The scatterplot for any two POD coefficients predominantly showed a circular distribution, representing a strong connection between the two modes. We speculate that the velocity and vorticity contours of spatial POD basis functions show presence of K-H instability in the flow. From DMD, eigenvalues away from the origin is observed for all the cases indicating the presence of a non-oscillatory structure. Spatial structures are also obtained from DMD.

¹The authors are grateful to Confederation of Indian Industry and General Electric India Pvt. Ltd. for partial funding of this project

Sunday, November 20, 2016 10:27AM - 12:32PM — Session B40 Presentation of Awards and DFD Fellowships (Otto LaPorte Lecture and Stanley Corrsin Award) Portland Ballroom - Kenneth Breuer, Karen Flack, Ellen Longmire, Brown University, US Naval Academy, University of Minnesota,

10:27AM B40.00001 AWARDS AND FELLOWSHIP PRESENTATION —

11:02AM B40.00002 Otto Laporte Award Lecture: Seeking simplicity in the flows of complex fluids, HOWARD STONE, Princeton University — Fluid mechanics is a discipline with rich phenomena, spanning a wide range of laminar and turbulent flows, instabilities, and applications in industry, nature, and biology and medicine. I will provide examples of our work highlighting (i) new features of classical instabilities triggered by changes in geometry, (ii) multiphase flows relevant to the design of liquid-infused substrates exhibiting effective slip, and, if there is a time, (iii) unexpected dynamics in flow at a T-junction.

11:47AM B40.00003 Stanley Corrsin Award Lecture: High Reynolds Number Wall

Turbulence , IVAN MARUSIC, University of Melbourne — A key consideration in the characterization of the mechanics of turbulent flows is to understand the generation, evolution and interactions of the large-scale structures and the range of eddying motions that make up the turbulent flow. The non-linearity of these processes makes the problem challenging, both computationally and experimentally. This is particularly true in wall-bounded flows where an increasing hierarchy of energy-containing eddy scales exists with increasing Reynolds number. In this talk we will review recent studies in high Reynolds number flow facilities and from the atmospheric surface layer documenting unique high Reynolds number phenomena in wall turbulence. The focus will be the logarithmic region, looking at issues regarding its universality, coherent structures and how they interact across the boundary layer. These findings lead to a new consideration of so-called "inner-outer" interactions and form the basis of a new predictive model for the near-wall inner region and the wall-shear stress. The implications of this model will be discussed.

Sunday, November 20, 2016 12:42PM - 1:17PM –

Session C32 Jumping on Water Oregon Ballroom 201-202-203 - Anette Hosoi, Massachusetts Institute of Technology

12:42PM C32.00001 Jumping on water , HO-YOUNG KIM, Department of Mechanical and Aerospace Engineering,

Seoul National University — Water striders can jump on water as high as they can jump on land. Quick jumps allow them to avoid sudden dangers such as predators attacks, and therefore understanding how they make such a dramatic motion for survival can shed light on the ultimate level of semi-aquatic motility achievable through evolution. However, the mechanism of their vertical jumping from a water surface has eluded hydrodynamic explanations so far. By observing movements of water strider legs and theoretically analyzing their dynamic interactions with deforming liquid-air interface, we have recently found that different species of jumping striders always tune their leg rotation speed with a force just below that required to break the water surface to reach the maximum take-off velocity (J.-S. Koh et al. Jumping on water: Surface tension-dominated jumping of water striders and robotic insects, Science, vol. 349, pp. 517-521, 2015). Here, we start with discussing the fundamental theories of dynamics of floating and sinking of small objects. The theories then enable us to analyze forces acting on a water strider while it presses down the water surface to fully exploit the capillary force. We further introduce a 68-milligram at-scale robotic insect capable of jumping on water without splash, strikingly similar to the real strider, by utilizing the water surface just as a trampoline.

Sunday, November 20, 2016 12:42PM - 1:17PM –

Session C40 On the Frontier: Some Observations of Surface Tension Phenomena from the International Space Station 251-258 - Ronald Adrian, Arizona State University

12:42PM C40.00001 On the Frontier: Some Observations of Surface Tension Phenomena from the International Space Station , DON PETTIT, NASA - Johnson Space Center — Frontiers are interesting places; they offer the possibility to make observations outside our normal range of experience. The International Space Station is such a frontier offering a local reduction in acceleration force by nearly a factor of a million thus allowing the observation of subtle phenomena that are typically masked on Earth. In this presentation we show a variety of unusually large length scale fluid interfacial phenomena dominated by electro-hydrodynamics, Marangoni convection, and non-Newtonian fluid mechanics. The experimental results presented are largely performed during weekends and other brief time-off periods allotted to the astronaut crew on the ISS.

Sunday, November 20, 2016 2:57PM - 4:41PM –

Session D1 Mini-Symposium: Geophysical Turbulence Induced by Flow over Topography II A105 - Jody Klymak, University of Victoria

2:57PM D1.00001 Turbulence during the generation of internal tides at obstacles ,

SUTANU SARKAR, University of California at San Diego — Three-dimensional simulations are performed to resolve turbulent processes in tidally-forced flow over different types of two-dimensional obstacles. Our objective is to explore the dependence of the spatial and temporal distribution of turbulence on the obstacle geometry. Fine-scale variability in the flow is associated with features such as critical slope boundary layers, upslope thermal bores, breaking lee waves, downslope jets, internal wave beams and wave-wave interactions. Turbulence in these flow features is maintained through both convective and shear instabilities, and the mixing efficiency depends on the type of instability that is operative. The spatio-temporal pattern of turbulent kinetic energy and dissipation depends strongly on local slope criticality, the overall aspect ratio of the obstacle, and the appropriately defined excursion number. Implications of the simulation results for simple parameterizations of the turbulent dissipation and mixing are discussed. The work presented here was done with Masoud Jalali.

3:23PM D1.00002 The Role of Non-Propagating Form Drag in Mixing the Ocean ,

KURT POLZIN, Woods Hole Oceanographic Institution — Topographically induced internal waves are common features in the atmosphere and ocean. In the former, they give rise to the so-called "mountain-waves" which form as winds blow over rough topography and give rise to severe downslope windstorms. In the ocean, boundary forced waves are related to flow of tides, inertial oscillations and geostrophic flows over rough topography. Depending upon length scale, stratification, latitude and frequency, the hydrodynamic response can be bottom trapped or freely upward propagating. Robust quantitative constraints on ocean mixing are few. They come from a restricted number of control volume budgets for abyssal upwelling and observations of diapycnal dispersion from anthropogenic tracer releases. This talk will summarize control volume budgets and tracer observations from the South Scotia Sea, the Brazil Basin, the Gulf of Mexico and the Western North Atlantic and discuss extant observations of fine- and microstructure that relate to the mechanisms of mixing. There are significant disparities between the fine- and microstructure observations and large scale budgets in all but the Brazil Basin. There, mixing is regarded as being associated with internal tide generation and near boundary breaking. Elsewhere, sub-inertial flow over finite amplitude topography is inferred to be the significant process. Rather than discussing the efficacy of quasi-stationary lee waves in mixing, we discuss closing the large scale budgets in the oceanic allegory of downslope windstorms: flow over finite amplitude topography giving rise to hydraulic like effects that can be summarized as converting non-propagating form drag into mixing.

3:49PM D1.00003 Internal tide convergence and mixing in a submarine canyon , AMY

WATERHOUSE, University of California, San Diego — Observations from Eel Canyon, located on the north coast of California, show that elevated turbulence in the full water column arises from the convergence of remotely-generated internal wave energy. The incoming semidiurnal and bottom-trapped diurnal internal tides generate complex interference patterns. The semidiurnal internal tide sets up a partly standing wave within the canyon due to reflection at the canyon head, dissipating all of its energy within the canyon. Dissipation in the near-bottom is associated with the diurnal trapped tide, while midwater isopycnal shear and strain is associated with the semidiurnal tide. Dissipation is elevated up to 600 m off the bottom, in contrast to observations over flat continental shelf where dissipation occurs closer to the topography. Slope canyons are sinks for internal wave energy and may have important influences on the global distribution of tidally-driven mixing.

4:15PM D1.00004 Transition from interior to boundary turbulence: implications for the buoyancy flux divergence. , ALI MASHAYEK, MIT — I will discuss results from a cascade of nested simulations of flow over rough topography in the Southern Ocean with focus on transition from turbulence induced from breaking of upward propagating internal waves in the interior and above rough topography to that induced by boundary trapped modes in vicinity of the ocean floor. I will show that this transition leads to a change in sign of the buoyancy flux divergence, which implies a change from interior downwelling to intense upwelling in the boundary layer. I will show that in the Drake Passage region, the latter dominates the former leading to a net upwelling. I will finish by highlighting the potentially important implications of the results for the abyssal branch of the global ocean meridional overturning circulation.

Sunday, November 20, 2016 2:57PM - 5:07PM –

Session D2 Wind Turbines: Simulations A106 - Dennice Gayme, Johns Hopkins University

2:57PM D2.00001 A simple dynamic wake model for time dependent wind turbine yaw¹ , CARL SHAPIRO, CHARLES MENEVEAU, DENNICE GAYME, Johns Hopkins University

— This work develops a time dependent wake model for wind farms that better captures the spanwise and streamwise propagation of fluctuations generated by changes in turbine thrust and yaw angle. The model builds on classic wake models by incorporating time dependence and turbine yawing. These extensions enable us to capture the spanwise skewness in the yawed turbine wake as well as the dynamic advection of the wake downstream. This model is then compared to large eddy simulations of a wind farm with upstream rows of wind turbines dynamically yawing their rotors. An important advantage of the model is it allows us to take advantage of predictions of dynamic flow phenomena to coordinate the action of individual wind turbines for farm level control. We use the model to further explore the potential of wind farms to use wind turbine yaw to provide important services to the power grid through power tracking.

¹This work is supported by NSF (SEP-1230788 and OISE-1243482, the WINDINSPIRE project).

3:10PM D2.00002 An LES study of vertical-axis wind turbine wakes aerodynamics , MAHDİ ABKAR, Center for Turbulence Research, Stanford University, Stanford, California 94305, USA, JOHN O. DABIRI, Dept. of Civil and Environmental Eng. & Dept. of Mechanical Eng., Stanford University, Stanford, CA 94305, USA

— In this study, large-eddy simulation (LES) combined with a turbine model is used to investigate the structure of the wake behind a vertical-axis wind turbine (VAWT). In the simulations, a recently developed minimum dissipation model is used to parameterize the subgrid-scale stress tensor, while the turbine-induced forces are modeled with an actuator-line technique. The LES framework is first tested in the simulation of the wake behind a model straight-bladed VAWT placed in the water channel, and then used to study the wake structure downwind of a full-scale VAWT sited in the atmospheric boundary layer. In particular, the self-similarity of the wake is examined, and it is found that the wake velocity deficit is well characterized by a two-dimensional elliptical Gaussian distribution. By assuming a self-similar Gaussian distribution of the velocity deficit, and applying mass and momentum conservation, an analytical model is developed and tested to predict the maximum velocity deficit downwind of the turbine.

3:23PM D2.00003 An actuator line model simulation with optimal body force projection length scales¹ , LUIS MARTINEZ-TOSSAS, Johns Hopkins Univ, MATTHEW J. CHURCHFIELD, National Renewable Energy Laboratory, CHARLES MENEVEAU, Johns Hopkins Univ — In recent work (Martínez-Tossas et al. “Optimal smoothing length scale for actuator line models of wind turbine blades”, preprint), an optimal body force projection length-scale for an actuator line model has been obtained. This optimization is based on 2-D aerodynamics and is done by comparing an analytical solution of inviscid linearized flow over a Gaussian body force to the potential flow solution of flow over a Joukowski airfoil. The optimization provides a non-dimensional optimal scale ϵ/c for different Joukowski airfoils, where ϵ is the width of the Gaussian kernel and c is the chord. A Gaussian kernel with different widths in the chord and thickness directions can further reduce the error. The 2-D theory developed is extended by simulating a full scale rotor using the optimal body force projection length scales. Using these values, the tip losses are captured by the LES and thus, no additional explicit tip-loss correction is needed for the actuator line model. The simulation with the optimal values provides excellent agreement with Blade Element Momentum Theory.

¹This research is supported by the National Science Foundation (grant OISE-1243482, the WINDINSPIRE project).

3:36PM D2.00004 Assessing the Impacts of Low Level Jets’ Negative Wind Shear over Wind Turbines.¹ , WALTER GUTIERREZ, Department of Mechanical Engineering, Texas Tech University, Lubbock, Texas 79409, USA, ARQUIMEDES RUIZ-COLUMBIE, National Wind Institute, Texas Tech University, Lubbock, Texas 79409, USA, MURAT TUTKUN, Institute for Energy Technology (IFE), Kjeller, Norway, LUCIANO CASTILLO, Department of Mechanical Engineering, Texas Tech University, Lubbock, Texas 79409, USA

— Nocturnal Low Level Jets (LLJs) are defined as relative maxima in the vertical profile of the horizontal wind speed at the top of the stable boundary layer. Such peaks constitute major power resources, since they are observed at altitudes within the heights of commercial-size wind turbines. However, a wind speed maximum implies a transition from a positive wind shear below the maximum height to a negative one above. The effect that such transition inflicts on wind turbines has not been thoroughly studied. Here we focused on the impacts that the LLJ negative wind shears have over commercial size wind turbines. Using actual atmospheric LLJ data of high frequency as input for the NREL aeroelastic simulator FAST, different scenarios were created varying the LLJ maximum height with respect to the wind turbine hub height. We found only slight changes in the deflection and load averages for those scenarios, whereas the corresponding variances appear to decrease when a larger portion of the wind turbine sweeping area is affected by the negative shear. The exception was observed in the junction between the tower top and the nacelle, where a deflection maximum was detected that might reveal a critical structural point.

¹The authors gratefully acknowledge the following grants for this research: NSFCBET 1157246, NSFCMMI 1100948, NSFOISE1243482.

3:49PM D2.00005 Modeling velocity space-time correlations in wind farms¹ , LAURA J.

LUKASSEN, Max Planck Institute for Dynamics and Self-Organization, RICHARD J.A.M. STEVENS, University of Twente, CHARLES MENEVEAU, Johns Hopkins University, MICHAEL WILCZEK, Max Planck Institute for Dynamics and Self-Organization — Turbulent fluctuations of wind velocities cause power-output fluctuations in wind farms. The statistics of velocity fluctuations can be described by velocity space-time correlations in the atmospheric boundary layer. In this context, it is important to derive simple physics-based models. The so-called Tennekes-Kraichnan random sweeping hypothesis states that small-scale velocity fluctuations are passively advected by large-scale velocity perturbations in a random fashion. In the present work, this hypothesis is used with an additional mean wind velocity to derive a model for the spatial and temporal decorrelation of velocities in wind farms. It turns out that in the framework of this model, space-time correlations are a convolution of the spatial correlation function with a temporal decorrelation kernel. In this presentation, first results on the comparison to large eddy simulations will be presented and the potential of the approach to characterize power output fluctuations of wind farms will be discussed.

¹ Acknowledgements: ‘Fellowships for Young Energy Scientists’ (YES!) of FOM, the US National Science Foundation grant IIA 1243482, and support by the Max Planck Society.

4:02PM D2.00006 Using Reconstructed POD Modes as Turbulent Inflow for LES Wind Turbine Simulations , JORDAN NIELSON, KIRAN BHAGANAGAR, University of Texas San Antonio, VEJAPONG JUTTIJUDATA, Kaetsart University, SIROD SIRISUP, None —

Currently, in order to get realistic atmospheric effects of turbulence, wind turbine LES simulations require computationally expensive precursor simulations. At times, the precursor simulation is more computationally expensive than the wind turbine simulation. The precursor simulations are important because they capture turbulence in the atmosphere and as stated above, turbulence impacts the power production estimation. On the other hand, POD analysis has been shown to be capable of capturing turbulent structures. The current study was performed to determine the plausibility of using lower dimension models from POD analysis of LES simulations as turbulent inflow to wind turbine LES simulations. The study will aid the wind energy community by lowering the computational cost of full scale wind turbine LES simulations, while maintaining a high level of turbulent information and being able to quickly apply the turbulent inflow to multi turbine wind farms. This will be done by comparing a pure LES precursor wind turbine simulation with simulations that use reduced POD mod inflow conditions. The study shows the feasibility of using lower dimension models as turbulent inflow of LES wind turbine simulations. Overall the power production estimation and velocity field of the wind turbine wake are well captured with small errors.

4:15PM D2.00007 Proper orthogonal decomposition of a large eddy simulation during a diurnal cycle for very large wind farms , NASEEM ALI, Portland State University, GERARD CORTINA, University of Utah, NICHOLAS HAMILTON, Portland State University, MARC CALAF, University of Utah, RAL CAL, Portland State University

— The structure of the turbulent flow within large wind farms under different atmospheric flow stratification (stable, unstable and neutral) and compares it to the case when there are no turbines present. Spectral analysis is further applied to the corresponding proper orthogonal modes to identify the characteristic wavenumber and be able to relate it to the actual wind farm structure and wake-to-wake interactions. The variation in the number of needed modes between the different cases decreases with increasing value of cumulative energy, which confirms that the major difference between the different study cases resides at the largest turbulent kinetic energy containing scales of each case. The POD modes show the stratification impact of the flow structure and distinguish the flow layers. The spectral analysis displays the domain size and the distance between the rotors as distinctive scales within the wind farm.

4:28PM D2.00008 Anisotropy stress invariants of a large eddy simulation during a diurnal cycle for very large wind farms , RAL CAL, NASEEM ALI, NICHOLAS HAMILTON, Portland State University, GERARD CORTINA, MARC CALAF, University of Utah —

Reynolds stress invariants of the turbulent flow within large wind farms under different atmospheric flow stratification (stable, unstable and neutral); cases without turbines is also evaluated. Lumley triangle and barycentric map are used to quantify the anisotropic stress tensor within the wind farm and atmospheric boundary layer. Dependent on the thermal stratification, the unstable and neutral cases of the wind farm and no wind farm display the minimum second invariant in contrast to the stable case that shows the maximum invariants. Scaled color is used to present the invariant as a function of domain height. The unstable stratification approaches the isotropy limit at high layers of the domain and the stable stratification leads the turbulence flow to be one component flow. The principle eigenvalues are also shown, where they show its effects on the vicinity of the swept area of the rotor. Finally, spheroid visualization is pursued to understand and interpret the realizable turbulence flow within the wind farm and atmospheric boundary layer.

4:41PM D2.00009 Focused-based multifractal analysis of the wake in a wind turbine array utilizing proper orthogonal decomposition , HAWWA KADUM, NASEEM ALI, RAL CAL , Portland State University

— Hot-wire anemometry measurements have been performed on a 3×3 wind turbine array to study the multifractality of the turbulent kinetic energy dissipations. A multifractal spectrum and Hurst exponents are determined at nine locations downstream of the hub height, and bottom and top tips. Higher multifractality is found at 0.5D and 1D downstream of the bottom tip and hub height. The second order of the Hurst exponent and combination factor show an ability to predict the flow state in terms of its development. Snapshot proper orthogonal decomposition is used to identify the coherent and incoherent structures and to reconstruct the stochastic velocity using a specific number of the POD eigenfunctions. The accumulation of the turbulent kinetic energy in top tip location exhibits fast convergence compared to the bottom tip and hub height locations. The dissipation of the large and small scales are determined using the reconstructed stochastic velocities. The higher multifractality is shown in the dissipation of the large scale compared to small-scale dissipation showing consistency with the behavior of the original signals.

4:54PM D2.00010 Investigation of wake characteristics in wind farm varying turbulent inflow condition¹ , JISUNG NA, Yonsei Univ, EUNMO KOO, Los Alamos National Lab, MUÑOZ-ESPARZA DOMINGO, National Center for Atmospheric Research, EMILIA KYUNG JIN, Korea Institute of Atmospheric Prediction Systems, RODMAN LINN, Los Alamos National Lab, JOON SANG LEE, Yonsei Univ —

In this study, we investigate the wake characteristics in wind farm varying turbulent property at inlet condition. To solve the flow with wind turbines and its wake, we use large eddy simulation (LES) technique with actuator line method (ALM). The wake characteristics in wind farm is important mainly in performance of wind farm because non-fully recovered wake induced by upstream wind turbines interferes power generation at downstream wind turbines. Turbulent inflow which contains the information of turbulence in atmospheric boundary layer is one of the key factors for describing the wake in wind farm accurately. We perform the quantitative analysis of velocity deficit and turbulent intensity in whole cases. In the comparison between cases with and without turbulent inflow, we observe that wake in case with turbulent inflow is more diffused to span-wise direction. And we analyze the coherent structures behind wind turbines at each row. Through above-analysis, we reveal how the wake is interacted with performance of wind farm.

¹This work was supported by the National Research Foundation of Korea (NRF) Grant funded by the Korean Government (MSIP) (No.2015R1A5A1037668)

Sunday, November 20, 2016 2:57PM - 4:41PM –

Session D3 Vortex Dynamics: Applications on the Living Environment B110-111 - Douglas Hart, Massachusetts Institute of Technology

2:57PM D3.00001 PIV Analysis of Wake Induced by Real Harbor Seal Whiskers , JOSEPH BUNJEVAC, AIDAN RINEHART, JUSTIN FLAHERTY, WEI ZHANG, Cleveland State Univ — Harbor Seals are able to accurately detect minute disturbances in the ambient flow using their whiskers, which is attributed to the exceptional capability of the whiskers to suppress vortex-induced vibrations in the wake. To explore potential applications for designing smart devices, such as high-sensitivity underwater flow sensors and drag reduction components, research has studied the role of key parameters of the whisker morphology on wake structure. Due to the inherent variation in size and angle of incidence along the length of whiskers, it is not well understood how a real seal whisker changes wake structure, in particular the vortex shedding behavior. This work aims to understand the flow around a single real seal whisker using Particle Image Velocimetry at low Reynolds numbers (i.e. a few hundred) in a water channel. Variations in flow structure are inspected between several different real whiskers and whisker models. The results will provide insights of the effects of the natural geometry of the harbor seal whiskers on wake flow compared to idealized whisker-like models.

3:10PM D3.00002 How do seal whiskers suppress vortex shedding , AIDAN RINEHART, JUSTIN FLAHERTY, JOSEPH BUNJAVICK, Cleveland State Univ, VIKRAM SHYAM, NASA Glenn Research Center, WEI ZHANG, Cleveland State Univ — Certain seal whiskers possess a unique geometry that significantly reduces the vortex-induced vibration; which has attracted great attention to understand how the unique shape re-organizes the wake structure and its potential for passive flow control. The shape of the whiskers can be described as an elliptical cross-section that is lofted along the length of the whisker. Along the entire length of the whisker the ellipse varies in major and minor axis as well as angle of incidence with respect to the axis of the whisker. Of particular interest in this study is to identify what effect the angle of incidence has on the flow structure around the whisker, which has been overlooked in the past. The study will analyze the wake structure behind various scaled-up whisker models using particle image velocimetry (PIV). These whisker models share common geometry dimensions except for the angle of incidence. Flow conditions are created in a water channel and a wind tunnel, covering a wide range of Reynolds number (a few hundreds to thousands), similar to the ambient flow environment of seals and to the targeted aero-propulsion applications. This study will help address knowledge gaps in understanding of how certain geometry features of seal whiskers influence the wake and establish best practices for its application as effective passive flow control strategy.

3:23PM D3.00003 The effect of bottom friction on tidal dipolar vortices and the associated transport¹ , MATIAS DURAN-MATUTE, LEON KAMP, GERTJAN VAN HEIJST, Eindhoven Univ of Tech — Tidal dipolar vortices can be formed in a semi-enclosed basin as the tides flow in and out through an inlet. If they are strong enough to overcome the opposing tidal currents, these vortices can travel away from the inlet due to their self-propelling mechanism, and hence, act as an efficient transport agent for suspended material. We present results of two-dimensional numerical simulations of the flow through an idealized tidal inlet, with either a linear or a nonlinear parameterization of the bottom friction. We then quantify the effect of the bottom friction on the propagation of the dipolar vortex and on its ability as a transport agent by computing the flushing and residence times of passive particles. Bottom friction is detrimental to the ability of tidal dipolar vortices to propagate and hinders transport away from the inlet. The magnitude of this effect is related to the relative duration of the tidal period as compared to the typical decay time scale of the vortex dipole.

¹This research is funded by NWO (the Netherlands) through the VENI grant 863.13.022

3:36PM D3.00004 Turbulent coherent-structure dynamics in a natural surface storage zone: Mechanisms of mass and momentum transport in rivers¹ , CRISTIAN ESCAURIAZA, JÓRGE SANDOVAL, Pontif Univ Católica de Chile, EMMANUEL MIGNOT, Laboratoire de mécanique des fluides et d'acoustique, INSA de Lyon, France, LUCA MAO, Pontif Univ Católica de Chile — Turbulent flows developed in surface storage zones (SSZ) in rivers control many physical and biogeochemical processes of contaminants in the water. These regions are characterized by low velocities and long residence times, which favor particle deposition, nutrient uptake, and flow interactions with reactive sediments. The dynamics of the flow in SSZ is driven by a shear layer that induces multiple vortical structures with a wide range of temporal and spatial scales. In this work we study the flow in a lateral SSZ of the Lluta River, a high-altitude Andean stream (4,000 masl), with a Re=45,800. We describe the large-scale turbulent coherent structures using field measurements and 3D numerical simulations. We measure the bed topography, instantaneous 3D velocities at selected points, the mean 2D free-surface velocity field, and arsenic concentration in the sediment. Numerical simulations of the flow are also performed using a DES turbulence model. We focus on the mass and momentum transport processes, analyzing the statistics of mass exchange and residence times in the SSZ. With this information we provide new insights on the flow and transport processes between the main channel and the recirculating region in natural conditions.

¹supported by Fondecyt 1130940

3:49PM D3.00005 PIV measurements and flow characteristics downstream of mangrove root models ., AMIRKHOSRO KAZEMI, OSCAR CURET, Ocean and Mechanical Engineering — Mangrove forests attracted attentions as a solution to protect coastal areas exposed to sea-level rising, frequent storms, and tsunamis. Mangrove forests found in tide-dominated flow regions are characterized by their massive and complex root systems, which play a prominent role in the structure of tidal flow currents. To understand the role of mangrove roots in flow structure, we modeled mangrove roots with rigid and flexible arrays of cylinders with different spacing between them as well as different configurations. In this work, we investigate the fluid dynamics downstream of the models using a 2-D time-resolved particle image velocimetry (PIV) and flow visualization. We carried out experiments for four different Reynolds number based on cylinder diameters ranges from 2200 to 12000. We present time-averaged and time-resolved flow parameters including velocity distribution, vorticity, streamline, Reynolds shear stress and turbulent kinetic energy. The results show that the flow structure has different vortex shedding downstream of the cylinders due to interactions of shear layers separating from cylinders surface. The spectral analysis of the measured velocity data is also performed to obtain Strouhal number of the unsteady flow in the cylinder wake.

4:02PM D3.00006 Vortex wake of tip loaded rotors at low Reynolds numbers, OMER SAVAS, ONUR BILGI, Univ of California - Berkeley — The effect of tip tabs on the flow characteristics of a three bladed rotor is investigated using strain gauge thrust measurements, flow visualization and particle image velocimetry at chord Reynolds numbers of $0.4 - 2.9 \times 10^5$. The tab angles of attack of 0, $\pm 3^\circ$ & $\pm 5^\circ$ with respect to the rotation of the rotor are used to vary the tip loading. The rotor wakes and thrust characteristics at positive angles of attack, when the tip loading is outward, are qualitatively similar to those with no-tabs. In contrast, when the tip loading is inward at zero and negative angles of attack, the vortex wake is radically altered; the thrust nearly vanishes, even reverses with increasing inward loading. The key factors influencing the behavior of the wake are the vortex system off the tabs and their associated *downwash*, which is inward for the outward tab loading and causes increased volume and momentum flux and outward for the inward tab loading and causes expansion of the wake and nearly complete loss of thrust. At negative angles of attack, the flow fields exhibit a quasi-steady bound ring vortex system around at the edge of the rotor disk and the flow direction on the pressure side of the rotor disk reverses: it flows toward the rotor disk.

4:15PM D3.00007 Numerical study of the effects of rotating forced downdraft in reproducing tornado-like vortices, JINWEI ZHU, SHUYANG CAO, Tongji Univ, TETSURO TAMURA, Tokyo Institute of Technology, TOKYO INSTITUTE OF TECHNOLOGY COLLABORATION, TONGJI UNIV COLLABORATION — Appropriate physical modeling of a tornado-like vortex is a prerequisite to studying near-surface tornado structure and tornado-induced wind loads on structures. Ward-type tornado simulator modeled tornado-like flow by mounting guide vanes around the test area to provide angular momentum to converging flow. Iowa State University, USA modified the Ward-type simulator by locating guide vanes at a high position to allow vertical circulation of flow that creates a rotating forced downdraft in the process of generating a tornado. However, the characteristics of the generated vortices have not been sufficiently investigated till now. In this study, large-eddy simulations were conducted to compare the dynamic vortex structure generated with/without the effect of rotating forced downdraft. The results were also compared with other CFD and experimental results. Particular attention was devoted to the behavior of vortex wander of generated tornado-like vortices. The present study shows that the vortex center wanders more significantly when the rotating forced downdraft is introduced into the flow. The rotating forced downdraft is advantageous for modeling the rear flank downdraft phenomenon of a real tornado.

4:28PM D3.00008 From salps to robots: estimating thrust in propulsive pulsed jets using wake kinematics¹, ATHANASIOS ATHANASSIADIS, DOUGLAS HART, Massachusetts Inst of Tech-MIT — Both animals and robots can achieve high maneuverability underwater by using pulsed jets for propulsion. However, in cases where multiple jets are required, it remains unclear how jet placement and timing will affect propulsive performance. In recent experiments,² we demonstrate how vortex interactions reduce thrust production for simultaneously pulsed jets. Our results rely on force estimates using high-speed laser fluorescence imaging of the jet wakes. By combining measurements of wake kinematics with analytical models, we are able to estimate force production from just the fluorescence videos. In this talk, I will discuss the force estimation technique, and how this approach helped to reveal design strategies that would benefit from the wake interactions.

¹This work was supported by the Office of Naval Research

²A.G. Athanassiadis, and D.P Hart. Phys. Rev. Fluids, 1(3), 034501. (2016).

Sunday, November 20, 2016 2:57PM - 5:07PM – Session D4 Acoustics II: Aeroacoustics (Jets) B112 - Joseph Nichols, University of Minnesota

2:57PM D4.00001 Sensitivity analysis for the control of supersonic impinging jet noise¹, JOSEPH W. NICHOLS, NATHANIEL HILDEBRAND, Aerospace Engineering and Mechanics, University of Minnesota — The dynamics of a supersonic jet that impinges perpendicularly on a flat plate depend on complex interactions between fluid turbulence, shock waves, and acoustics. Strongly organized oscillations emerge, however, and they induce loud, often damaging, tones. We investigate this phenomenon using unstructured, high-fidelity Large Eddy Simulation (LES) and global stability analysis. Our flow configurations precisely match laboratory experiments with nozzle-to-wall distances of 4 and 4.5 jet diameters. We use multi-block shift-and-invert Arnoldi iteration to extract both direct and adjoint global modes that extend upstream into the nozzle. The frequency of the most unstable global mode agrees well with that of the emergent oscillations in the LES. We compute the “wavemaker” associated with this mode by multiplying it by its corresponding adjoint mode. The wavemaker shows that this instability is most sensitive to changes in the base flow slightly downstream of the nozzle exit. By modifying the base flow in this region, we then demonstrate that the flow can indeed be stabilized. This explains the success of microjets as an effective noise control measure when they are positioned around the nozzle lip.

¹Computational resources were provided by the Argonne Leadership Computing Facility.

3:10PM D4.00002 Linear frequency response analysis of a high subsonic and a supersonic jet, OLIVER SCHMIDT, TIM COLONIUS, California Institute of Technology, GUILLAUME BRS, CASCADE Technologies — A linear frequency response, or resolvent analysis of two turbulent jet mean flows is conducted. The mean flows are obtained from two high-fidelity large eddy simulations of a Mach 0.9 and a Mach 1.5 turbulent jet at Reynolds numbers of 1×10^6 and 3×10^5 , respectively. For both cases, curves of the optimal and sub-optimal output gains are calculated as a function of frequency for different azimuthal wavenumbers. The gain curves bring to light pseudo-resonances associated with different linear instability mechanisms. The same mechanisms are recovered in global stability analyses, and the results are compared. In the case of the Mach 0.9 jet, the resolvent analysis allows for a detailed study of trapped acoustic modes inside the potential core that were subject to previous stability studies. The structure of the resolvent and global modes are compared to POD mode estimates of the LES data. Additionally, the projection of the LES data onto the modes allows for quantitative assessment of how well the modal structures represent the coherent structures in the jet.

3:23PM D4.00003 Asymptotic structure of low frequency supersonic heated jet noise using LES data to re-construct a turbulence model¹, MOHAMMED AFSAR, University of Strathclyde, ADRIAN SESCU, VASILEIOS SASSANIS, Mississippi State University, GUILLAUME BRES, Cascade Technologies Inc., AARON TOWNE, SANJIVA LELE, Stanford University — The Goldstein-Sescu-Afsar (2012, vol. 695, pp. 199- 234) asymptotic theory postulated that the appropriate distinguished limit in which non-parallel mean flow effects introduces a leading order change in the propagator (which is related adjoint linearized Euler Greens function) within Goldstein's acoustic analogy must be when the jet spread rate is the same order as Strouhal number. We analyze the low frequency structure of the acoustic spectrum using Large-eddy simulations of two axi-symmetric jets (heated & unheated) at constant supersonic jet Mach number to obtain the mean flow for the asymptotic theory. This approach provides excellent quantitative agreement for the peak jet noise when the coefficients of the turbulence model are tuned for good agreement with the far-field acoustic data. Our aim in this talk, however, is to show the predictive capability of the asymptotics when the turbulence model in the acoustic analogy is exactly re-constructed by numerically matching the length scale coefficients of an algebraic-exponential model for the 1212-component of the Reynolds stress auto-covariance tensor (1 is streamwise & 2 is radial direction) with LES data at any spatial location and temporal frequency. In this way, all information is obtained from local unsteady flow.

¹We thank Professor Parviz Moin for supporting this work as part of the Center for Turbulence Research Summer Program 2016.

3:36PM D4.00004 Impact of surface proximity on flow and acoustics of a rectangular supersonic jet, EPHRAIM GUTMARK, FLORIAN BAIER, PABLO MORA, University of Cincinnati, KAILAS KAILSANATH, KAMAL VISWANATH, RYAN JOHNSON, Naval Research Laboratory — Advances in jet technology have pushed towards faster aircraft, leading to more streamlined designs and configurations, pushing engines closer to the aircraft frame. This creates additional noise sources stemming from interactions between the jet flow and surfaces on the aircraft body, and interaction between the jet and the ground during takeoff and landing. The paper studies the impact of the presence of a flat plate on the flow structures and acoustics in an $M=1.5$ ($NPR=3.67$) supersonic jet exhausting from a rectangular C-D nozzle. Comparisons are drawn between baseline cases without a plate and varying nozzle-plate distance at $NPRs$ from 2.5 to 4.5, and temperature ratios of up to 3.0. At the shielded side and sideline of the plate noise is mitigated only when the plate is at the nozzle lip ($h=0$). Low frequency mixing noise is increased in the downstream direction only for $h=0$. Screech tones that exist only for low NTR are fully suppressed by the plate at $h=0$. However, for $h>0$ the reflection enhances screech at both reflected side and sideline. Low frequency mixing noise is enhanced by the plate at the reflected side at all plate distances, while broad band shock associated noise is reduced only at the sideline for $h=0$. Increased temperature mitigates the screech tones across all test conditions. The results are compared to a circular nozzle of equivalent diameter with an adjacent plate.

3:49PM D4.00005 Toward forced-wavepacket jet-noise models, AARON TOWNE, Center for Turbulence Research, Stanford University, SANJIVA K. LELE, Stanford University, GUILLAUME A. BRES, Cascade Technologies — Large-scale hydrodynamic wavepackets have been identified by numerous studies as an important source of turbulent jet noise. Linear models have proven capable of predicting the average statistics of these wavepackets but severely under-predict the associated acoustic radiation in subsonic jets in particular. Further studies have suggested that this under-prediction can be attributed to the sensitivity of the far-field noise to second order statistics of the wavepackets that are not properly reproduced by fully linear models. One approach to incorporating nonlinear effects is the computation of so-called resolvent modes, which represent the linear response of the flow to a nonlinear forcing that is presumed to be temporally and spatially uncorrelated, i.e., white noise. This approach has delivered promising results (see for example Jeun et al., *Phys. Fluids* 2016), but its quantitative accuracy is limited by its implicit white-noise assumption. In this talk, we will show how correlated forcing can be systematically incorporated into a resolvent-based model and demonstrate the effect of applying modeled forcing that is designed to mimic the actual nonlinear terms present in a Mach 0.9 turbulent jet.

4:02PM D4.00006 Acoustic wavepackets and sound radiation by jets, UNNIKRISHNAN SASIDHARAN NAIR, DATTA GAITONDE, Ohio State Univ - Columbus — The three-dimensional spatio-temporal evolution of the acoustic mode in a supersonic jet is analyzed using Doak's Momentum Potential Theory on an LES database. The acoustic mode exhibits a well-defined wavepacket nature in the core and convects at sonic speed. Its spatial coherence is significantly higher than the hydrodynamic component, resulting in an efficient sound radiation mechanism dominated by the axisymmetric and the first helical modes. Enthalpy transport by the acoustic mode yields insight into the sound energy flux emitted by the jet. Intrusion and ejection of coherent vortices into the core and ambient outer fluid respectively are found to be major intermittent sources of acoustic radiation. The scalar potential which defines the acoustic mode is found to satisfy the homogenous wave propagation equation in the nearfield which makes it a suitable variable to predict farfield radiation. The propagated acoustic field closely resembles the corresponding nearfield LES result. The acoustic mode thus provides a physically consistent wavepacket model to predict sound radiation from jets. Ongoing efforts on subsonic jets will discern the influence, if any, of the Mach number on the model.

4:15PM D4.00007 Jet crackle: skewness transport budget and a mechanistic source model, DAVID BUCHTA, JONATHAN FREUND, University of Illinois at Urbana-Champaign — The sound from high-speed (supersonic) jets, such as on military aircraft, is distinctly different than that from lower-speed jets, such as on commercial airliners. Atop the already loud noise, a higher speed adds an intense, fricative, and intermittent character. The observed pressure wave patterns have strong peaks which are followed by relatively long shallows; notably, their pressure skewness is $S_k \geq 0.4$. Direct numerical simulation of free-shear-flow turbulence show that these skewed pressure waves occur immediately adjacent to the turbulence source for $M \geq 2.5$. Additionally, the near-field waves are seen to intersect and nonlinearly merge with other waves. Statistical analysis of terms in a pressure skewness transport equation show that starting just beyond δ_{99} the nonlinear wave mechanics that add to S_k are balanced by damping molecular effects, consistent with this aspect of the sound arising in the source region. A gas dynamics description is developed that neglects rotational turbulence dynamics and yet reproduces the key crackle features. At its core, this mechanism shows simply that nonlinear compressive effects lead directly to stronger compressions than expansions and thus $S_k > 0$.

4:28PM D4.00008 A combustion model for studying the effects of ideal gas properties on jet noise, JERIN JOSEPH, The University of Texas at Austin, Aerospace Engineering Department, CHARLES TINNEY, The University of Texas at Austin, Applied Research Laboratories — A theoretical combustion model is developed to simulate the influence of ideal gas effects on various aeroacoustic parameters over a range of equivalence ratios. The motivation is to narrow the gap between laboratory and full-scale jet noise testing. The combustion model is used to model propane combustion in air and kerosene combustion in air. Gas properties from the combustion model are compared to real lab data acquired at the National Center for Physical Acoustics at the University of Mississippi as well as outputs from NASA's Chemical Equilibrium Analysis code. Different jet properties are then studied over a range of equivalence ratios and pressure ratios for propane combustion in air, kerosene combustion in air and heated air. The findings reveal negligible differences between the three constituents where the density and sound speed ratios are concerned. Albeit, the area ratio required for perfectly expanded flow is shown to be more sensitive to gas properties, relative to changes in the temperature ratio.

4:41PM D4.00009 Towards the characterization of noise sources in a supersonic three-stream jet using advanced analysis tools, CHRISTOPHER RUSCHER, SIVARAM GOGINENI, Spectral Energies, LLC — Strict noise regulation set by governing bodies currently make supersonic commercial aviation impractical. One of the many challenges that exist in developing practical supersonic commercial aircraft is the noise produced by the engine's exhaust jet. A promising method of jet noise reduction for supersonic applications is through the addition of extra exhaust streams. Data for an axisymmetric three-stream nozzle were generated using the Naval Research Laboratory's JENRE code. This data will be compared to experimental results obtained by NASA for validation purposes. Once the simulation results show satisfactory agreement to the experiments, advanced analysis tools will be applied to the simulation data to characterize potential noise sources. The tools to be applied include methods that are based on proper orthogonal decomposition, wavelet decomposition, and stochastic estimation. Additionally, techniques such as empirical mode decomposition and momentum potential theorem will be applied to the data as well.

4:54PM D4.00010 Understanding sideline jet noise using input-output analysis¹, JINAH JEUN, JOSEPH W. NICHOLS, Department of Aerospace Engineering and Mechanics, Univ of Minnesota - Twin Cities, MIHAIRO R. JOVANOVIC, Department of Electrical and Computer Engineering, Univ of Minnesota - Twin Cities — We apply input-output analysis to high-speed turbulent jets to obtain the far-field acoustic response at different radiation angles. We consider both axisymmetric and higher azimuthal modes over a range of different frequencies to investigate the resulting noise spectra. At each frequency, singular value decomposition of the resolvent operator distinguishes between the optimal mode and several sub-optimal input-output modes by the magnitude of corresponding singular value. While both types of modes resemble wavepackets, the optimal mode associated with the largest singular value is superdirective in the peak noise radiation angle. Sub-optimal modes, in contrast, appear increasingly omnidirectional, rotating progressively to the sideline direction. Our analysis also recovers a broadening of the far-field acoustic spectra as the radiation angle increases. We show that a significant amount of the entire acoustic field can be captured by a superposition of a small number of coherent input-output modes.

¹The LES data was produced as part of a Cascade Technologies STTR project sponsored by NAVAIR, under the supervision of Dr. John T. Spyropoulos. The LES calculations were performed at the ERDC supercomputer center.

Sunday, November 20, 2016 2:57PM - 5:07PM –
Session D5 Compressible Flow: Shock-Boundary Layer Interaction B113 - Venkateswaran Narayanaswamy, North Carolina State University

2:57PM D5.00001 Experimental Investigation of SBLI to Unravel Inlet Unstart Physics¹, MORGAN FUNDERBURK, VENKATESWARAN NARAYANASWAMY, North Carolina State Univ — The phenomenon of shock boundary layer interaction (SBLI) driven inlet unstart persists as one of the most significant problems facing supersonic ramjet/scramjet engines. In order to determine how the characteristics of the SBLI units specific to rectangular inlets evolve during an unstart event, an experimental investigation is made using surface streakline methods and pitot/wall pressure measurements in the vicinity of the floor and corner SBLI induced by a compression ramp in a rectangular channel. Mean and unsteady measurements were taken at a variety of shock strengths to simulate the evolution of the combustion-induced back pressure ratio during unstart. The freestream Mach number was also varied. Statistical correlation methods were used to determine the degree of interaction between the floor and corner SBLI with different flowfield locations for the various test conditions. Finally, comparison to a two-dimensional compression ramp SBLI was made to determine any modification caused by the introduction of the corner SBLI. Results indicate that the floor and corner SBLI transition from distinct units to members of a global separated flow with increasing back pressure, and that considerable modification of the floor SBLI by the corner flow occurs.

¹AFOSR Grant FA9550-15-1-0296

3:10PM D5.00002 Sensitivity of shock boundary-layer interactions to weak geometric perturbations¹, JI HOON KIM, JOHN K. EATON, Stanford University — Shock-boundary layer interactions can be sensitive to small changes in the inlet flow and boundary conditions. Robust computational models must capture this sensitivity, and validation of such models requires a suitable experimental database with well-defined inlet and boundary conditions. To that end, the purpose of this experiment is to systematically document the effects of small geometric perturbations on a SBLI flow to investigate the flow physics and establish an experimental dataset tailored for CFD validation. The facility used is a Mach 2.1, continuous operation wind tunnel. The SBLI is generated using a compression wedge; the region of interest is the resulting reflected shock SBLI. The geometric perturbations, which are small spanwise rectangular prisms, are introduced ahead of the compression ramp on the opposite wall. PIV is used to study the SBLI for 40 different perturbation geometries. Results show that the dominant effect of the perturbations is a global shift of the SBLI itself. In addition, the bumps introduce weaker shocks of varying strength and angles, depending on the bump height and location. Various scalar validation metrics, including a measure of shock unsteadiness, and their uncertainties are also computed to better facilitate CFD validation.

¹Ji Hoon Kim is supported by an OTR Stanford Graduate Fellowship.

3:23PM D5.00003 3D Plenoptic PIV Measurements of a Shock Wave Boundary Layer Interaction¹, BRIAN THUROW, JOHNATHAN BOLTON, Auburn University, NISHUL ARORA, FARRUKH ALVI, Florida State University — Plenoptic particle image velocimetry (PIV) is a relatively new technique that uses the computational refocusing capability of a single plenoptic camera and volume illumination with a double-pulsed light source to measure the instantaneous 3D/3C velocity field of a flow field seeded with particles. In this work, plenoptic PIV is used to perform volumetric velocity field measurements of a shock-wave turbulent boundary layer interaction (SBLI). Experiments were performed in a Mach 2.0 flow with the SBLI produced by an unswept fin at 15° angle of attack. The measurement volume was $38 \times 25 \times 32 \text{ mm}^3$ and illuminated with a 400 mJ/pulse Nd:YAG laser with 1.7 microsecond inter-pulse time. Conventional planar PIV measurements along two planes within the volume are used for comparison. 3D visualizations of the fin generated shock and subsequent SBLI are presented. The growth of the shock foot and separation region with increasing distance from the fin tip is observed and agrees with observations made using planar PIV. Instantaneous images depict 3D fluctuations in the position of the shock foot from one image to the next.

¹The authors acknowledge the support of the Air Force Office of Scientific Research

3:36PM D5.00004 The Effect of Configuration on Shock/Boundary Layer Interaction Unsteadiness, JAMES THREADGILL, University of Arizona, PAUL BRUCE, Imperial College London — Low-frequency flow unsteadiness associated with shock/boundary layer interactions (SBLIs) remain poorly understood. Upstream and downstream mechanisms have been observed to drive the dynamics, with the latter more prevalent in higher strength interactions. Studies have typically focused on single SBLI configurations within a given environment, limiting identification of unique characteristic behaviors. An investigation has been conducted to assess the unsteady behavior of various 2D SBLIs, each with a range of interaction strengths, all tested within a single facility. Experiments were conducted in Mach 2 flow with $Re_\theta = 8000$, featuring 14° and 20° compression ramps, and impinging shock reflections from 7° , 8° , 9° , and 10° shock generators. Resultant SBLIs were analyzed using high-speed planar PIV and a streamwise array of fast-response wall-pressure transducers. High-frequency energy content of the shock motion is observed to be independent of the configuration. The dominance of the downstream mechanism in low-frequency unsteadiness is related to the SBLI configuration as well as the interaction strength. In addition, correlations between shock position and angle, and the separated near-wall flow structure are directly established.

3:49PM D5.00005 50 kHz PIV of a Swept-Ramp Shock-Wave Boundary-Layer Interaction at Mach 2.¹, LEON VANSTONE, The University of Texas at Austin, MUSTAFA NAIL MUSTA, Necmettin Erbakan University, SERDAR SECKIN, MOHAMMAD SALEEM, NOEL CLEMENS, The University of Texas at Austin — The interaction from a 30° sweep, 22.5° compression ramp in a Mach 2 flow is examined using wide-field 5Hz and 50 kHz PIV. The high-speed PIV is fast enough to resolve the large-scale unsteady motions of the SWBLI and can be band-pass filtered to investigate the driving mechanisms of unsteadiness and the widefield PIV allows comparisons with mean flow-fields. Preliminary investigation looked at three distinct frequency bands: 10-50 kHz ($0.025\text{--}0.25 U_\infty/\delta_{99}$), 1-10 kHz ($0.025\text{--}0.25 U_\infty/\delta_{99}$), and 0-1 kHz ($0\text{--}0.025 U_\infty/\delta_{99}$). The unsteadiness associated with 10-50 kHz shows no correlation with the upstream boundary layer and accounts for 40

¹This work is sponsored by the AFOSR under grant FA9550-14-1-0167 with Ivett Leyva as the program manager. This source of support is gratefully acknowledged. Further, Mustafa Musta thanks the Scientific and Technological research Council of Turkey.

4:02PM D5.00006 Influence of Mach Number and Incoming Boundary Layer on Shock Boundary Layer Interaction, ILONA STAB, JAMES THREADGILL, JESSE LITTLE, University of Arizona — Wall pressure fluctuations, schlieren imaging, oil flow visualization and PIV measurements have been performed on the shock boundary layer interaction (SBLI) formed by a 10° compression ramp. The incoming Mach number and boundary layer characteristics are varied to examine their influence on the SBLI. Focus is placed on understanding the effect of these parameters on the structure and unsteadiness of the resultant interaction. Lower Mach numbers $M = 2.3$ ($\delta_0 = 1.7 \text{ mm}$, $\theta = 0.29 \text{ mm}$, $Re_\theta = 3115$, $H = 1.4$) and $M = 3$ ($\delta_0 = 1.3 \text{ mm}$, $\theta = 0.25 \text{ mm}$, $Re_\theta = 1800$, $H = 1.8$) show a turbulent or transitional approach boundary layer with no apparent separation at the ramp. Mach 4 has a large separated region which is seemingly a result of a now laminar or transitional approach boundary layer. Pulsations in the separated region correspond to the expected low frequency SBLI dynamics showing a broad peak around a Strouhal number of $St = fL_{int}/U_\infty = 0.27$ which is lower than the characteristic frequency of the turbulent boundary layer. Additional results examining the influence of boundary layer modifications (e.g. sweep) and wind tunnel side-walls are also included.

4:15PM D5.00007 Shock wave boundary layer interaction in jet injection into supersonic crossflow, NITHIYARAJ MUNUSWAMY, RAGHURAMAN N GOVARDHAN, Indian Institute of Science — Jet injection into supersonic crossflow results in a bow shock forming upstream of the injected jet. In the present work, we study the unsteady interactions of this shock with the structures in the incoming boundary layer. The studies are done with a sonic air jet injected into a supersonic air crossflow at a Mach number of 2.5 with jet momentum ratios from 1.5 to 3. The interactions of the shock and the incoming boundary layer are measured using PIV in two perpendicular planes, one perpendicular to the wall from which the jet is injected and the other parallel to the wall and within the boundary layer. These measurements enable determination of both structures within the boundary layer, such as low and high speed streaks, and the instantaneous location of the bow shock, in addition to the jet penetration at that instant. The detailed analysis of instantaneous and mean flow quantities for different momentum flux ratios obtained from a large set of instantaneous PIV fields will be presented at the conference.

4:28PM D5.00008 Swept Impinging Oblique Shock/Boundary-Layer Interactions¹, JESSE LITTLE, JAMES THREADGILL, ILONA STAB, ADAM DOEHRMANN, University of Arizona — Oblique shock waves impinging on boundary layers are common flow features associated with high-speed flows around complex body geometries and through internal channel flows. The increasingly three-dimensional surface geometries of modern vehicles has led to a prevalence of complex shock/boundary-layer interactions. Sweep has been observed to vary the interaction structure, unsteadiness, and similarity scalings. Sharp-fins and highly-swept ramps have been noted to induce a quasi-conical development of the interaction, in contrast to a quasi-cylindrical scaling observed in low-sweep interactions. However, swept impinging oblique shock cases have largely been overlooked, with evidence of only cylindrical similarities observed in hypersonic conditions. Flow deflection beyond the maximum turning angle has been proposed as the mechanism for conical interaction development but such behavior has not been established for the present configuration. This study examines the effect of sweep on the interaction induced by a 12.5° generator in Mach 2.3 flow using oil-flow, Schlieren and PIV. Results document the development of similarity scalings at various angles of sweep, and highlight the difficulty in replicating a quasi-infinite span conditions in a moderately sized wind tun

¹Supported by the Air Force Office of Scientific Research (FA9550-15-1-0430) and Raytheon Missile Systems

4:41PM D5.00009 Contraction ratio effect on boundary layer separation induced by shockwave boundary layer interactions, SEONGKYUN IM, GIOVANNI DI CRISTINA, Univ of Notre Dame, HYUNGROK DO, Seoul National University — Boundary layer separations induced by shockwave boundary layer interaction at various contraction ratios were investigated at a Mach 4.5 flow. Stagnation pressure and temperature condition of 10 bars and 295 K were used, and a high-speed schlieren system visualized the flow features. A shockwave generator with 12 degree wedge generated an impinging shockwave onto a laminar boundary layer on a flat plate. The contraction ratio of the flow was varied by changing the distance between the shockwave generator and the flat plate. The location of the shockwave impingement was fixed while the contraction ratios were changed. Flow visualization showed that the flow separation and its size were influenced by the contraction ratio although overall flow features were similar. At higher contraction ratio, stronger impinging shockwave and more severe flow separation were observed.

4:54PM D5.00010 DSMC simulations of leading edge flat-plate boundary layer flows at high Mach number, DR. SAHADEV PRADHAN, Department of Chemical Engineering, Indian Institute of Science, Bangalore-560 012, India. — The flow over a 2D leading-edge flat plate is studied at Mach number $Ma = (U_{inf}/\sqrt{k_B T_{inf}/m})$ in the range $< Ma < 10$, and at Reynolds number $Re = (L_T U_{inf} \rho_{inf}) / \mu_{inf}$ equal to 10 using two-dimensional (2D) direct simulation Monte Carlo (DSMC) simulations to understand the flow phenomena of the leading-edge flat plate boundary layer at high Mach number. Here, L_T is the characteristic dimension, U_{inf} and T_{inf} are the free stream velocity and temperature, ρ_{inf} is the free stream density, m is the molecular mass, μ_{inf} is the molecular viscosity based on the free stream temperature T_{inf} , and k_B is the Boltzmann constant. The variation of streamwise velocity, temperature, number-density, and mean free path along the wall normal direction away from the plate surface is studied. The qualitative nature of the streamwise velocity at high Mach number is similar to those in the incompressible limit (parabolic profile). However, there are important differences. The amplitudes of the streamwise velocity increase as the Mach number increases and turned into a more flatter profile near the wall. There is significant velocity and temperature slip ((Pradhan and Kumaran, J. Fluid Mech-2011); (Kumaran and Pradhan, J. Fluid Mech-2014)) at the surface of the plate, and the slip increases as the Mach number is increased. It is interesting to note that for the highest Mach numbers considered here, the streamwise velocity at the wall exceeds the sound speed, and the flow is supersonic throughout the flow domain.

Sunday, November 20, 2016 2:57PM - 4:41PM –

Session D6 Aerodynamics: Rotation and Passive Control Strategies B114 - Tim Colonius, California Institute of Technology

2:57PM D6.00001 On the rotation and pitching of flat plates, YAQING JIN, SHENG JI,

LEONARDO P. CHAMORRO, University of Illinois at Urbana-Champaign — Wind tunnel experiments were performed to characterize the flow-induced rotation and pitching of various flat plates as a function of the thickness ratio, the location of the axis of rotation and turbulence levels. High-resolution telemetry, laser tachometer, and hotwire were used to get time series of the plates motions and the signature of the wake flow at a specific location. Results show that a minor axis offset can induce high-order modes in the plate rotation under low turbulence due to torque unbalance. The spectral decomposition of the flow velocity in the plate wake reveals the existence of a dominating high-frequency mode that corresponds to a static-like vortex shedding occurring at the maximum plate pitch, where the characteristic length scale is the projected width at maximum pitch. The plate thickness ratio shows inverse relation with the angular velocity. A simple model is derived to explain the linear relation between pitching frequency and wind speed. The spectra of the plate rotation show nonlinear relation with the incoming turbulence, and the dominating role of the generated vortices in the plate motions.

3:10PM D6.00002 An Experimental Study on the aerodynamic and aeroacoustic performances of Maple-Seed-Inspired UAV Propellers¹, HUI HU, ZHE NING, Iowa State University — Due to the auto-rotating trait of maple seeds during falling down process, flow characteristics of rotating maple seeds have been studied by many researchers in recent years. In the present study, an experimental investigation was performed to explore maple-seed-inspired UAV propellers for improved aerodynamic and aeroacoustic performances. Inspired by the auto-rotating trait of maple seeds, the shape of a maple seed is leveraged for the planform design of UAV propellers. The aerodynamic and aeroacoustic performances of the maple-seed-inspired propellers are examined in great details, in comparison with a commercially available UAV propeller purchased on the market (i.e., a baseline propeller). During the experiments, in addition to measuring the aerodynamic forces generated by the maple-seed-inspired propellers and the baseline propeller, a high-resolution Particle Image Velocimetry (PIV) system was used to quantify the unsteady flow structures in the wakes of the propellers. The aeroacoustic characteristics of the propellers are also evaluated by leveraging an anechoic chamber available at the Aerospace Engineering Department of Iowa State University.

¹The research work is supported by National Science Foundation under award numbers of OSIE-1064235.

3:23PM D6.00003 Dynamic flow reattachment on a rotating blade undergoing dynamic stall, VRISHANK RAGHAV, NARAYANAN KOMERATH, Georgia Institute of Technology — A 2-bladed rigid rotor undergoing retreating blade dynamic stall in a low-speed wind tunnel was used to study the 3-dimensional flow reattachment at the end of the dynamic stall cycle. Phase-locked stereoscopic Particle Image Velocimetry was used to capture the velocity field during reattachment. Continuing from prior studies on the inception and progression of 3-D rotating dynamic stall for this test case, phase-resolved, ensemble-averaged results are presented for different values of rotor advance ratio at varying spanwise stations along the blade. The results show the nominal reattachment getting delayed in rotor azimuth with higher advance ratio. At low advance ratio reattachment starts at the leading-edge and progresses towards the trailing-edge with vortex shedding transporting excess vorticity away from the leading-edge. At higher advance ratio, vortex shedding is not observed, instead the vortical structure shrinks in size while the flow close to the trailing-edge appears to reattach. At the higher advance ratio conditions, spanwise vorticity transport appears to be the mechanism to transport excess vorticity away from the leading-edge. The possible causes for a switch in mechanism of vorticity transport are also presented.

3:36PM D6.00004 Lift-optimal aspect ratio of a revolving wing at low Reynolds number¹, THIERRY JARDIN, Institut Supérieur de l'Aéronautique et de l'Espace (ISAE-Supaero), Université de Toulouse, 31055 Toulouse Cedex 4, France, TIM COLONIUS, Division of Engineering and Applied Science, California Institute of Technology, Pasadena, CA 91125, USA — Lentink & Dickinson (2009) showed that rotational acceleration stabilized the leading-edge vortex on revolving, low-aspect-ratio wings, and hypothesized that a Rossby number of around 3, which is achieved during each half-stroke for a variety of hovering insects, seeds, and birds, represents a convergent high-lift solution across a range of scales in nature. Subsequent work has verified that, in particular, the Coriolis acceleration is responsible for LEV stabilization. Implicit in these results is that there exists an optimal aspect ratio for wings revolving about their root, because it is otherwise unclear why, apart from possible physiological reasons, the convergent solution would not occur for an even lower Rossby number. We perform direct numerical simulations of the flow past revolving wings where we vary the aspect ratio and Rossby numbers independently by displacing the wing root from the axis of rotation. We show that the optimal lift coefficient represents a compromise between competing trends where the coefficient of lift increases monotonically with aspect ratio, holding Rossby number constant, but decreases monotonically with Rossby number, when holding aspect ratio constant. For wings revolving about their root, this favors wings of aspect ratio between 3 and 4.

¹The authors gratefully acknowledge support from Fondation ISAE-Supaero

3:49PM D6.00005 The effects of leading edge roughness on dynamic stall , JOHN HRYNUK,

Army Research Lab — Dynamic stall is a fundamental flow phenomenon that is commonly observed for insect flight and rotorcraft. Under certain conditions a leading edge vortex forms generating large but temporary lift forces. Historically, computations studying dynamic stall on airfoil shapes have struggled to predict this vortex formation time and separation point. Reduced order models and CFD have performed well when experiments have been performed to develop separation models, but this has limited the development of robust design tools. The current study looks at the effect of leading edge surface roughness on the formation of the Dynamic Stall Vortex (DSV). Roughness elements were applied to the leading edge of a NACA 0012 airfoil and PIV data of the vortex formation process was recorded. Measurements were taken at a Reynolds number of $Re = 12,000$ and baseline smooth NACA 0012 data was also recorded for comparison. Surface roughness elements, below the typical scale modeled by CFD, are shown to change DSV formation angle and location.

4:02PM D6.00006 Turbulent separation delay via tuned wall-impedance on a NACA 4412 airfoil in pre-stalled conditions. , JULIEN BODART, GRIGORY SHELEKHOV, ISAE-Supaero, DAEP, CARLO SCALO, University of Purdue, School of Engineering, LAURENT JOLY, ISAE-Supaero, DAEP

— We have performed large-eddy simulations of turbulent separation control via imposed wall-impedance on a NACA-4412 airfoil in near-stalled conditions (Mach, $M_\infty = 0.3$, and chord-Reynolds numbers, $Re_c = 1.5 \times 10^6$ and angle of attack, $\alpha = 14^\circ$), inspired by the experimental setup of Coles & Woodcock (1979). We impose complex impedance boundary conditions (IBCs) using the implementation developed by Scalo, Bodart and Lele, *Phys. Fluids* **27**, 035107 (2015), representing an array of sub-surface-mounted tunable Helmholtz cavities with resonant frequency, f_{res} , covered by a porous sheet with permeability inversely proportional to the impedance resistance. Generation of spanwise-oriented Kelvin-Helmholtz (KH) rollers of size $l_{KH,0} \simeq U_\infty/f_{res}$ observed in areas of sustained mean shear, which are convectively amplified along the shear-layer and reenergizing the separated flow via vortical-induced mixing and entrainment of irrotational fluid. Their characteristic initial size $l_{KH,0}$ is determined by the periodic wall-transpiration pattern induced, in turn, by acoustic resonance in the cavities. Several resonant frequencies and impedance have been tested, bracketing optimal conditions for control.

4:15PM D6.00007 MOVED TO G35.006 —

4:28PM D6.00008 Aerodynamic performance of an airfoil with a prescribed wall protuberance at low Reynolds numbers. , CARLOS DUQUE-DAZA, CRISTIAN MEJIA, DIEGO CAMACHO, Dept. of Mechanical and Mechatronics Engineering, Universidad Nacional de Colombia, DUNCAN LOCKERBY, School of Engineering, The University of Warwick — Numerical simulations of flow around a modified NACA0012 airfoil, featuring a small surface perturbation on the upper wall, were performed at two low Reynolds numbers. The aerodynamic performance was examined under conditions of incompressible steady state flow. Simulations at different angles of attack (AOA) were performed: 0, 6, 9.25 and 12 degrees for $Re=5000$, and 6, 9.25 and 12 for $Re=50000$. The effect of the wall-perturbation was assessed in terms of changes of drag and lift coefficients, and alterations of the upper wall turbulent boundary layer. Examination of mean velocity profiles reveals that the wall perturbation promotes boundary-layer separation near the leading edge and increase of the skin friction drag. An arguably improvement of the effectiveness, i.e. ratio of lift to drag, was observed for the modified profile for $Re = 5000$, especially at AOA of 6 degrees. This effect seems to be caused by a double effect: boundary layer separation approaching the leading edge and an increase of the lift coefficient caused by the larger pressure drop on the upper surface. The effect of the perturbation was always negative for the airfoil operating at $Re=50000$, independently of AOA.

Sunday, November 20, 2016 2:57PM - 5:07PM —

Session D7 Flow Control: Drag Reduction B115 - Mohamed Gad-el-Hak, Virginia Commonwealth University

2:57PM D7.00001 Active flow control for a NACA-0012 Profile: Part II , H. OUALLI, M.

MAKADEM, H. OUCHENE, A. FERFOURI, École 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 to a NACA-0012 profile. The experiments are conducted in a wind tunnel. Using a high-resolution visible-light camera and tomography, flow visualizations are carried out. LES finite-volume 3D 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 (B. Am. Phys. Soc., vol. 60, no. 27, p. 247, 2015) for the same profile but clipped at an angle of 60 deg, instead of the original 90 deg. The Reynolds number range is extended to 500,000, thus covering the flying regimes of micro-UAVs, UAVs, as well as small aircraft. When the nascent cavity is open and the attack angle is 30 deg, the drag coefficient is increased by 1,300%, as compared to the uncontrolled case. However, when the cavity is covered and $Re \leq 10^5$, a relatively small frequency, $f \leq 30$ Hz, is required for the drag coefficient to drop to negative values. At the maximum Reynolds number, thrust is generated but only at much higher frequencies, $12 \leq f \leq 16$ kHz.

3:10PM D7.00002 Direct numerical simulation of turbulent channel flow over a liquid-infused micro-grooved surface , JAEHEE CHANG, Seoul National University, TAEYONG JUNG, LG Electronics, HAECHON CHOI, Seoul National University, JOHN KIM, University of California, Los Angeles — Recently a superhydrophobic surface has drawn much attention as a passive device to achieve high drag reduction. Despite the high performance promised at ideal conditions, maintaining the interface in real flow conditions is an intractable problem. A non-wetting surface, known as the slippery liquid-infused porous surface (SLIPS) or the lubricant-impregnated surface (LIS), has shown a potential for drag reduction, as the working fluid slips at the interface but cannot penetrate into the lubricant layer. In the present study, we perform direct numerical simulation of turbulent channel flow over a liquid-infused micro-grooved surface to investigate the effects of this surface on the interfacial slip and drag reduction. The flow rate of water is maintained constant corresponding to $Re_\tau \sim 180$ in a fully developed turbulent channel flow, and the lubricant layer is shear-driven by the turbulent water flow. The lubricant layer is also simulated with the assumption that the interface is flat (i.e. the surface tension effect is neglected). The solid substrate in which the lubricant is infused is modelled as straight ridges using an immersed boundary method. DNS results show that drag reduction by the liquid-infused surface is highly dependent on the viscosity of the lubricant.

3:23PM D7.00003 Drag reduction of boat-tailed bluff bodies through transverse grooves. Part I: experiments, ALESSANDRO MARIOTTI, GUIDO BURESTI, MARIA VITTORIA SALVETTI, DICI - University of Pisa — The reduction of the aerodynamic drag of elongated axisymmetric bluff bodies is interesting for several applications. One well-known method to reduce the drag of this type of body is a geometrical modification called boat-tailing, consisting in a gradual reduction of the body cross-section before a sharp-edged base. We combine boat-tailing with properly contoured transverse grooves to further delay boundary-layer separation and reduce drag. The considered geometry is axisymmetric with an elliptical forebody and a cylindrical main body followed by a circular-arc boat-tail. The effectiveness of the contoured grooves was assessed through experiments and simulations. In this talk the experimental investigation is presented. Pressure measurements show that the introduction of a single transverse groove leads to a significant increase of the pressure on the body base and, consequently, to a reduction of drag compared with the boat-tail without the groove. Velocity measurements and flow visualizations highlight that this is due to a delay of flow separation over the boat tail. A steady local recirculation is present inside the groove and downstream its reattachment the boundary layer is thinner and has higher momentum than in the case with no groove, allowing separation to be delayed.

3:36PM D7.00004 Drag reduction of boat-tailed bluff bodies through transverse grooves. Part II: large-eddy simulations, MARIA VITTORIA SALVETTI, ALESSANDRO MARIOTTI, GUIDO BURESTI, DICI - University of Pisa — The present work focuses on strategies for aerodynamic drag reduction of elongated axisymmetric bluff bodies, which can be viewed as simplified models of a road vehicles. We combine boat-tailing, i.e. a gradual reduction of the body cross-section before a sharp-edged base, with properly contoured transverse grooves. The effectiveness of this strategy was assessed through experiments and simulations. Experiments showed that the introduction of a single groove leads to a further delay of boundary-layer separation and to a reduction of drag compared with the boat-tail configuration without grooves. In this talk, we present Large-Eddy Simulations (LES). LES results agree with the experimental findings. The success of the proposed flow control strategy is due to the relaxation of the no-slip condition in the small recirculation region inside the groove, which reduces the momentum losses near the wall and thus delays boundary layer separation. The effects of the introduction of the groove on the mean topology and on the dynamics of the near wake are also highlighted. Finally, a sensitivity analysis of the proposed control strategy efficiency to the groove location and to the boat-tail geometry is shown.

3:49PM D7.00005 Drag reduction of a streamlined body at incidence using rotating cylinders, JAMES SCHULMEISTER, MICHAEL TRIANTAFYLLOU, MIT — The flow past a streamlined body at incidence is characterized by cross-flow separation that induces large forces and moments. We investigate the use of counter-rotating control cylinders to delay separation and reduce the drag on a streamlined body at incidence in water tunnel experiments. A streamlined body model with rotating control cylinders was fixed at angles of incidence up to 30 degrees in a water tunnel while the forces and moments were monitored. The control cylinders have diameters equal to 10% of the maximum diameter of the streamlined body and are embedded in the model such that part of the circumference is exposed to the flow. The control cylinders are counter-rotated so that the moving surface imparts momentum to the flow, encouraging the delay of cross-flow separation and the reduction of drag.

4:02PM D7.00006 Turbulent drag reduction with liquid-infused surfaces¹, ALEXANDER SMITS, Princeton University and Monash University, TYLER VAN BUREN, Princeton University — We present turbulent skin friction reduction over liquid-impregnated surfaces in Taylor-Couette flow. The surface of the inner cylinder of the facility contains square grooves, with widths from 100 μm to 800 μm and a fixed liquid area of half the total area. Alkane liquids are infused in the surface with viscosities from $\sim 1/3$ to 2 times that of water. For Reynolds numbers up to $Re_d=10,500$ corresponding to a flow shear of $\tau=50 \text{ Pa}$, we achieve drag reduction exceeding 30%, three times higher than ever reported for liquid-infused surfaces.

¹Supported by the ONR through MURI Grant Nos. N00014-12-1-0875 and N00014-12-1-0962

4:15PM D7.00007 Large Structures of Drag-Reducing Pipe Flow by Surfactant Additives., YUKI KISHITA, Tokyo Institute of Technology, YOSHITSUGU NAKA, Meiji University, YUKI MINAMOTO, MASAYASU SHIMURA, MAMORU TANAHASHI, Tokyo Institute of Technology — Characteristics of drag-reducing turbulent pipe flows with surfactant additives have been investigated using stereoscopic particle image velocimetry. Measurements have been performed for a case with surfactant solution of 150 ppm at different Reynolds numbers: $Re_d = 31254, 58268, 85556$, around the maximum drag-reduction. Two distinct peaks are observed in streamwise velocity fluctuations around $y/R=0.07, 0.25$ and weak peaks are observed in radial velocity fluctuations at the same locations, where the Reynolds shear stress is negative. The deviations toward $u'_z > 0, u'_r > 0$ are observed at $y/R = 0.215$, and these components are proved to contribute to the negative Reynolds stress. Drag reducing turbulent structures are investigated by means of snapshot POD analysis. The most energetic POD modes show flat periodic structures along the wall, and such structures indicate the relation with these fluctuation peaks and negative Reynolds shear stress.

4:28PM D7.00008 Reynolds number dependence of large-scale friction control in turbulent channel flow, JACOPO CANTON, RAMIS ÖRLÜ, Linné FLOW Centre, KTH Mechanics, CHENG CHIN, NICHOLAS HUTCHINS, JASON MONTY, Department of Mechanical Engineering, University of Melbourne, PHILIPP SCHLATTER, Linné FLOW Centre, KTH Mechanics — The present study reconsiders the control scheme proposed by Schoppa & Hussain (Phys. Fluids 10, 1049–1051 1998), using new direct numerical simulations (DNS). The DNS are performed in a turbulent channel at friction Reynolds number (Re_τ) between 104 (employed value in original study) and 550. The aim is to better characterise the physics of the control, investigate the optimal parameters and Re dependence. The former purpose lead to a re-design of the method: moving from imposing the mean flow to the application of a volume force. Results show that the original method only provides transient drag reduction (DR) but actually increases the drag for longer times. The forcing method, instead, leads to sustained DR, and is therefore superior for all wavelengths investigated. A DR of 18% is obtained at the lowest Re_τ for a viscous-scaled spanwise wavelength of the vortices of 230; the optimal wavelength increases with Re_τ , but the efficiency is reduced, leading to a zero DR for $Re_\tau = 550$, confining the method to low Re for internal flows. Although the findings by Schoppa & Hussain are invalidated, the forcing method is currently implemented in a spatially developing boundary layer to check whether it might lead to a different conclusion in external flows.

4:41PM D7.00009 Turbulent Boundary Layer Drag Reduction by Active Control of Streak Transient Growth¹, FLINT THOMAS, THOMAS CORKE, University of Notre Dame, FAZLE HUSSAIN, Texas Tech University , ALAN DUONG, RYAN MCGOWAN, CHRISOTOPHER JASINSKI, DANIEL SIMMONS, University of Notre Dame — Experiments are reported employing a novel method of large-scale active flow control that was designed to intervene in Streak Transient Growth (STG) which was first postulated by Schoppa and Hussain (Phys. Fluids 1998, JFM 2002) as the primary mechanism in the production of streamwise vorticity in wall-bounded turbulence. We term the actuator SLIPPS ("Smart Longitudinal Instability Prevention via Plasma Surface). It consists of a new pulsed-DC plasma actuator array that is mounted flush with the wall in a zero pressure gradient, high Reynolds number turbulent boundary layer. The array induces a near-wall spanwise mean velocity component that is comparable in magnitude to the local friction velocity. This prevents the lift-up of low-speed streaks, thereby limiting their flanking wall-normal vorticity, which has been shown to be a critical parameter is STG. Experiments demonstrate friction drag reduction of up to 68%. Measured drag reduction is found to scale with the actuator array spanwise inter-electrode spacing, with the maximum drag reduction corresponding to the simultaneous control of approximately 8-10 low-speed streaks. Due to the unique voltage-current characteristics of the pulsed-DC actuator, the drag reduction is obtained with net power savings.

¹Supported by NASA Langley under NNX15CL65P

4:54PM D7.00010 A relation between velocity-vorticity correlations and skin friction in wall-bounded turbulent flows¹, MIN YOON, JUNSUN AHN, JINYUL HWANG, HYUNG JIN SUNG², KAIST — The relationship between the skin friction and the velocity-vorticity correlations in wall-bounded turbulent flows is derived from the mean vorticity equation. A formula for the skin friction coefficient (C_f) is proposed and evaluated with regards to three canonical wall-bounded flows: turbulent boundary layer, turbulent channel flow, and turbulent pipe flow. The skin friction coefficient can be derived from the mean spanwise vorticity at the wall. Double integration with respect to the wall-normal direction (from 0 to y) is needed to derive C_f from the second derivative of the mean spanwise vorticity in the mean spanwise vorticity equation. One more integration is needed to find the contribution of each component to C_f from the wall to the boundary layer edge (from 0 to δ). The present formula encompasses four terms: advective vorticity transport, vortex stretching, viscous, and inhomogeneous terms. Drag-reduced channel flow with the slip condition is used to test the reliability of the formula. The advective vorticity transport and vortex stretching terms are found to dominate the contributions to the frictional drag.

¹This work was supported by the Creative Research Initiatives (No. 2016-004749) program of the National Research Foundation of Korea (MSIP).

²Correspondence to Hyung Jin Sung

Sunday, November 20, 2016 2:57PM - 5:07PM –
Session D8 Nonlinear Dynamics: Topology & Theoretical B116 - Jeffrey Tithof, University of Rochester

2:57PM D8.00001 Characterizing Mixing in a Quasi-Two-Dimensional Flow using Persistent Homology, JEFFREY TITHOF, DOUGLAS KELLEY, University of Rochester — Fluid mixing is a tremendously important phenomenon present in numerous physical systems, both natural and human-made. Describing, understanding, and predicting the mixing behavior of fluid flows poses an immense challenge. In this work, we explore the utility of topological data analysis in quantifying fluid mixing. We analyze Eulerian and Lagrangian quantities obtained from a quasi-two-dimensional flow realized by driving a thin layer of fluid with electromagnetic forces. Our analysis employs persistent homology, which offers a unique framework for quantifying topological features associated with connectivity in the fluid flow. Preliminary results suggest that this topological approach offers new physical insight, complementing existing methods for quantifying fluid mixing.

3:10PM D8.00002 Topological Chaos in a Three-Dimensional Spherical Vortex, SPENCER SMITH, Mt Holyoke College, JOSHUA ARENSON, KEVIN MITCHELL, University of California Merced — Topological techniques have proven to be powerful tools for characterizing the complexity of advection in many 2D fluid flows. However, the path to extending many techniques to three dimensions is filled with roadblocks, which prevent their application to a wider variety of interesting flows. We successfully extend the homotopic lobe dynamics (HLD) technique, previously developed for 2D area-preserving flows, to 3D volume-preserving flows. Specifically, we use intersecting two-dimensional stable and unstable invariant manifolds to construct a symbolic representation of the topological dynamics. This symbolic representation can be used to classify the trajectories of passively advected particles and to compute mixing measures, such as the topological entropy. In this talk, we apply the 3D HLD technique to an explicit numerical example: a time-periodic perturbation of Hill's spherical vortex, modified to break both rotational symmetry and integrability. For this system, the 3D HLD technique is able to detect a distinction between the topologically forced 2D stretching rate of material surfaces and the 1D stretching rate of material curves, illustrating the truly 3D nature of our approach.

3:23PM D8.00003 Transition to turbulence: highway through the edge of chaos is charted by Koopman modes, T. S. EAVES, DAMTP, University of Cambridge, C. P. CAULFIELD, BPI & DAMTP, University of Cambridge, I. MEZIC, Mechanical Engineering, UCSB — We present evidence of low-dimensional dynamical state-space structures enabling transition to turbulence using an extension of the recently advanced operator-theoretic approach to turbulence of Mezi (2005). To do this, we use the dynamic-mode-decomposition (DMD) algorithm of Schmid & Sesterhenn (2008) on the minimal seed trajectories in plane Couette flow of Rabin et al. (2012) and Eaves & Caulfield (2015), which transition to turbulence via the most energy-efficient finite amplitude perturbation from the laminar state. The methodology enables identification of low dimensional structures associated with stable and unstable manifolds of exact solutions to the Navier-Stokes equations, even though the state space is very high-dimensional. In consequence, the results provide a low-dimensional representation of the transition to turbulence and also identify the first known dynamical signature of the importance of edge states in this transition.

3:36PM D8.00004 Using Persistent Homology to Identify Localised Defects in Rayleigh Bénard Convection¹, BALACHANDRA SURI, JEFFREY TITHOF, MICHAEL SCHATZ, Georgia Institute of Technology, RACHEL LEVANGER, JACEK CYRANKA, KONSTANTIN MISCHAIKOW, Rutgers University, MU XU, MARK PAUL, Virginia Institute of Technology, MIROSLAV KRAMAR, AIMR Tohoku University — Complex spatiotemporal convective roll patterns are observed when a sufficiently large temperature gradient is created across a thin layer of fluid. These roll patterns are often characterized by the presence of localised defects such as centers, spirals, disclinations, grain boundaries, which play a crucial dynamical role. Our research focuses on using persistent homology (a branch of algebraic topology) to identify these defects in an experimental realization of the Rayleigh Bénard convection in a cylindrical container. Persistent homology provides a powerful mathematical formalism in which the topological characteristics of a pattern (shadowgraph image in our case) are encoded in a so-called persistence diagram. By identifying several instants in the experiment that correspond to the appearance of a certain type of defect and computing the persistence diagrams for the corresponding shadowgraph images, we extract signatures in the persistence diagram which characterize the defect. Then, for a spatiotemporally resolved series of shadowgraph images we show that using signatures from the persistence diagrams one can automate identifying the instants when localized defects appear.

¹NSF Grants DMS-1125302, CMMI-234436

3:49PM D8.00005 An effective diffusivity model based on Koopman mode decomposition, HASSAN ARBABI, IGOR MEZIC, UC Santa Barbara — In the previous work, we had shown that the Koopman mode decomposition (KMD) can be used to analyze mixing of passive tracers in time-dependent flows. In this talk, we discuss the extension of this type of analysis to the case of advection-diffusion transport for passive scalar fields. Application of KMD to flows with complex time-dependence yields a decomposition of the flow into mean, periodic and chaotic components. We briefly discuss the computation of these components using a combination of harmonic averaging and Discrete Fourier Transform. We propose a new effective diffusivity model in which the advection is dominated by mean and periodic components whereas the effect of chaotic motion is absorbed into an effective diffusivity tensor. The performance of this model is investigated in the case of lid-driven cavity flow.

4:02PM D8.00006 Could time itself be logarithmic?, WILLIAM GEORGE, Department of Aeronautics, Imperial College, SW7 2AZ London UK — This presentation hypothesizes that increments of time may be logarithmic and measured from an initial instant – the log of absolute time if you will. In this alternative view all equations involving time must be written with $\ln t/t_o$ where t is measured in linear increments from the beginning of the universe and t_o is the universal time scale. All equations involving time derivatives must be written not as d/dt but $d/d\ln t/t_o = \ln t/t_o$. An immediate consequence, for example, is that our definition of mass in Newton's Law must change as well: from $mdv/dt = F$ to $m_*dv/d\ln t/t_o = m_*tdv/dt = F$ where F is force applied and v is velocity (however defined). $m_* = m/t$ is the 'true' or absolute mass. Since we have been measuring for only about 500 years and the universe is estimated to be about 18 billion years (millions of billions of seconds) old, the differences are impossible to measure; i.e., $\ln(t + \delta t) - \ln t \approx \delta t/t$. It is only when we look backwards towards the beginning of the universe that we notice the difference – mass, $m = m_*t$, appears to be missing. So we need "dark matter" to make our equations balance – when in fact it might be our "linear-time" equations and definitions that are wrong.

4:15PM D8.00007 Metriplectic simulated annealing for quasigeostrophic flow¹, 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 embodies the first and second laws of thermodynamics, energy conservation and entropy production. The formalism provides an H -theorem for relaxation to nontrivial equilibrium states. Upon choosing enstrophy as entropy and potential vorticity of the form $q = \nabla^2\psi + T(x)$, recent results of computations, akin to those of [3], will be described for various topography functions $T(x)$, including ridge ($T = \exp(-x^2/2)$) and random functions. Interpretation of the results, in particular their sensitivity to the chosen entropy function will be discussed.

[1] P.J. Morrison, Physica D **18**, 410 (1986).

[2] A.M. Bloch, P.J. Morrison, and T.S. Ratiu, in Recent Trends in Dynamical Systems **35**, 371 (2013).

[3] G.R. Flierl and P.J. Morrison, Physica D **240**, 212 (2011).

¹PJM supported by U.S. Dept. of Energy Contract # DE-FG05-80ET-53088.

4:28PM D8.00008 Data-driven discovery of partial differential equations, SAMUEL RUDY, Department of Applied Mathematics, University of Washington, Seattle, STEVEN BRUNTON, Department of Mechanical Engineering, University of Washington, Seattle, JOSHUA PROCTOR, Institute for Disease Modeling, J. NATHAN KUTZ, Department of Applied Mathematics, University of Washington, Seattle — Fluid dynamics is inherently governed by spatial-temporal interactions which can be characterized by partial differential equations (PDEs). Emerging sensor and measurement technologies allowing for rich, time-series data collection motivate new data-driven methods for discovering governing equations. We present a novel computational technique for discovering governing PDEs from time series measurements. A library of candidate terms for the PDE including nonlinearities and partial derivatives is computed and sparse regression is then used to identify a subset which accurately reflects the measured dynamics. Measurements may be taken either in a Eulerian framework to discover field equations or in a Lagrangian framework to study a single stochastic trajectory. The method is shown to be robust, efficient, and to work on a variety of canonical equations. Data collected from a simulation of a flow field around a cylinder is used to accurately identify the Navier-Stokes vorticity equation and the Reynolds number to within 1%. A single trace of Brownian motion is also used to identify the diffusion equation. Our method provides a novel approach towards data enabled science where spatial-temporal information bolsters classical machine learning techniques to identify physical laws.

4:41PM D8.00009 Approximate Solutions to the Linearized Navier-Stokes Equations, ANTHONY LEONARD, California Institute of Technology — The linearized Navier-Stokes equations for incompressible channel flow are considered in which the flow is homogeneous in two directions. We study the initial-value problem for v and ω_y , where y is the coordinate normal to the wall. After a Laplace transform in time and a double Fourier transform in space we use the WKB approximation on the resulting system of ODEs in y . For example, for the inviscid equations we can construct analytically the Greens function for such solutions in terms of the Bessel functions $J_{+1/3}, J_{-1/3}, J_1$, and Y_1 and their modified counterparts. In this approach the critical layer or the y location where $U(y) = \omega/k_x$ requires special attention, as might be expected, as well as the location of the turning point where $d^2U/dy^2 = (k_x^2 + k_z^2)(\omega/k_x - U(y))$, if it exists.

4:54PM D8.00010 Test-filter scale effects on spectral energy transfer in direct numerical simulations of stratified turbulence, SINA KHANI, Princeton University, MICHAEL WAITE, University of Waterloo — The spectral kinetic and potential energy transfers around a test-filter scale will be presented in this talk. We use direct numerical simulations of stratified turbulence and study the up- and downscale energy transfers when different test-filter scales are applied. Our results suggest that the spectral energy transfer depends on the buoyancy Reynolds number Re_b and test-filter scale Δ_{test} . In particular, an up-scale energy transfer (i.e. backscatter) from sub-filter scales to intermediate scales are seen when Δ_{test} is around the dissipation scale L_d . However, we find that this spectral backscatter is due to viscous effects and not a turbulent mechanisms of stratified turbulence. In addition, our results demonstrate that effective turbulent Prandtl number spectra show constant values around $Pr_t \approx 1$ for the local energy transfer or when the buoyancy Reynolds number is large.

Sunday, November 20, 2016 2:57PM - 4:28PM –
Session D9 Fluid Dynamics - Education, Outreach, and Diversity I B117 - Frank Jacobitz,
University of San Diego

2:57PM D9.00001 Seeing Fluid Physics via Visual Expertise Training, JEAN HERTZBERG, University of Colorado Boulder, KATHERINE GOODMAN, University of Colorado Denver, TIM CURRAN, University of Colorado Boulder — In a course on Flow Visualization, students often expressed that their perception of fluid flows had increased, implying the acquisition of a type of visual expertise, akin to that of radiologists or dog show judges. In the first steps towards measuring this expertise, we emulated an experimental design from psychology. The study had two groups of participants: "novices" with no formal fluids education, and "experts" who had passed at least one fluid mechanics course. All participants were trained to place static images of fluid flows into two categories (laminar and turbulent). Half the participants were trained on flow images with a specific format (Von Kármán vortex streets), and the other half on a broader group. Novices' results were in line with past perceptual expertise studies, showing that it is easier to transfer learning from a broad category to a new specific format than vice versa. In contrast, experts did not have a significant difference between training conditions, suggesting the experts did not undergo the same learning process as the novices. We theorize that expert subjects were able to access their conceptual knowledge about fluids to perform this new, visual task. This finding supports new ways of understanding conceptual learning.

3:10PM D9.00002 Woooooahhh! vs Aha!, is the choice obvious?, FARAZ MEHDI, Hypertherm Inc. — There has been a lot of focus towards attracting people, especially under-represented groups, to STEM fields. One of the ways to accomplish this is short demonstrations and workshops, where young students are exposed to "exciting" experiments in an effort to make STEM more appealing. We tried an alternative approach by making the students perform a deliberately "boring" experiment but one which made them think scientifically. This was tested on a small group of high school students during Girls Technology Day in New Hampshire.

3:23PM D9.00003 F* Yeah Fluid Dynamics: Inside the science communication process**, NICOLE SHARP, FYFD — Communicating scientific research to general audiences may seem daunting, but it does not have to be. For six years, fluid dynamics outreach blog FYFD has been sharing the community's scientific output with an audience of nearly a quarter of a million readers and viewers of all ages and backgrounds. This talk will focus on the process behind science communication and some of the steps and exercises that can help scientists communicate to broad audiences more effectively. Using examples from the FYFD blog and YouTube channel, the talk will illustrate this communication process in action.

3:36PM D9.00004 The Fluids RAP, IVAYLO NEDYALKOV, University of New Hampshire — After fifteen years of experience in rap, and ten in fluid mechanics, "I am coming here with high-Reynolds-number stamina; I can beat these rap folks whose flows are... laminar." The rap relates fluid flows to rap flows. The fluid concepts presented in the song have varying complexity and the listeners/viewers will be encouraged to read the explanations on a site dedicated to the rap. The music video will provide an opportunity to share high-quality fluid visualizations with a general audience. This talk will present the rap lyrics, the vision for the video, and the strategy for outreach. Suggestions and comments will be welcomed.

3:49PM D9.00005 Attracting Students to Fluid Mechanics with Coffee, WILLIAM RISTENPART, Dept. Chemical Engineering, University of California Davis — We describe a new class developed at U.C. Davis titled "The Design of Coffee," which serves as a nonmathematical introduction to chemical engineering as illustrated by the process of roasting and brewing coffee. Hands-on coffee experiments demonstrate key engineering principles, including material balances, chemical kinetics, mass transfer, conservation of energy, and fluid mechanics. The experiments lead to an engineering design competition where students strive to make the best tasting coffee using the least amount of energy - a classic engineering optimization problem, but one that is both fun and tasty. "The Design of Coffee" started as a freshmen seminar in 2013, and it has exploded in popularity: it now serves 1,533 students per year, and is the largest and most popular elective course at U.C. Davis. In this talk we focus on the class pedagogy as applied to fluid mechanics, with an emphasis on how coffee serves as an engaging and exciting topic for teaching students about fluid mechanics in an approachable, hands-on manner.

4:02PM D9.00006 Creative Turbulence: Experiments in Art and Physics¹, ENRICO FONDA, R. LUKE DUBOIS, SARA CAMNASIO, MAURIZIO PORFIRI, KATEPALLI R. SREENIVASAN, New York University, DANIEL P. LATHROP, DANIEL SERRANO, University of Maryland, College Park, DEVESH RANJAN, Georgia Institute of Technology — Effective communication of basic research to non-experts is necessary to inspire the public and to justify support for science by the taxpayers. The creative power of art is particularly important to engage an adult audience, who otherwise might not be receptive to standard didactic material. Interdisciplinarity defines new trends in research, and works at the intersection of art and science are growing in popularity, even though they are often isolated experiments. We present a public-facing collaboration between physicists/engineers performing research in fluid dynamics, and audiovisual artists working in cutting-edge media installation and performance. The result of this collaboration is a curated exhibition, with supporting public programming. We present the artworks, the lesson learned from the interactions between artists and scientists, the potential outreach impact and future developments.

¹This project is supported by the APS Public Outreach Mini Grant.

4:15PM D9.00007 The making of a cavitation childrens book¹ , MARC HENRY DE FRAHAN,

BRANDON PATTERSON², ERIKA LAZAR, Univ of Michigan - Ann Arbor — Engaging young children in science is particularly important to future scientific endeavors. From thunderstorms to the waterpark, children are constantly exposed to the wonders of fluid dynamics. Among fluid phenomena, bubbles have always fascinated children. Yet some of the most exciting aspects of bubbles, such as cavitation, are scarcely known to non-experts. To introduce cavitation to a five year old audience, we wrote Brooke Bubble Breaks Things, a childrens book about the adventures of a cavitation bubble learning about all the things she could break. In this talk, we discuss how a childrens book is made by walking through the steps involved in creating the book from concept to publication. We focus on strategies for successfully communicating a technical message while balancing entertainment and fidelity to nature. To provide parents, teachers, and young inquiring minds with a detailed explanation of the physics and applications of cavitation, we also created a website with detailed explanations, animations, and links to further information. We aim to convince the fluids community that writing picture books is an intellectually stimulating and fun way of communicating fluids principles and applications to children.

¹ArtsEngine Microgrant at the University of Michigan

²All three authors contributed equally to this work.

Sunday, November 20, 2016 2:57PM - 5:07PM –

Session D10 DFD GPC: Convection and Buoyancy Driven Flows: General B118-119 -

Herman Clercx, Technische Universiteit Eindhoven

2:57PM D10.00001 Rotating thermal convection at very large Rayleigh numbers¹ ,

STEPHAN WEISS, Max Planck Institute f. Dynamics and Self-Organisation, DENNIS VAN GILS, University of Twente, GUENTER AHLERS, University of California, Santa Barbara, EBERHARD BODENSCHATZ, Max Planck Institute f. Dynamics and Self-Organisation — The large scale thermal convection systems in geo- and astrophysics are usually influenced by Coriolis forces caused by the rotation of their celestial bodies. To better understand the influence of rotation on the convective flow field and the heat transport at these conditions, we study Rayleigh-Bénard convection, using pressurized sulfur hexafluoride (SF_6) at up to 19 bars in a cylinder of diameter $D=1.12\text{ m}$ and a height of $L=2.24\text{ m}$. The gas is heated from below and cooled from above and the convection cell sits on a rotating table inside a large pressure vessel (the "Uboot of Göttingen"). With this setup Rayleigh numbers of up to $Ra = 10^{15}$ can be reached, while Ekman numbers as low as $Ek = 10^{-8}$ are possible. The Prandtl number in these experiment is kept constant at $Pr \simeq 0.8$. We report on heat flux measurements (expressed by the Nusselt number Nu) as well as measurements from more than 150 temperature probes inside the flow.

¹We thank the Deutsche Forschungsgemeinschaft (DFG) for financial support through SFB963: Astrophysical Flow Instabilities and Turbulence. The work of GA was supported in part by the US National Science Foundation through Grant DMR11-58514.

3:10PM D10.00002 An experimental study on columnar vortex structures in rotating

Rayleigh-Bénard convection , YUJI TASAKA, KODAI FUJITA, YUICHI MURAI, Hokkaido Univ., TAKATOSHI YANAGI-SAWA, JAMSTEC — A scanning PIV system was developed to investigate columnar vortex structures in rotating Rayleigh-Bénard convection in a range of Taylor number, $6.0 \times 10^6 \leq Ta \leq 1.0 \times 10^8$, at constant Rayleigh number, $Ra = 1.0 \times 10^7$. Horizontal vortex advection that is much slower than the vertical scanning motion by a motor driven stage of a laser light sheet allows capturing quasi-instantaneous 3D vortex structures. Vortex distributions at each scanning plane were represented by contour of stream function calculated from a planar velocity vector field measured by PIV with assuming quasi-two dimensional flow field at the planes. 3D structure at each Ta number was visualized by iso-surface of the stream function and the vertical velocity component was estimated from the planar velocity fields via equation of continuity for incompressible fluids. These results suggested that the flow transportation is emphasized by straightening of the columnar vortices with increasing Ta . This may correspond to improvement of Nusselt number with background rotation at the present range of Ta .

3:23PM D10.00003 Thermal boundary layer measurements in turbulent rotating Rayleigh-Bénard convection¹ , JIN-QIANG ZHONG, HUI-MIN LI, WEN-DAN YAN, Tongji University — We report measurements of the temperature boundary layer (BL) profile $T(z)$ in rotating Rayleigh-Bénard convection along the centerline of a cylindrical sample. The measurements are taken for two Prandtl numbers $Pr=6$ and 40 , with Rayleigh numbers in the range $5.8 \times 10^8 \leq Ra \leq 6.3 \times 10^9$ and inversed Rossby numbers $0 \leq 1/Ro \leq 8$. In this parameter range pronounced enhancement of the heat transport (up to 22% in Nu) is observed. Measurements of $T(z)$ reveal two regimes in which the effects of the applied rotations on the BL profiles differ markedly. In a low- Ra and high- Pr regime, the thermal BL thickness λ decreases with increasing $1/Ro$ in accord with the global Nu -enhancement. In the other regime with relatively high $Ra > 2.2 \times 10^9$ and low Pr , however, λ is found to be independent of $1/Ro$. The root-mean-square temperature profiles $\sigma(z)$ exhibit power-low dependence $\sigma(z) \propto z^{\beta(Ro)}$ when $z > \lambda$. The exponent $\beta(Ro)$ is -0.6 ± 0.05 when $1/Ro=0$, but increases with increasing $1/Ro$ and is asymptotic to $\beta(\infty)=-0.2 \pm 0.02$ at large $1/Ro$, indicating temperature fluctuation remains dominant in a broader domain outside the thermal BL that is ascribed to the Ekman pumping effect.

¹Supported by the NSF of China through grant no. 11572230 and 1561161004.

3:36PM D10.00004 Flow anisotropy in rotating buoyancy-driven turbulence , HADI

RAJAEI, KIM ALARDS, RUDIE KUNNEN, FEDERICO TOSCHI, HERMAN CLERCX, Eindhoven Univ of Tech — We report a combined experimental-numerical study of the effects of background rotation on large- and small-scale isotropy in rotating Rayleigh-Bénard convection (RBC) from both Eulerian and Lagrangian points of view. 3D particle tracking velocimetry (3D-PTV) and direct numerical simulations (DNS) are employed at three different heights within the cylindrical cell. The Lagrangian velocity fluctuation is utilized to evaluate the large-scale isotropy for different rotation rates. Furthermore, we examine the experimental measurements of the Lagrangian acceleration of neutrally buoyant particles and the second-order Eulerian structure function to evaluate the small-scale isotropy as a function of rotation rate. It is found that background rotation enhances large-scale anisotropy at the cell center and close to the top plate, while decreases it at intermediate height. The large-scale anisotropy, induced by rotation, has negligible effect on the small scales at the cell center, whereas the small scales remain anisotropic close to the top plate.

3:49PM D10.00005 Geometry of tracer trajectories in turbulent rotating convection

, KIM ALARDS, HADI RAJAEI, RUDIE KUNNEN, FEDERICO TOSCHI, HERMAN CLERCX, Eindhoven University of Technology — In Rayleigh-Bénard convection rotation is known to cause transitions in flow structures and to change the level of anisotropy close to the horizontal plates. To analyze this effect of rotation, we collect curvature and torsion statistics of passive tracer trajectories in rotating Rayleigh-Bénard convection, using both experiments and direct numerical simulations. In previous studies, focusing on homogeneous isotropic turbulence (HIT), curvature and torsion PDFs are found to reveal pronounced power laws. In the center of the convection cell, where the flow is closest to HIT, we recover these power laws, regardless of the rotation rate. However, near the top plate, where we expect the flow to be anisotropic, the scaling of the PDFs deviates from the HIT prediction for lower rotation rates. This indicates that anisotropy clearly affects the geometry of tracer trajectories. Another effect of rotation is observed as a shift of curvature and torsion PDFs towards higher values. We expect this shift to be related to the length scale of typical flow structures. Using curvature and torsion statistics, we can characterize how these typical length scales evolve under rotation and moreover analyze the effect of rotation on more complicated flow characteristics, such as anisotropy.

the National Research Foundation of Korea (NRF) funded by the Korean government (MEST)

4:02PM D10.00006 Study on analysis of ionic wind for heat transfer

HAN SEO KO, DONG HO SHIN, Sungkyunkwan University — Local heat transfer technology was studied in this study. Characteristics of ionic wind using wire and plate electrodes were studied by experimental velocimetry (PIV) test was conducted for a study of a boundary layer controlled by the ionic wind. It was found that the coulombic force consistently acted on the surface to reduce the effect of the viscous force formed on the heated surface and controlled by the ionic wind regardless of the Reynolds number of the increased and decreased, 11% and 19% in average on the heated surface by the ionic wind, for the (2500~3500) Reynolds numbers of the bulk flow, respectively. It was concluded that the ionic wind has a great influence on the heat transfer rate or insulating the local surface according to its operating condition. The results of the ionic wind were applied for the heat exchanger and the performance was confirmed by the experiments.

(No. 2013R1A2A2A01068653).

4:15PM D10.00007 Interfacial condensation induced by sub-cooled liquid jet

, ENRIQUE RAME, USRA, R. BALASUBRAMANIAM, Case Western Reserve University — When a sub-cooled liquid jet impinges on the free surface between a liquid and its vapor, vapor will condense at a rate dependent on the sub-cooling, the jet strength and fluid properties. In 1966 and during the examination of a different type of condensation flow, Shekrladeze found an approximate result, valid at large condensation rates, that decouples the flow in the liquid phase from that of the vapor, without putting it in the context of a formal asymptotic approximation. In this talk we will develop an asymptotic approximation that contains Shekrladze's result, and extend the calculations to the case when a non-condensable gas is present in the vapor phase.

4:28PM D10.00008 On the thermal convection in a viscoelastic Jeffreys fluid layer heated from below confined between walls of finite thickness and thermal conductivity.¹

, ILDEBRANDO PREZ-REYES, Universidad Autnoma de Chihuahua , REN O. VARGAS-AGUILAR, Instituto Politcnico Nacional — The thickness and thermal conductivity of the bounding walls are of interest in the hydrodynamic stability of a viscoelastic fluid layer. In this work the linear hydrodynamic stability is studied by means of the Galerkin method. The two ideal cases of thermal insulating and perfect thermal conducting walls are bridged by taking into account these two properties. Curves of criticality for the Rayleigh number, the wavenumber and the frequency of oscillation against the thermal conductivity for fixed wall thickness, Prandtl number and relaxation and retardation times are presented. Here, the dimensionless retardation time E was set to 0.05 and 0.1 while the dimensionless relaxation time F was set to 0.1 and 100. The role of the thermal conductivity and of the thickness of the walls are discussed. One important result of this investigation is that for non ideal thermal conducting conditions the system is more stable when the thickness of the fluid layer is larger in comparison to that of the boundaries. A discussion on the effect of E and F on the stability is given as well.

¹Ciencia Bsica - Conacyt through project No. 255839

4:41PM D10.00009 Thermal Convection on an Irradiated Target¹

, IGBAL MEHMEDAGIC, U. S. Army ARDEC, Picatinny Arsenal, NJ, SIVA THANGAM, Stevens Institute of Technology, Castle Point, Hoboken, NJ — The present work involves the computational modeling of metallic targets subject to steady and high intensity heat flux. The ablation and associated fluid dynamics when metallic surfaces are exposed to high intensity laser fluence at normal atmospheric conditions is modelled. The incident energy from the laser is partly absorbed and partly reflected by the surface during ablation and subsequent vaporization of the melt. 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. 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 intensity. The relevance of the findings to various manufacturing processes as well as for the development of protective shields is discussed.

¹Funded in part by U. S. Army ARDEC, Picatinny Arsenal, NJ

4:54PM D10.00010 CFD Validation Benchmark Dataset for Natural Convection in Nuclear Fuel Rod Bundles¹

, BARTON SMITH, KYLE JONES, Utah State University — The present study provide CFD validation benchmark data for coupled fluid flow/convection heat transfer on the exterior of heated rods arranged in a 2×2 array. The rod model incorporates grids with swirling veins to resemble a nuclear fuel bundle. The four heated aluminum rods are suspended in an open-circuit wind tunnel. Boundary conditions (BCs) are measured and uncertainties calculated to provide all quantities necessary to successfully conduct a CFD validation exercise. System response quantities (SRQs) are measured for comparing the simulation output to the experiment. Stereoscopic Particle Image Velocimetry (SPIV) is used to non-intrusively measure 3-component velocity fields. A through-plane measurement is used for the inflow while laser sheet planes aligned with the flow direction at several downstream locations are used for system response quantities. Two constant heat flux rod surface conditions are presented (400 W/m^2 and 700 W/m^2) achieving a peak Rayleigh number of $\sim 10^{10}$. Uncertainty for all measured variables is reported. The boundary conditions, system response, and all material properties are now available online for download.

¹The U.S. Department of Energy Nuclear Engineering University Program provided the funding for these experiments under grant 00128493

Sunday, November 20, 2016 2:57PM - 5:07PM –

Session D11 Electrokinetic Flows: General C120-121-122 - Alexander Yarin, University of Illinois-Chicago

2:57PM D11.00001 Electrostatic atomization: Effect of electrode materials on electrostatic atomizer performance¹, ABHILASH SANKARAN, CHRISTOPHER STASZEL, BABAK KASHIR, ANTHONY PERRI, FARZAD MASHAYEK, ALEXANDER YARIN, Univ of Illinois - Chicago — Electrostatic atomization was studied experimentally with a pointed electrode in a converging nozzle. Experiments were carried out on poorly conductive canola oil where it was observed that electrode material may affect charge transfer. This points at the possible faradaic reactions that can occur at the surfaces of the electrodes. The supply voltage is applied to the sharp electrode and the grounded nozzle body constitutes the counter-electrode. The charge transfer is controlled by the electrochemical reactions on both the electrodes. The electrical performance study of the atomizer issuing a charged oil jet was conducted using three different nozzle body materials – brass, copper and stainless steel. Also, two sharp electrode materials – brass and stainless steel – were tested. The experimental results revealed that both the nozzle body material, as well as the sharp electrode material affected the spray and leak currents. Moreover, the effect of the sharp electrode material is quite significant.

¹This research is supported by NSF grant 1505276.

3:10PM D11.00002 Confinement effect on liquid and ion transport in nanochannels coated with environmental-stimuli-responsive polyelectrolyte(PE) brushes, GUANG CHEN, SIDDHARTH DAS, The Department of Mechanical Engineering, University of Maryland — We study the confinement effect in the electrokinetic transport in polyelectrolyte(PE)-brush-grafted nanochannels. Starting with thermodynamically self-consistent description, i.e., accounting for the elastic, excluded volume and electrostatic effects of the PE brush and the effects of the induced electric double layer, we first probe the equilibrium brush height. We show that this height is dictated by PE size, grafting density, concentration of electrolyte solution and the extent of confinement. Shrinking-swelling behavior of PE brush with various configurations are compared: 1) short sparse end-charged PE brush swells as the salt concentration increases, while long dense end-charged PE brush shrinks; 2) PE brush with constant volume charge along the backbone always shrinks with the increase of the salt concentration. This shrinking-swelling behavior as well as the monomer distribution of PE interplay with the PE-induced drag force to dictate the overall electroosmotic and ionic current transport in such PE-brush-grafted nanochannels. We exhibit that among other factors, height of the nanochannels can be tuned to regulate this transport. We anticipate that our study will shed new light on structure of nano confined PE brushes with implications in ionic current rectifier design.

3:23PM D11.00003 Unsteady Capillary Filling By Electrocapillarity¹, IN SEOK KANG, JUNG A LEE, Dept of Chem Eng, POSTECH, Pohang — Unsteady filling of electrolyte solution inside a nanochannel by the electrocapillarity effect is studied. The filling rate is predicted as a function of the bulk concentration of the electrolyte, the surface potential (or surface charge density), and the cross sectional shape of the channel. Since the driving force of the flow is the electrocapillarity, it is first analyzed by using the solution of the Poisson-Boltzmann equation. From the analysis, it is found that the results for many different cross sectional shapes can be unified with good accuracy if the hydraulic radius is adopted as the characteristic length scale of the problem. Especially in the case of constant surface potential, for both limits of $\kappa h \rightarrow 0$ and $\kappa h \rightarrow \infty$, it can be shown theoretically that the electrocapillarity is independent of the cross sectional shape if the hydraulic radius is the same. In order to analyze the geometric effects more systematically, we consider the regular N -polygons with the same hydraulic radius and the rectangles of different aspect ratios. Washburn's approach is then adopted to predict the filling rate of electrolyte solution inside a nanochannel. It is found that the average filling velocity decreases as N increases in the case of regular N -polygons with the same hydraulic radius. This is because of that the regular N -polygons of the same hydraulic radius share the same inscribing circle. .

¹This work has been supported by BK21+ program.

3:36PM D11.00004 Sample Preconcentration in Nanochannels with Tunable Surface Charge, ALEXANDER EDEN, CHRISTOPHER MCCALLUM, Univ of California - Santa Barbara , BRIAN STOREY, Olin College, CARL MEINHART, SUMITA PENNATHUR, Univ of California - Santa Barbara — We present a novel method for field amplified sample stacking (FASS) and focusing in nanochannels by taking advantage of the nonuniform ion distributions produced by thick electric double layers (EDLs) in channels with heterogeneous surface charge. This is accomplished by applying a voltage bias to a gate electrode embedded within the channel wall, tuning the surface charge in a region of the channel and significantly altering the charge density and ionic strength in that region relative to the rest of the channel. The resulting nonuniform electromigration fluxes in the different regions serve to stack charged sample ions at an interface where a step change in zeta potential occurs, providing enhancement ratios superior to those exhibited in traditional microchannel FASS. Numerical simulations are performed to demonstrate the phenomenon, and resulting velocity and salt concentration profiles show good agreement with analytical results.

3:49PM D11.00005 Membraneless water filtration using CO₂, SANGWOO SHIN, OREST SHARDT, Princeton University, PATRICK WARREN, Unilever RD Port Sunlight, HOWARD STONE, Princeton University — Water purification technologies such as ultrafiltration and reverse osmosis utilize porous membranes to remove suspended particles and solutes. These membranes, however, cause many drawbacks such as a high pumping cost and a need for periodic replacement due to fouling. Here we show an alternative membraneless method for separating suspended particles by exposing the colloidal suspension to CO₂. Dissolution of CO₂ into the suspension creates solute gradients that drive phoretic motion of particles, or so-called diffusiophoresis. Due to the large diffusion potential built up by the dissociation of carbonic acid, colloidal particles move either away from or towards the gas-liquid interface depending on their surface charge. Our findings suggest a means to separate particles without membranes or filters, thus reducing operating and maintenance costs. Using the directed motion of particles induced by exposure to CO₂, we demonstrate a scalable, continuous flow, membraneless particle filtration process that exhibits very low pressure drop and is essentially free from fouling.

4:02PM D11.00006 Diffusiophoresis of a charged drop, FAN YANG, SANGWOO SHIN, HOWARD STONE, Princeton Univ — Diffusiophoresis describes the motion of colloids in an electrolytic solution under a concentration gradient. Most of the previous studies in diffusiophoresis have dealt with motion of rigid particles. Here, we study the diffusiophoresis of fluid particles analytically and experimentally. We obtain the analytical solution of the diffusiophoretic velocity of fluid particles by perturbation methods. Using charged oil droplets, we measure the droplet speed under solute concentration gradient and compare it with the analytical solution. Our findings have potential applications for oil recovery and drug delivery.

4:15PM D11.00007 The Influence of Soft Layer Electrokinetics on Electroporation of Gram-positive Bacteria, NAGA NEEHAR DINGARI, JEFFREY L. MORAN, PAULO A. GARCIA, CULLEN R. BUIE, Massachusetts Inst of Tech-MIT — Bacterial electroporation involves subjecting cells to intense (~10 kV/cm) electric pulses, to open pores on the cell membrane for intracellular delivery of exogenous molecules. Its high efficiency in genetic transformation makes it an attractive tool for synthetic biology. While mammalian cell electroporation has received extensive theoretical and experimental investigation, bacterial electroporation has received markedly less attention. In this work, we develop a theoretical model of electroporation for gram-positive bacteria, taking into account the effect of the bacterial cell envelope on the cell's response to an electroporation pulse. We model the influence of the cell wall charge on the electrokinetic transport (and hence the pore properties) around the bacterial cell envelope using the Poisson-Nernst-Planck equations. Further, we account for the influence of the cell wall's mechanical elasticity on the pore radius evolution during electroporation, which is typically neglected in mammalian cell electroporation. This yields valuable information about favorable conditions for pore formation and will enable designing optimal platforms for bacteria electroporation.

4:28PM D11.00008 Electro-hydrodynamic force field and flow patterns generated by a DC corona discharge in the air.¹, NICOLAS MONROLIN, FRANCK PLOURABOUE, OLIVIER PRAUD, None — Ionic wind refers to the electro-convection of ionised air between high voltage electrodes. Microscopic ion-neutral collisions are responsible for momentum transfer from accelerated ions, subjected to the electric field, to the neutral gas molecules resulting in a macroscopic airflow acceleration. In the past decades it has been investigated for various purposes from food drying through aerodynamic flow control and eventually laptop cooling. One consequence of air acceleration between the electrodes is thrust generation, often referred to as the Biefeld-Brown effect or electro-hydrodynamic thrust. In this experimental study, the ionic wind velocity field is measured with the PIV method. From computing the acceleration of the air we work out the electrostatic force field for various electrodes configurations. This enables an original direct evaluation of the force distribution as well as the influence of electrodes shape and position. Thrust computation based on the flow acceleration are compared with digital scale measurements. Complex flow features are highlighted such as vortex shedding, indicating that aerodynamic effects may play a significant role. Furthermore, the aerodynamic drag force exerted on the electrodes is quantified by choosing an appropriate control volume.

¹ Authors thank Region Midi-Pyrenees and CNES Launcher Directorate for financial support.

4:41PM D11.00009 Induced Charge Capacitive Deionization, SHIMON RUBIN, MATTHEW SUSS, Technion - Israel Institute of Technology, MAARTEN BIESHEUVEL, Wetsus, European Centre of Excellence for Sustainable Water Technology, MORAN BERCOVICI, Technion - Israel Institute of Technology — We demonstrate the phenomenon of induced-charge capacitive deionization (ICCDI) that occurs around a porous and conducting particle immersed in an electrolyte, under the action of an external electrostatic field. The external electric field induces an electric dipole in the porous particle, leading to capacitive charging of its volume by both cations and anions at opposite poles. This regime is characterized both by a large RC charging time and a small electrochemical charge relaxation time, which leads to rapid and significant deionization of ionic species from a volume which is on the scale of the particle. We show by theory and experiment that the transient response around a cylindrical particle results in spatially non-uniform charging and non-steady growth of depletion regions which emerge around the particle's poles. Potentially, ICCDI can be useful in applications where fast concentration changes of ionic species are required over large volumes.

4:54PM D11.00010 A perturbative thermal analysis for an electro-osmotic flow in a slit microchannel based on a Lubrication theory, ALI RAMOS, FEDERICO MENDEZ, Univ Nacl Autonoma de Mexico, OSCAR BAUTISTA, Instituto Politcnico Nacional, JOS LIZARDI, Universidad Autonoma de la Ciudad de Mexico — In this work, we develop a new thermal analysis for an electro-osmotic flow in a rectangular microchannel. The central idea is very simple: the Debye length that defines the length of the electrical double-layer depends on temperature T . Therefore, if exists any reason to include variable temperature effects, the above length should be utilized with caution because it appears in any electro-osmotic mathematical model. For instance, the presence of the Joule effect is a source that can generate important longitudinal temperature gradients along the microchannel and the isothermal hypothesis is no longer valid. In this manner, the Debye length is altered and as a consequence, new longitudinal temperature gradient terms appear into the resulting governing equations. These terms are enough to change the electric potential and the flow field. Taking into account the above comments, in the present study the momentum equations together with the energy, Poisson and Ohmic current conservation equations are solved by using a regular perturbation technique. For this purpose, we introduce a dimensionless parameter α that measures the temperature deviations of a reference temperature.

Sunday, November 20, 2016 2:57PM - 5:07PM –
Session D12 Multiphase Flows: Turbulence C123 - Antonino Ferrante, University of Washington

2:57PM D12.00001 On the effects of density ratio on droplet-laden isotropic turbulence, ANTONINO FERRANTE, MICHAEL DODD, University of Washington, Seattle — Our objective is to determine the effects of varying the droplet- to carrier-fluid density ratio (ρ_d/ρ_c) on the interaction of droplets with turbulence. We performed DNS of 3130 finite-size, non-evaporating droplets of diameter approximately equal to the Taylor lengthscale and with 5 % droplet volume fraction in decaying isotropic turbulence at initial Taylor-scale Reynolds number $Re_\lambda = 83$. We varied ρ_d/ρ_c from 1 to 100 while keeping the Weber number and dynamic viscosity ratio constant, $We_{rms}=1$ and $\mu_d/\mu_c=1$. We derived the turbulence kinetic energy (TKE) equations for the two-fluid, carrier-fluid and droplet-fluid flow. These equations allow us to explain the pathways for TKE exchange between the carrier turbulent flow and the flow inside the droplet. We show that increasing ρ_d/ρ_c increases the decay rate of TKE in the two-fluid flow. The TKE budget shows that this increase is caused by an increase in the dissipation rate of TKE and a decrease in the power of the surface tension. The underlying physical mechanisms for these behaviors will be presented.

3:10PM D12.00002 On the effects of isotropic turbulence on the evaporation rate of a liquid droplet, MICHAEL DODD, ANTONINO FERRANTE, University of Washington, Seattle — Our objective is to explain the effects of isotropic turbulence on the vaporization rate of a liquid droplet in conditions that are relevant to spray combustion applications. To this end, we have performed direct numerical simulation (DNS) of a single droplet in homogeneous isotropic turbulence using the volume-of-fluid method for resolving fully the process of momentum, heat, and mass transfer between the liquid droplet and the gas. The simulations were performed using 1024^3 grid points. The effect of turbulence on the droplet vaporization rate is investigated by varying the gas-phase Reynolds number based on the Taylor microscale, Re_λ . Re_λ is increased from 0 to 75 by increasing the r.m.s. velocity of the gas phase while keeping all other physical properties constant. We will present the droplet evaporation rate as a function of turbulence Reynolds number and investigate the physical mechanisms.

3:23PM D12.00003 On the dispersion of finite-size droplets in isotropic turbulence.

, M. ROSSO, H. WANG, S. ELGHOBASHI, University of California, Irvine — The paper presents a comparison between the dispersion characteristics of finite size liquid droplets and finite size solid particles in isotropic turbulence at initial $R_\lambda = 75$. The droplets and particles have equal diameters (about 15 times the initial Kolmogorov length scale) and equal densities. The immersed boundary method is used for direct numerical simulations (DNS) of the solid particles case. The level set method is used for DNS of the droplet-laden turbulence where a variable-density projection method is used to impose the incompressibility constraint. We discuss the effects of varying the surface tension (Weber number) of the liquid droplets on their dispersion and acceleration characteristics.

3:36PM D12.00004 Angular dynamics of a small particle in turbulence , BERNHARD MEHLIG, University of Gothenburg, FABIEN CANDELIER, Universite Aix Marseille, JONAS EINARSSON, University of Gothenburg

— We compute the angular dynamics of a neutrally buoyant nearly spherical particle immersed in an unsteady fluid. We assume that the particle is small, that its translational slip velocity is negligible, and that unsteady and convective inertia are small perturbations. We derive an approximation for the torque on the particle that determines the first inertial corrections to Jeffery's equation. These corrections arise as a consequence of local vortex stretching, and can be substantial in turbulence where local vortex stretching is strong and closely linked to the irreversibility of turbulence.

3:49PM D12.00005 Entrainment at a sediment concentration interface in turbulent channel flow¹ , JORGE SALINAS, MRUGESH SHRINGARPURE, Department of Mechanical and Aerospace Engineering, University of Florida, Gainesville, FL, MARIANO CANTERO, Department of computational mechanics, CNEA, Bariloche, Argentina., S. BALACHANDAR, Department of Mechanical and Aerospace Engineering, University of Florida, Gainesville, FL — In this work we address the role of turbulence on entrainment at a sediment concentration interface. This process can be conceived as the entrainment of sediment-free fluid into the bottom sediment-laden flow, or alternatively, as the entrainment of sediment into the top sediment-free flow. We have performed direct numerical simulations for fixed Reynolds and Schmidt numbers while varying the values of Richardson number and particle settling velocity. The analysis performed shows that the ability of the flow to pick up a given sediment size decreases with the distance from the bottom, and thus only fine enough sediment particles are entrained across the sediment concentration interface. For these cases, the concentration profiles evolve to a final steady state in good agreement with the well-known Rouse profile. The approach towards the Rouse profile happens through a transient self-similar state. Detailed analysis of the three dimensional structure of the sediment concentration interface shows the mechanisms by which sediment particles are lifted up by tongues of sediment-laden fluid with positive correlation between vertical velocity and sediment concentration. Finally, the mixing ability of the flow is addressed by monitoring the center of mass of the sediment-laden layer.

¹With the support of ExxonMobil, NSF, ANPCyT, CONICET.

4:02PM D12.00006 ABSTRACT WITHDRAWN —

4:15PM D12.00007 An efficient parallel flow solver for two-way coupled turbulent flows with deformable bodies , ROBERTO VERZICCO, University of Rome "Tor Vergata", VAMSI SPANDAN, Physics of Fluids, University of Twente, VALENTINA MESCHINI, University of Rome "Tor Vergata", DETLEF LOHSE, Physics of Fluids, University of Twente, MARCO D DE TULLIO, Politecnico di Bari — There are countless examples in Nature and technology in which a flow and a deformable structure interact dynamically and determine each other's behaviour. Among many, two contexts in which this is particularly relevant is in two-phase flows with finite size deformable bubbles or immiscible drops and in cardiovascular flows of heart valves and deformable vessels. Since the standard methods become terminally expensive when the number of deformable bodies become large or the set-up has a complex geometric configuration, in this work, we discuss a simple yet effective approach to cope with the above problems. The main ingredients are: i) an efficient Navier-Stokes solver, ii) an interaction potential approach for the dynamics of a deformable structure, iii) an immersed boundary procedure to deal with the geometrical complexity iv) a set of fluid/structure interaction approaches (strong or loose) and v) a simple and efficient parallelisation strategy to handle large-scale simulations. Several complex examples will be shown and discussed with the results validated either by ad-hoc experiments or by comparisons with results from the literature.

4:28PM D12.00008 Understanding the link between deformability and drag reduction in sheared turbulent flows , VAMSI SPANDAN, Physics of Fluids, University of Twente, ROBERTO VERZICCO, University of Rome "Tor Vergata", DETLEF LOHSE, Physics of Fluids, University of Twente — Injection of a small concentration of gas bubbles into a carrier fluid can result in significant drag reduction in wall bounded turbulent flows. While experimental studies have shown that deformability of the dispersed phase is crucial for strong drag reduction, measurement of local flow conditions to understand the governing mechanism is extremely challenging. In this work we attempt to understand the underlying physics between deformability and drag reduction across a regime of scales in a turbulent Taylor-Couette flow using Direct Numerical Simulations of the carrier flow while a mixture of approaches are used to simulate the dispersed phase (i) Euler-Lagrangian tracking of sub-Kolmogorov ellipsoidal bubbles with a sub-grid deformation model (ii) Fully resolved finite size bubbles with an interaction potential approach to capture the deformation dynamics. We will study and compare the boundary layer profiles, dispersion of the bubbles and shape oscillations of the bubbles as they are transported between the boundary layers and bulk back and forth to get a detailed understanding of the link between deformability and drag reduction.

4:41PM D12.00009 Parameter dependences of the onset location of turbulent liquid jet breakup¹ , ALAN KERSTEIN, AMIRREZA MOVAGHAR, Chalmers Univ., MARK LINNE, Univ. of Edinburgh, MICHAEL OEVERMANN, Chalmers Univ. — A previous study of primary breakup of turbulent liquid jets obtained a $We^{-0.67}$ dependence of breakup onset location on jet Weber number We based on reasonable agreement with measurements and closeness to a theoretical prediction $We^{-2/5}$ inferred from inertial-range phenomenology [1]. It is proposed that breakup onset is instead controlled by the residual presence of the boundary-layer structure of the nozzle flow in the near field of the jet. Assuming that the size of the breakup-inducing eddy is within the scale range of the log-law region, We^{-1} dependence is predicted. This dependence agrees with the measurements more closely than the $We^{-0.67}$ dependence. To predict the dependence on Reynolds number Re , either the friction velocity based on the Blasius friction law or the bulk velocity can be used, where the former yields $Re^{3/8}$ dependence and the latter implies no Re dependence. The latter result is consistent with measurements, but not with the boundary-layer interpretation of breakup onset, so the origin of the measured lack of Re dependence merits further investigation. A preliminary assessment has been made using a computational model of primary breakup. [1] P.-K. Wu, G. M. Faeth, Phys. Fluids 7, 2915 (1995).

¹Work supported by the Knut and Alice Wallenberg Foundation.

4:54PM D12.00010 Experimental study on immiscible jet breakup using refractive index matched oil-water pair¹, XINZHI XUE, JOSEPH KATZ, Johns Hopkins University — A subsea oil well blowout creates an immiscible crude oil jet. This jet fragments shortly after injection, resulting in generation of a droplet cloud. Detailed understanding of the processes involved is crucial for modeling the fragmentation and for predicting the droplet size distribution. High density of opaque droplets near nozzle limits our ability to visualize and quantify the breakup process. To overcome this challenge, two immiscible fluids: silicone oil and sugar water with the same index of refraction (1.4015) are used as surrogates for crude oil and seawater, respectively. Their ratios of kinematic viscosity (5.64), density (0.83) and interfacial tension are closely matched with those of crude oil and seawater. Distribution of the oil phase is visualized by fluorescent tagging. Both phases are also seeded with particles for simultaneous PIV measurements. The measurements are performed within atomization range of Ohnesorge and Reynolds numbers. Index matching facilitates undistorted view of the phase distribution in illuminated section. Ongoing tests show that the jet surface initially rolls up into Kelvin-Helmholtz rings, followed by development of dispersed phase ligaments further downstream, which then break into droplets. Some of these droplets are re-entrained into the high momentum core, resulting in secondary breakup. As the oil layer and ligaments evolve, they often entrain water, resulting in generation of multiple secondary water droplets encapsulated within the oil droplets.

¹This research is made possible by a grant from Gulf of Mexico Research Initiative

Sunday, November 20, 2016 2:57PM - 5:07PM –
Session D13 DFD GPC: Geophysical Fluid Dynamics: Rotating Stratified Flows C124
- Monica Martinez, University of California - Riverside

2:57PM D13.00001 Vorticity Transport in a Two Layer, Double Gyre Ocean Basin, BRYAN KAISER, MIT/WHOI Joint Program, CAROL ANNE CLAYSON, STEVE JAYNE, Woods Hole Oceanographic Institution — The double gyre ocean circulations predicted by strongly frictional, barotropic, linearized ocean models qualitatively agree with the patterns of large scale gyres in the world ocean. However, nonlinear ocean models featuring less intense eddy diffusion parameterization can converge to an infinite number of statistically stationary circulations, depending on the parameterization of dissipation of energy and vorticity. Patterns of vorticity flux and dissipation in a barotropic ocean have been examined previous studies; in this work the inclusion of the first baroclinic mode is examined. The first vertical mode permits the model to be split into two layers, the top approximating the thermocline and the bottom approximating the abyssal circulation. The separation into two layers not only adds realism and but also removes the nonphysical direct restraint of the upper ocean by bottom friction. Steady state circulations for various boundary conditions, sources and sinks of vorticity, and Reynolds numbers are simulated using a parallel pseudo-spectral quasi-geostrophic flow solver and mechanisms of vorticity flux and dissipation are discussed.

3:10PM D13.00002 The sensitivity of rotating Rayleigh-Bénard convection to the Ekman number, MEREDITH PLUMLEY, KEITH JULIEN, PHILIPPE MARTI, University of Colorado Boulder, STEPHAN STELLMACH, Institut für Geophysik, Westfälische Wilhelms-Universität, Münster, JONATHAN AURNOU, EMILY HAWKINS, University of California, Los Angeles — Many geophysical and astrophysical applications of rotating Rayleigh-Bénard convection require no-slip boundaries. These boundaries lead to Ekman pumping, which has a dominant impact on the heat transport and affects the transfer of energy within the system. Here I present the 2D surface of the Nusselt number as a function of the Rayleigh number (Ra) and the Ekman number (E) for no-slip boundaries, generated through a combination of results from experiments, DNS, rescaled DNS, and asymptotic simulations. The Ra - E space is mapped from the transition of the weakly-rotating into the rotation-dominated regime ($E \approx 10^{-7}$) to lower E in the rapidly-rotating regime ($E \approx 10^{-11}$). This exploration provides insight into the sensitivity of the flow to the Ekman number, specifically the effect of the boundaries on the types and ranges of flow structures and the difference between stress-free and no-slip boundaries at low E , a regime of interest for modeling planetary interiors.

3:23PM D13.00003 On the lifetime of a pancake anticyclone in a rotating stratified flow, GIULIO FACCHINI, MICHAEL LE BARS, Aix-Marseille University, CNRS, Ecole Centrale Marseille, Institut sur les Phénomènes Hors Équilibre, UMR 7342, Marseille, France — We present an experimental study of the time evolution of an isolated anticyclonic pancake vortex in a laboratory rotating stratified flow. Motivations come from the variety of compact anticyclones observed to form and persist for a strikingly long lifetime in geophysical and astrophysical settings combining rotation and stratification. We generate anticyclones by injecting a small amount of isodense fluid at the center of a rotating tank filled with salty water linearly stratified in density. Our two control parameters are the Coriolis parameter f and the Brunt-Väisälä frequency N . We observe that anticyclones always slowly decay by viscous diffusion, spreading mainly in the horizontal direction irrespective of the initial aspect ratio. This behavior is correctly explained by a linear analytical model in the limit of small Rossby and Ekman numbers, where density and velocity equations reduce to a single equation for the pressure. Direct numerical simulations further confirm the theoretical predictions. Notably, they show that the azimuthal shear stress generates secondary circulations, which advect the density anomaly: this mechanism is responsible for the slow time evolution, rather than the classical viscous dissipation of the azimuthal kinetic energy.

3:36PM D13.00004 Cylindrical gravity currents in a rotating system, CHING-SEN WU, ALBERT DAI, National Taiwan University — This study aims at investigating the dynamical processes in the formation of stable cylindrical gravity currents, by a full-depth lock release, in a rotating system conducted by direct numerical simulations. The simulations reproduce the major features observed in the laboratory and provide more detailed flow information. Both the qualitative and quantitative measures are provided through the flow patterns and the predicted energy budgets. At the initial stage, during tenth of a revolution of the system, the Kelvin-Helmholtz vortices form and the flow structure maintain nearly axisymmetric. Afterwards, three-dimensionality of flow quickly develops and the outer rim of current breaks away from the body, which gives rise to the maximum dissipation rate in the system. The detached outer rim continues to propagate outward until a maximum radius of propagation is attained. Then the body of current exhibits a regularly contraction-relaxation motion in a period, the energy is transformed back and forth between potential energy and kinetic energy. With the use of high-resolution of numerical computations, the formation of lobe-and-cleft structure and swirling strength for the rotating gravity currents are clearly observed.

3:49PM D13.00005 Convection in rotating flows with simultaneous imposition of radial and vertical temperature gradients, AYAN KUMAR BANERJEE, AMITABH BHATTACHARYA, SRIDHAR BALASUBRAMANIAN, Indian Inst of Tech-Bombay — Laboratory experiments, with a rotating cylindrical annulus and thermal gradient in both radial and vertical directions (so that radial temperature difference decreases with the elevation), were conducted to study the convection dynamics and heat transport. Temperature data captured using thermocouples, combined with ANSYS Fluent simulation hinted at the co-existence of thermal plume and baroclinicity (inclined isotherms). Presence of columnar plume structure parallel to the rotation axis was found, which had a phase velocity and aided in vertical heat transport. Nusselt number (Nu) plotted as a function of Taylor number (Ta) showed the effect of rotation on heat transport in such systems, where the interplay of plumes and baroclinic waves control the scalar transport. Laser based PIV imaging at a single vertical plane also showed evidence of such flow structures.

4:02PM D13.00006 Experiments on point plumes in a rotating environment, DARIA FRANK, JULIEN LANDEL, STUART DALZIEL, PAUL LINDEN, University of Cambridge — Motivated by the Deepwater Horizon oil spill in the Gulf of Mexico we study the dynamics of point plumes in a stratified and homogeneous rotating environment. To this end, we conduct small-scale experiments in the laboratory on salt water and bubble plumes over a wide range of Rossby numbers. The rotation modifies the entrainment into the plume and also inhibits the lateral spreading of the plume fluid which leads to various instabilities in the flow. In particular, we focus on the plume behaviour in the near-source region (where the plume is dominated by the source conditions) and at intermediate water depths, e.g., lateral intrusions at the neutral buoyancy level in the stratified environment. One of the striking features in the rotating environment is the anticyclonic precession of the plume axis which leads to an enhanced dispersion of the plume fluid in the ambient and which is absent in the non-rotating system. In this talk, we present our experimental results and develop simple models to explain the observed plume dynamics.

4:15PM D13.00007 Quasigeostrophic investigations of non-hydrostatic, stably-stratified and rapidly rotating flows¹, KEITH JULIEN, DAVID NIEVES, IAN GROOMS, Dept. Applied Mathematics, University of Colorado at Boulder, JEFFREY WEISS, Dept. Atmospheres and Oceans, University of Colorado at Boulder — We present an investigation of rapidly rotating stratified turbulence where the stratification strength is varied from weak to strong. The investigation is set in the context of a reduced model derived from the Boussinesq equations that retains anisotropic inertia-gravity waves with order-one frequencies and highlights a regime of wave-eddy interactions. Numerical simulations are performed where energy is injected by a stochastic forcing of vertical velocity, which forces wave modes only. The simulations reveal two regimes characterized by the presence of well-formed, persistent and thin turbulent layers of locally weakened stratification at small Froude numbers, and by the absence of layers at large Froude numbers. Both regimes are characterized by a large-scale barotropic dipole enclosed by small-scale turbulence. When the Reynolds number is not too large, a direct cascade of barotropic kinetic energy is observed, leading to total energy equilibration. We examine net energy exchanges that occur through vortex stretching and vertical buoyancy flux. We find that the baroclinic motions inject energy directly to the largest scales of the barotropic mode, implying that the large-scale barotropic dipole is not the end result of an inverse cascade within the barotropic mode.

¹NSF DMS 1317666, 1444503; NSF EAR 1067944, NSF OCE 1245944

4:28PM D13.00008 Observations of Instabilities in Stratified Taylor-Couette Flow¹, BRUCE RODENBORN, Centre College, RUY IBANEZ, Baylor University, HARRY L. SWINNEY, Center for Nonlinear Dynamics and Department of Physics, University of Texas at Austin — Inviscid analyses by Molemaker et al. (Phys. Rev. Lett. 86, 5270, 2001) and by Dubrulle et al. (Astron. Astrophys. 29, 1, 2005) predicted that a fluid with a vertically varying density will be less stable than a uniform fluid when the fluid is contained inside a concentric rotating cylinder system and subject to anticyclonic shear. Dubrulle et al. named this instability the stratorotational instability and a subsequent viscous theory by Shalybkov and Rudiger (Astron. Astrophys. 438, 411, 2005) hypothesized that such stratified flow is stable when the ratio of outer and inner cylinder rotation rates μ is less than the ratio of the inner and outer cylinder radii η . Le Bars and Le Gal (Phys. Rev. Lett. 99, 064502, 2007) confirmed this hypothesis in experiments for $Re < 1200$ with $Re \equiv (r_o - r_i)\Omega_i r_i / \nu$. However, we find the SRI exists for $\mu > \eta$ when the density gradient is large. We also find that the axial wavelength scales linearly with the internal Froude number and that the onset of the SRI is suppressed for $Re > 4000$, a region previously unexplored in experiments. For $Re > 8000$, we find that the fluid does not exhibit the SRI but transitions to a spatially nonperiodic state that mixes the fluid.

¹This research was supported in part by the Sid W. Richardson Foundation.

4:41PM D13.00009 Characterisation of a quasi-periodic mixing mechanism in stratified turbulent Taylor-Couette flow¹, KANWAR NAIN SINGH, JAMIE PARTRIDGE, STUART DALZIEL, DAMTP, University of Cambridge, C.P. CAULFIELD, DAMTP, University of Cambridge & BPI, University of Cambridge, MATHEMATICAL UNDERPINNINGS OF STRATIFIED TURBULENCE (MUST) TEAM — We conduct experiments to examine a quasi-periodic mixing event that occurs in stratified Taylor-Couette flow, i.e. axially-stratified flow in the annular region between two concentric cylinders which can rotate at different angular velocities. It has been previously observed that, in two-layer density stratified Taylor-Couette flow, there is an intermittent periodic mixing event which is continuously advected around the annulus. We track this mixing event within the annular gap of the Taylor-Couette apparatus by continuously measuring density perturbations at the sharp interface separating the two layers as a function of radial location. It has been seen that when $Ri = \frac{g' R_o}{(R_i \Omega_i)^2} \sim 7$, where R_i , R_o are the inner and outer cylinder radius, respectively, g' the reduced gravity characterising the density jump between the layers and Ω_i is the rotation rate of the inner cylinder, the power of the mixing event in the frequency spectrum of the density data drops significantly. This process seems to be consistent at all radial locations throughout the annulus. This phenomenon is further investigated using velocity information obtained from particle image velocimetry (PIV).

¹EPSRC programme grant EP/K034529/1 & SGPC-CCT Scholarship

4:54PM D13.00010 Quantifying mixing in stratified plane Couette flows in a tracer-based coordinate¹, QI ZHOU, JOHN TAYLOR, DAMTP, University of Cambridge, COLM-CILLE CAULFIELD, BP Institute and DAMTP, University of Cambridge, PAUL LINDEN, DAMTP, University of Cambridge — The mixing properties of statically stable density interfaces subject to imposed vertical shear are investigated using direct numerical simulations. At the start of each simulation, a sharp density interface is introduced at the mid-plane between two flat, counter-moving horizontal walls. Particular attention is paid to the effects of varying Prandtl number over two orders of magnitude from 0.7, 7 to 70. The dynamics of the interface varies from shear-induced overturning at small Richardson numbers to internal waves at large Richardson numbers. At sufficiently large Peclet number, the dynamics allows the density interface to remain sharp. This is due to the combined effects of the ‘scouring’ induced by the turbulence external to the interface and comparatively weak molecular diffusion across the core region of the interface. The effective diapycnal diffusivity and irreversible density flux are quantified in the tracer-based coordinate proposed by Winters and D’Asaro (*J. Fluid Mech.*, 317, 1996) and Nakamura (*J. Atmos. Sci.*, 53, 1996). We further discuss the dependence of the effective diffusivity on the characteristic parameters of the flow, and the implications for mixing efficiency and layer formation.

¹Supported by UK EPSRC Programme Grant EP/K034529/1 ‘Mathematical Underpinnings of Stratified Turbulence’

Sunday, November 20, 2016 2:57PM - 5:07PM – Session D14 Waves: Internal and Interfacial Waves C125-126 - T. R. Akylas, MIT

2:57PM D14.00001 3D instability of internal gravity wave beams¹, T. R. AKYLAS, MIT, TAKESHI KATAOKA, Kobe University, Japan, BOYU FAN, MIT — The stability of isolated and interacting internal gravity wave beams (IGWB) to 3D perturbations is studied, based on the beam-mean-flow equations derived in Kataoka & Akylas (2015). Depending on the beam profile and amplitude, a single uniform IGWB as well as two interacting IGWB which propagate in the same or opposite directions, can be subject to 3D modulational instability brought about by a purely inviscid nonlinear mechanism. Moreover, for moderate viscous dissipation, the mean flow induced by a mechanism analogous to acoustic streaming can cause significant distortion, leading to breakdown of forced IGWB with small lateral amplitude variations. These findings suggest that modulational and streaming instabilities are central to 3D IGWB dynamics, in contrast to the widely-studied PSI of sinusoidal wavetrains, which is most relevant to beams with nearly monochromatic profile only.

¹Supported by NSF

3:10PM D14.00002 Effect of background rotation on the evolution of 3D internal gravity wave beams¹, BOYU FAN, T. R. AKYLAS, MIT — The effect of background rotation on the 3D propagation of internal gravity wave beams (IGWB) is studied, assuming that variations in the along-beam and transverse directions are of long length scale relative to the beam width. The present study generalizes the asymptotic model of KA (Kataoka & Akylas 2015) who considered the analogous problem in the absence of rotation. It is shown that the role of mean vertical vorticity in the earlier analysis is now taken by the flow mean potential vorticity (MPV). Specifically, 3D variations enable resonant transfer of energy to the flow MPV, resulting in strong nonlinear coupling between a 3D IGWB and its induced mean flow. This coupling mechanism is governed by a system of two nonlinear equations of the same form as those derived in KA. Accordingly, the induced mean flow features a purely inviscid modulational component, as well as a viscous one akin to acoustic streaming; the latter grows linearly with time for a quasi-steady IGWB. On the other hand, owing to background rotation, the induced mean flow in the vicinity of the IGWB is no longer purely horizontal and develops an asymmetric behavior.

¹Supported by NSF

3:23PM D14.00003 Harmonic generation by nonlinear refraction of a single internal wave mode with rotation, SCOTT WUNSCH, Johns Hopkins University — Weakly nonlinear theory is used to explore the dynamics of a mode-1 internal tide in variable stratification with rotation. Nonlinear refraction at the pycnocline generates a perturbation which is forced with double the original frequency and wavenumber. The dynamics of the perturbation are analogous to a forced harmonic oscillator, with the steady state solution matching the forcing frequency and wavenumber. The perturbation exhibits resonance when its frequency is close to a natural frequency of the system. Enhanced dissipation due to the harmonic occurs near resonance, and its contribution to ocean tidal dissipation may be significant in some environments. The results are relevant to recent observations of harmonics of the diurnal tide in the South China Sea. More generally, nonlinear refraction may contribute to the dissipation of oceanic internal tides and the transfer of energy to smaller scales.

3:36PM D14.00004 An experimental investigation of energy transmission from an evanescent to a propagating region., ALLISON LEE, JULIE CROCKETT, Brigham Young University — Tidal flow over oceanic bathymetry is a well known generator of internal waves. However, in the deep ocean there are many regions of weak stratification and the tides will generate only evanescent waves which decay exponentially as they propagate away from their source. In locations where stronger stratification exists above the weak, evanescent waves can form propagating internal waves as they approach a depth with a stratification corresponding to their frequency (turning depth). An experimental study of the energy transfer from evanescent regions to propagating regions due to evanescent waves passing through a turning depth is presented. The effects of varying exponential stratification profiles and single and multi-peak topographical features are described and results are compared with linear theory approximations.

3:49PM D14.00005 Effect of pycnocline thickness on internal wave bolus transport¹, MICHAEL ALLSHOUSE², Department of Mechanical and Industrial Engineering, Northeastern University, HARRY SWINNEY, Department of Physics, University of Texas at Austin — Internal waves shoaling on a continental slope can produce bulges, which are vortices that develop and travel upslope with the shoaling internal wave. In contrast to propagating solitary waves, bulges can trap and transport nutrient rich water upslope. Past laboratory investigations of bulge generation and transport have examined systems that have two layers of uniform density. The present laboratory experiment examines bulge formation and transport as a function of the thickness of a model pycnocline where there is a continuous variation in density between two regions of constant density. Our dye based measurements for transition layers varying in thickness from 2 to 30 cm demonstrate that fluid transport by bulges exhibits a maximum as the thickness of the transition layer is varied. Complementary Navier-Stokes direct numerical simulations, analyzed using Lagrangian coherent structure techniques, compare well with the laboratory observations.

¹ONR MURI Grant No. N000141110701

²Department of Physics, University of Texas at Austin

4:02PM D14.00006 Internal Wave Scattering in Idealized and Realistic Continental Slope Canyons¹, ROBERT NAZARIAN, SONYA LEGG, Princeton Univ — When internal waves interact with topography, such as continental slopes, they can deposit their energy to local dissipation and mixing. Submarine canyons comprise about ten percent of global continental slopes, and can enhance the local dissipation of internal wave energy, yet parameterizations of canyon mixing processes are currently missing from ocean models. As a first step in developing such parameterizations, a parameter space study of M2 tidal-frequency, low-mode internal waves interacting with idealized canyon topographies was conducted. A two-pronged approach was employed in which a suite of MITgcm simulations was compared with a novel, analytical ray tracing scheme. The most noticeable result was that, as the ratio of the canyon mouth width to canyon length decreased, there was a marked increase in the relative energy loss. This energy loss also increased as the canyon sidewall steepness increased. Processes leading to this increased energy loss include increased energy focusing, increasing vertical wavenumber via multiple reflections for non-vertical sidewalls and the presence of arrested lee waves for vertical sidewalls. To test the robustness of these results, we model the energy lost from remotely-generated M2 internal tides in three realistic canyons with very different geometries: Veatch, La Jolla and Eel Canyons, comparing results with both idealized simulations and microstructure data taken from these locations. We also discuss how current parameterizations of tidally-driven diapycnal mixing can be extended to include the effects of continental slope canyons.

¹NOAA award NA08OAR4320752

4:15PM D14.00007 Generation of realistic tsunami waves using a bottom-tilting wave maker¹, YONG SUNG PARK, University of Dundee, JIN HWAN HWANG, Seoul National University — Tsunamis have caused more than 260,000 human losses and \$250 billion in damage worldwide in the last ten years. Observations made during 2011 Japan Tohoku Tsunami revealed that the commonly used waves (solitary waves) to model tsunamis are at least an order-of-magnitude shorter than the real tsunamis, which calls for re-evaluation of the current understanding of tsunamis. To prompt the required paradigm shift, a new wave generator, namely the bottom-tilting wave generator, has been developed at the University of Dundee. The wave tank is fitted with an adjustable slope and a bottom flap hinged at the beginning of the slope. By moving the bottom flap up and down, we can generate very long waves. Here we will report characteristics of waves generated by simple bottom motions, either moving it upward or downward from an initial displacement ending it being horizontal. Two parameters, namely the initial displacement of the bottom and the speed of the motion, determine characteristics of the generated waves. Wave amplitudes scale well with the volume flux of the displaced water. On the other hand, due to combined effects of nonlinearity and dispersion, wavelengths show more complicated relationship with the two bottom motion parameters. We will also demonstrate that by combining simple up and down motions, it is possible to generate waves resembling the one measured during 2011 tsunami.

¹YSP acknowledges financial support from the Royal Society of Edinburgh through the Royal Society of Edinburgh and Scottish Government Personal Research Fellowship Co-Funded by the Marie-Curie Actions.

4:28PM D14.00008 Cylindrical waves at the interface of viscous immiscible fluids¹, RATUL DASGUPTA, PALAS KUMAR FARSOIYA, Dept. Chemical Engineering, Indian Inst of Tech-Bombay — We conduct Navier-Stokes simulations of cylindrical, axisymmetric standing gravity waves at the interface of radially unbounded, immiscible viscous fluids. The fluid motion generated by these oscillations are studied. Results from the numerical solutions are compared to the analytical solution of an integro-differential equation representing the amplitude of motion of the interface. Standing waves are initiated at the interface as zeroth order Bessel's mode at rest i.e. $h(r, 0) = H_0(1 + \epsilon J_0(kr))$ where H_0 is the undisturbed fluid depth in the simulation, chosen to be large enough for deep water approximation to hold. For small initial amplitudes (compared to $2\pi k^{-1}$), we obtain good agreement with the analytical solution at early times. As we increase initial amplitude, the time period of the first oscillation is found to increase. Diffusion of vorticity from the interface is studied as a function of initial amplitude. We compare our results to the analytical solution obtained from the corresponding planar problem (Prosperetti, 1981). We will discuss these results in the framework of the viscous Cauchy-Poisson (initial-value) problem between two fluids, and also compare our results to the viscous, single fluid case (Miles, 1968).

¹We thank IRCC, IIT Bombay for financial support.

4:41PM D14.00009 Study of downward annular pipe flow using combined laser-based approaches¹, JAE SIK AN, Imperial College London, ANDREY CHERDANTSEV, Institute of Thermophysics, Siberian Branch of Russian Academy of Science, IVAN ZADRAZIL, OMAR MATAR, CHRISTOS MARKIDES, Imperial College London — In downward annular flow, the liquid phase flows as a film along the pipe wall and the gas flows in the core of the pipe. The liquid free-surface is covered by a complex multiscale system of waves. The interaction dynamics of the interfacial waves with each other and with the gas stream exert a significant influence on the pressure drop, heat transfer and mass interchange between the phases. The complexity of the interface requires the application of measurement techniques with high spatiotemporal resolution. In this work, two approaches based on the principle of laser-induced fluorescence, namely planar LIF and brightness-based LIF, are applied simultaneously to study interfacial phenomena in these flows, while simultaneous LIF and PIV are used to obtain velocity field information in the liquid phase underneath the waves. Sources of measurement bias are then analysed: total internal reflection at the out-of-plane interface; steep longitudinal slopes and transverse wave curvature; presence of gas bubbles in the liquid film. Although each method has its own limitations, a combined technique can provide reliable spatiotemporal measurements of film thickness to accompany the velocity information. Finally, flow development is studied in a moving frame of reference over long lengths.

¹EPSRC UK Programme Grant MEMPHIS (EP/K003976/1)

4:54PM D14.00010 Characterization of interfacial waves in horizontal core-annular flow¹, SUMIT TRIPATHI, IITB-Monash Research Academy, Mumbai, India, AMITABH BHATTACHARYA, RAMESH SINGH, Department of Mechanical Engineering, IIT Bombay, India, RICO F. TABOR, School of Chemistry, Monash University, Australia — In this work, we characterize interfacial waves in horizontal core annular flow (CAF) of fuel-oil and water. Experimental studies on CAF were performed in an acrylic pipe of 15.5mm internal diameter, and the time evolution of the oil-water interface shape was recorded with a high speed camera for a range of different flow-rates of oil (Q_o) and water (Q_w). The power spectrum of the interface shape shows a range of notable features. First, there is negligible energy in wavenumbers larger than $2\pi/a$, where a is the thickness of the annulus. Second, for high Q_o/Q_w , there is no single dominant wavelength, as the flow in the confined annulus does not allow formation of a preferred mode. Third, for lower Q_o/Q_w , a dominant mode arises at a wavenumber of $2\pi/a$. We also observe that the power spectrum of the interface shape depends weakly on Q_w , and strongly on Q_o , perhaps because the net shear rate in the annulus appears to depend weakly on Q_w as well. We also attempt to build a general empirical model for CAF by relating the interfacial stress (calculated via the mean pressure gradient) to the flow rate in the annulus, the annular thickness and the core velocity.

¹Authors are thankful to Orica Mining Services (Australia) for the financial support

Sunday, November 20, 2016 2:57PM - 5:07PM – Session D15 Bio: Aneurysms E143/144 - Alberto Aliseda, University of Washington

2:57PM D15.00001 Volumetric PIV in Patient-Specific Cerebral Aneurysm, MELISSA BRINDISE, Purdue Univ, BEN DICKERHOFF, Marquette University, DAVID SALONER, University of California San Francisco, VITALIY RAYZ, University of Wisconsin - Milwaukee, PAVLOS VLACHOS, Purdue Univ — Cerebral aneurysms impose a unique challenge in which neurosurgeons must assess and decide between the risk of rupture and risk of treatment for each patient. Risk of rupture is often difficult to determine and most commonly assessed using geometric data including the size and shape of the aneurysm and parent vessel. Hemodynamics is thought to play a major role in the growth and rupture of a cerebral aneurysm, but its specific influence is largely unknown due to the inability of *in vivo* modalities to characterize detailed flow fields and limited *in vitro* studies. In this work, we use a patient-specific basilar tip aneurysm model and volumetric particle image velocimetry (PIV). *In vivo*, 4-D PC-MRI measurements were obtained for this aneurysm and the extracted pulsatile waveform was used for the *in vitro* study. Clinically relevant metrics including wall shear stress (WSS), oscillatory shear index (OSI), relative residence time (RRT), 3-D pressure contours, and pressure wave speed were subsequently computed. This is the first study to investigate *in vitro* 3-D pressure fields within a cerebral aneurysm. The results of this study demonstrate how these metrics influence the biomechanics of the aneurysm and ultimately their effect on the risk of rupture.

3:10PM D15.00002 ABSTRACT WITHDRAWN –

3:23PM D15.00003 Proper orthogonal decomposition analyses of high-frequency transient flow instabilities in intracranial aneurysms, MUHAMMAD OWAIS KHAN, CHRISTOPHE CHNAFA, University of Toronto, KRISTIAN VALEN-SENDSTAD, Simula Research Laboratory, DAVID A. STEINMAN, University of Toronto — Treatment of incidentally detected intracranial aneurysms (IAs) can exceed the natural risk of rupture. Abnormal hemodynamic forces, derived from image-based CFD, have, therefore, been proposed to assess the risk of IA rupture. Although majority of the CFD-literature has shown laminar and stable flows in IAs, recent high-resolution CFD simulations have highlighted the presence of transient high-frequency flow instabilities, or turbulent-like flows, consistent with experimental evidence from early 70s and 80s. However, whether flows in IAs are turbulent is still not fully understood. We performed DNS of 6 patient-specific IAs exhibiting varying levels of "turbulence". Proper orthogonal decomposition (POD) eigenspectra revealed that ~96% of the kinetic energy was concentrated in the first mode. Time-windowed POD showed that energy in higher modes ($k > 50$) was dominated by contributions from deceleration phase. Velocity fields were reconstructed from the higher modes to highlight presence of distinct flow structures. We also identified presence of discrete frequency bands by applying wavelet analyses to time-velocity traces, and used novel Fourier-based hemodynamic indices to characterize the nature of these turbulent-like flows.

3:36PM D15.00004 Wall Shear Stress Distribution in a Patient-Specific Cerebral Aneurysm Model using Reduced Order Modeling., SUYUE HAN, GARY HAN CHANG, University of Massachusetts Amherst, CLEMENS SCHIRMER, Geisinger Health System, YAHYA MODARRES-SADEGHI, University of Massachusetts Amherst — We construct a reduced-order model (ROM) to study the Wall Shear Stress (WSS) distributions in image-based patient-specific aneurysms models. The magnitude of WSS has been shown to be a critical factor in growth and rupture of human aneurysms. We start the process by running a training case using Computational Fluid Dynamics (CFD) simulation with time-varying flow parameters, such that these parameters cover the range of parameters of interest. The method of snapshot Proper Orthogonal Decomposition (POD) is utilized to construct the reduced-order bases using the training CFD simulation. The resulting ROM enables us to study the flow patterns and the WSS distributions over a range of system parameters computationally very efficiently with a relatively small number of modes. This enables comprehensive analysis of the model system across a range of physiological conditions without the need to re-compute the simulation for small changes in the system parameters.

3:49PM D15.00005 Modeling contrast agent flow in cerebral aneurysms: comparison of CFD with medical imaging¹, VITALIY RAYZ, University of Wisconsin - Milwaukee, ALIREZA VALI, Medical College of Wisconsin, MONICA SIGOVAN, Hospital de la Croix-Rousse - Creatis, Lyon, France, MICHAEL LAWTON, DAVID SALONER, University of California San Francisco, LOIC BOUSSEL, Hospital de la Croix-Rousse - Creatis, Lyon, France — PURPOSE: The flow in cerebral aneurysms is routinely assessed with X-ray angiography, an imaging technique based on a contrast agent injection. In addition to requiring a patient's catheterization and radiation exposure, the X-ray angiography may inaccurately estimate the flow residence time, as the injection alters the native blood flow patterns. Numerical modeling of the contrast transport based on MRI imaging, provides a non-invasive alternative for the flow diagnostics. METHODS: The flow in 3 cerebral aneurysms was measured *in vivo* with 4D PC-MRI, which provides time-resolved, 3D velocity field. The measured velocities were used to simulate a contrast agent transport by solving the advection-diffusion equation. In addition, the flow in the same patient-specific geometries was simulated with CFD and the velocities obtained from the Navier-Stokes solution were used to model the transport of a virtual contrast. RESULTS: Contrast filling and washout patterns obtained in simulations based on MRI-measured velocities were in agreement with those obtained using the Navier-Stokes solution. Some discrepancies were observed in comparison to the X-ray angiography data, as numerical modeling of the contrast transport is based on the native blood flow unaffected by the contrast injection.

¹NIH HL115267

4:02PM D15.00006 4D Magnetic Resonance Velocimetry in a 3D printed brain aneurysm, OMID AMILI, University of Minnesota, DANIELE SCHIAVAZZI, Stanford University, FILIPPO COLETTI, University of Minnesota — Cerebral aneurysms are of great clinical importance. It is believed that hemodynamics play a critical role in the development, growth, and rupture of brain arteries with such condition. The flow structure in the aneurysm sac is complex, unsteady, and three-dimensional. Therefore the time-resolved measurement of the three-dimensional three-component velocity field is crucial to predict the clinical outcome. In this study magnetic resonance velocimetry is used to assess the fluid dynamics inside a 3D printed model of a giant intracranial aneurysm. We reach sub-millimeter resolution while resolving sixteen instances within the cardiac cycle. The physiological flow waveform is imposed using an in-house built pump in a flow circuit where the cardiovascular impedance is matched. The flow evolution over time is reconstructed in detail. The complex flow structure is characterized by vortical and helical motions that reside in the aneurysm for most part of the cycle. The 4D pressured distribution is also reconstructed from the velocity field. The present case study was used in a previous CFD challenge, therefore these results may provide useful experimental comparison for simulations performed by other research groups.

4:15PM D15.00007 Physiologically-relevant measurements of flow through coils and stents: towards improved modeling of endovascular treatment of intracranial aneurysms, MICHAEL BARBOUR, MICHAEL LEVITT, University of Washington, CHRISTIAN GEINDREAU, SABINE ROLLAND DU ROSCOAT, Université Grenoble Alpes, LUKE JOHNSON, KESHAV CHIVUKULA, ALBERTO ALISEDA, University of Washington — The hemodynamic environment in cerebral aneurysms undergoing flow-diverting stent (FDS) or coil embolization treatment plays a critical role in long-term outcomes. Standard modeling approaches to endovascular coils and FDS simplify the complex geometry into a homogenous porous volume or surface through the addition of a Darcy-Brinkman pressure loss term in the momentum equation. The inertial and viscous loss coefficients are typically derived from published *in vitro* studies of pressure loss across FDS and coils placed in a straight tube, where the only fluid path is across the treatment – an unrealistic representation of treatment apposition *in vivo*. The pressure drop across FDS and coils in side branch aneurysms located on curved parent vessels is measured. Using PIV, the velocity at the aneurysm neck plane is reconstructed and used to determine loss coefficients for better models of endovascular coils or FDS that account for physiological placement and vessel curvature. These improved models are incorporated into CFD simulations and validated against *in vitro* model PIV velocity, as well as compared to microCT-based coil/stent-resolving CFD simulations of patient-specific treated aneurysm flow.

4:28PM D15.00008 Computational Study of Intracranial Aneurysms with Flow Diverting Stent: Correlation with Surgical Outcome, YIK SAU TANG, TIN LOK CHIU, ANDERSON CHUN ON TSANG, GILBERTO KA KIT LEUNG, KWOK WING CHOW, University of Hong Kong — Intracranial aneurysm, abnormal swelling of the cerebral artery, can cause massive internal bleeding in the subarachnoid space upon aneurysm rupture, leading to a high mortality rate. Deployment of a flow diverting stent through endovascular technique can obstruct the blood flow into the aneurysm, thus reducing the risk of rupture. Patient-specific models with both bifurcation and sidewall aneurysms have been investigated. Computational fluid dynamics analysis with physiological boundary conditions has been performed. Several hemodynamic parameters including volume flow rate into the aneurysm and the energy (sum of the fluid kinetic and potential energy) loss between the inlet and outlets were analyzed and compared with the surgical outcome. Based on the simulation results, we conjecture that a clinically successful case might imply less blood flow into the aneurysm after stenting, and thus a smaller amount of energy loss in driving the fluid flow in that portion of artery. This study might provide physicians with quantitative information for surgical decision making. (Partial financial support by the Innovation and Technology Support Program (ITS/011/13 & ITS/150/15) of the Hong Kong Special Administrative Region Government)

4:41PM D15.00009 A hemodynamic-based dimensionless parameter for predicting rupture of intracranial aneurysms.¹, HAFEZ ASGHARZADEH, NICOLE VARBLE, HUI MENG, IMAN BORAZJANI, The State University of New York at Buffalo — 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 hemodynamic factors are believed to be the most influential ones. Here, we carry out three-dimensional high resolution simulations on human subjects IAs to test a dimensionless number, denoted as An number, to classify the flow mode. An number is defined as the ratio of the time takes the parent artery flow transports through the expansion region to the time required for vortex formation. Furthermore, we investigate the correlation of IA flow mode and WSS/OSI on the human subject IAs. Finally, we test if An number can distinguish ruptured from unruptured IAs on a database containing 204 human subjects IAs.

¹This work was supported by National Institute Of Health (NIH) grant R03EB014860 and the Center of Computational Research (CCR) of University at Buffalo.

4:54PM D15.00010 Towards the evaluation of the pathological state of ascending thoracic aneurysms: integration of in-vivo measurements and hemodynamic simulations, ALESSANDRO BOCCADIFUOCO, Scuola Superiore Sant'Anna, ALESSANDRO MARIOTTI, DICI - University of Pisa, SIMONA CELI, NICOLA MARTINI, Fondazione Toscana Gabriele Monasterio, MARIA VITTORIA SALVETTI, DICI - University of Pisa — Ascending thoracic aortic aneurysms are cardiovascular diseases consisting in a dilation of the ascending thoracic aorta. Since indicating a weakness of the arterial wall, they can lead to major complications with significant mortality rate. Clinical decisions about surgery are currently based on the maximum aortic diameter, but this single index does not seem a reliable indicator of the pathological state of the aorta. Numerical simulations of the blood flow inside the aneurysm may give supplementary information by quantifying important indices that are difficult to be measured, like the wall shear stress. Our aim is to develop an efficient platform in which in-vivo measurements are used to perform the hemodynamic simulations on a patient-specific basis. In particular, we used real geometries of thoracic aorta and focused on the use of clinical information to impose accurate boundary conditions at the inlet/outlets of the computational model. Stochastic analysis was also performed, to evaluate how uncertainties in the boundary parameters affect the main hemodynamic indicators, by considering both rigid and deformable walls. Stochastic calibration of numerical parameters against clinical data is in progress and results will be possibly shown.

Sunday, November 20, 2016 2:57PM - 5:07PM –
Session D16 Drops: Evaporation D133/134 - Deniel Anderson, George Mason University

2:57PM D16.00001 Microregion model of a contact line including evaporation, kinetics and slip, DANIEL ANDERSON, George Mason University, VLADISLAV JANECEK, ArcelorMittal Global R and D — We consider the evaporation of a liquid on a uniformly heated solid substrate. In the framework of lubrication theory we consider hydrodynamics, heat conduction, phase change, evaporation kinetics, and slip. Our model focuses only on the contact line 'inner' region which allows us to quantify the impact of evaporation on the apparent contact angle and microregion heat transfer. The linearized problem with respect to the substrate overheating is solved analytically. The analytical solutions are compared with full numerical solutions and to predictions of Hocking (Physics of Fluids, 1995).

3:10PM D16.00002 Correlation for Sessile Drop Evaporation¹, PETER KELLY-ZION, CHRISTOPHER PURSELL, GREGORY WASSOM, BRENTON MANDELKORN, CHRIS NKINTHORN, Trinity Univ — To better understand how the evaporation of sessile drops and small puddles is controlled by the vapor phase transport mechanisms of mass diffusion and buoyancy-induced convection, the evaporation rates of eight liquids evaporating under a broad range of ambient conditions were correlated with physical and geometrical properties. Examination of the correlation provides valuable insight into how the roles of diffusive and convective transport change with physical and geometrical parameters. The correlation predicts measured evaporation rates to within a root-mean-square error of 7.3%. The correlation is composed of two terms, a term which provides the rate of evaporation under diffusion-only conditions, and a term which provides the influence of convection. This second term suggests the manner in which the processes of diffusion and convection are coupled. Both processes are dependent on the distribution of the vapor, through the molar concentration gradient for diffusion and through the mass density gradient for convection. The term representing the influence of convection is approximately inversely proportional to the square root of diffusivity, indicating the tendency of diffusive transport to reduce convection by making the vapor distribution more uniform.

¹Financial support was provided by the ACS Petroleum Research Fund.

3:23PM D16.00003 Understanding thermal Marangoni flow in water sessile evaporating drops via 3D-PTV¹, MASSIMILIANO ROSSI, ALVARO MARIN, CHRISTIAN J. KAEHLER, Bundeswehr University Munich — Understanding the flow inside sessile evaporating drops is of great interest both from a fundamental and technological point of view. Despite strong research efforts in the recent years, a complete picture on the phenomena involved in this process and a way to control them is still far to be reached. This is due to a lack of reliable experimental data on the internal flow but more dramatically on the interfacial flow. A relevant open debate concerns the role played by the Marangoni flow induced by thermal gradients. We recently show how 3D particle tracking techniques are suitable to measure the internal flow of drops and to derive quantities such as surface shear and surface tension differences (Marin et al., Soft Matter, 2016). Such experiments also indicated an increase of the thermal Marangoni flow as the droplet becomes thinner, in disagreement with current theoretical models and simulations. A possible reason for that could be a discrepancy of the imposed boundary conditions in the simulations and the experimental ones. This work follows up these observations with fully 3D time-resolved measurements of the flow inside drops evaporating on a quartz substrate, which temperature is controlled using a feedback temperature control and a microscope incubator system.

¹Supported by DFG, grant no. KA 1808/22

3:36PM D16.00004 Experimental investigation of interfacial phenomena in evaporating sessile droplets for evaporative cooling applications, BRENDAN MACDONALD, MD. ALMOSTASIM MAHMUD, University of Ontario Institute of Technology (UOIT) — Evaporation of sessile droplets has applications in many fields, including evaporative cooling technology. An example from nature is human perspiration. Evaporative cooling applications typically operate at atmospheric pressure and 20 to 80°C, and systems that mimic perspiration require droplets that are continuously fed fluid. A number of studies have investigated phenomena associated with evaporating sessile droplets including (1) interfacial energy transport, (2) distribution of the evaporation flux along the interface, and (3) temperature discontinuities at the liquid-vapor interface; however, many of these studies were not undertaken in the regime relevant to evaporative cooling and used low pressures and temperatures or droplets that were not continuously fed fluid and changed shape as they were depleted. We will present the results from our experimental study, which examined these phenomena in the regime relevant to evaporative cooling to determine if they are present and if they have an impact on the evaporation behavior. In this regime we found that conduction provided a majority of the energy required for evaporation, the local evaporation flux changed depending on thermocapillary convection, and interfacial temperature discontinuities were present.

3:49PM D16.00005 Sharp Interface Level Set Method based Study for Evaporation of a Sessile Droplet on Hydrophilic and Hydrophobic Substrates, JAVED SHAIKH¹, ATUL SHARMA², RAJNEESH BHARDWAJ³, Department of Mechanical Engineering, Indian Institute of Technology Bombay — The evaporation of a sessile droplet is important in many applications like hot-spot cooling, surface patterning etc. An understanding of the droplet dynamics in terms of evaporation rate, evaporative cooling and substrate wettability is important for designing the droplet based devices. Extensive theoretical and experimental research has been conducted on evaporating droplets in recent years; however, the effect of surrounding vapors on the evaporation dynamics of a sessile droplet is not found in the literature. In this work, an in-house sharp-interface level set code based on the Ghost Fluid Method (GFM) is used. Energy, species, and momentum equations are coupled for studying the sessile droplet evaporation phenomenon on hydrophilic and hydrophobic substrates. Different modes of droplet evaporation i.e. constant contact radius (CCR), constant contact angle (CCA) are observed for the two types of substrates. The coupling of energy and species equations is used for capturing the evaporating cooling induced dip in the droplet surface temperature. The effect of surrounding vapors like fluorocarbon vapors, on the evaporative cooling, is presented for water droplet on the hydrophilic and hydrophobic substrates.

¹Research Scholar

²Professor

³Associate Professor

4:02PM D16.00006 Geometrically-controlled drop evaporation: Dynamics and universal scaling law¹, KHELLIL SEFIANE, The University of Edinburgh, PEDRO SAENZ, Massachusetts Institute of Technology, ALEXANDER WRAY, ZHIZHAO CHE, OMAR MATAR, Imperial College London, PRASHANT VALLURI, The University of Edinburgh, JUNGHO KIM, The University of Maryland — The evaporation of a liquid drop on a solid substrate is a remarkably common phenomenon. Yet, the complexity of the underlying mechanisms has constrained previous studies to spherically-symmetric configurations. Here we present an investigation of well-defined, non-spherical evaporating drops of pure liquids and binary mixtures. We deduce a new universal scaling law for the evaporation rate valid for any shape and demonstrate that more curved regions lead to preferential localized depositions in particle-laden drops. Furthermore, geometry induces well-defined flow structures within the drop that change according to the driving mechanism and spatially-dependent thresholds for thermocapillary instabilities. In the case of binary mixtures, geometry dictates the spatial segregation of the more volatile component as it is depleted. In the light of our results, we believe that the drop geometry can be exploited to facilitate precise local control over the particle deposition and evaporative dynamics of pure drops and the mixing characteristics of multicomponent drops.

¹Memphis Multiphase (EPSRC EP/K003976/1) & ThermaPOWER (EU IRSESPIRSES GA-2011-294905)

4:15PM D16.00007 Influence of relative humidity and ambient temperature on hydrothermal waves (HTWs) of organic solvent volatile droplets¹, DANIEL OREJON, YUTAKU KITA, I2CNER, YUYA OKAUCHI, YUKI FUKATANI, MASAMICHI KOHNO, Kyushu University, YASUYUKI TAKATA, I2CNER, KHELLIL SEFIANE, University of Edinburgh, JUNGHO KIM, University of Maryland — Droplets of organic solvents undergoing evaporation have been found to display distinctive hydrothermal patterns or HTWs at the liquid-vapor interface. Since the evaporation of mentioned organic solvents in ambient conditions is ubiquitous, in this work we investigate the effect of ambient temperature and relative humidity on the self-generated HTWs by means of infrared thermography. The intensity of the HTWs was found to decrease when lowering the ambient temperature due to a reduction in droplet evaporative cooling. On other hand, the enhancement or suppression of the HTWs was also possible by controlling the relative humidity of the system. Absorption and/or condensation of water vapor onto the evaporating droplet was found to be the main cause for the differences observed on the HTWs retrieved at the liquid-vapor interface. To account for the water adsorbed or condensed we perform in-situ gas chromatography analysis at different droplet lifetimes. Experimental results showed an increase in the amount of water condensed when increasing the relative humidity of the system as expected. In addition, for the same ambient temperature ethanol evaporation was enhanced by high relative humidity.

¹The authors acknowledge the support of WPI-I2CNER.

4:28PM D16.00008 Evaporation-triggered microdroplet nucleation and the four life phases of an evaporating Ouzo drop, HUANSHU TAN, University of Twente, CHRISTIAN DIDDENS, Eindhoven University of Technology, PENGYU LV, University of Twente, J. G. M. KUERTEN, Eindhoven University of Technology, XUEHUA ZHANG, Royal Melbourne Institute of Technology University, DETLEF LOHSE, University of Twente — Evaporating liquid droplets are omnipresent in nature and technology, such as in inkjet printing, coating, deposition of materials, medical diagnostics, agriculture, the food industry, cosmetics, or spills of liquids. Here we show that the evaporation of such ternary mixtures can trigger a phase transition and the nucleation of microdroplets of one of the components of the mixture. As a model system, we pick a sessile Ouzo droplet (as known from daily life) and reveal and theoretically explain its four life phases: In phase I, the spherical cap-shaped droplet remains transparent while the more volatile ethanol is evaporating, preferentially at the rim of the drop because of the singularity there. This leads to a local ethanol concentration reduction and correspondingly to oil droplet nucleation there. This is the beginning of phase II, in which oil microdroplets quickly nucleate in the whole drop, leading to its milky color that typifies the so-called Ouzo effect. Once all ethanol has evaporated, the drop, which now has a characteristic nonspherical cap shape, has become clear again, with a water drop sitting on an oil ring (phase III), finalizing the phase inversion. Finally, in phase IV, all water has evaporated, leaving behind a tiny spherical cap-shaped oil drop.

4:41PM D16.00009 A study of the evaporation of heterogeneous water droplets under active heating¹, MAXIM PISKUNOV, National Research Tomsk Polytechnic University, JEAN CLAUDE LEGROS, National Research Tomsk Polytechnic University, Université Libre de Bruxelles, PAVEL STRIZHAK, National Research Tomsk Polytechnic University — Using high-speed video registration tools with a sample rate of 10^2 – 10^4 frames per second (fps), we studied the patterns in the evaporation of water droplets containing 1 and 2 mm individual metallic inclusions in a high-temperature gas environment. The materials of choice for the inclusions were steels (AISI 1080 carbon steel and AISI type 316L stainless steel) and pure nickel. We established the lifetimes τ_h of the liquid droplets under study with a controlled increase in the gas environment temperature up to 900 K. We also considered the physical aspects behind the τ_h distribution in the experiments conducted and specified the conditions for more effective cooling of metallic inclusions. Following the experimental research findings, a method was devised for effective reactor vessel cooling to avoid a meltdown at a nuclear power plant.

¹The optimization of heat and mass transfer modes was performed within the framework of the strategic plan for the development of National Research Tomsk Polytechnic University as one of the world-leading universities.

4:54PM D16.00010 Fluid dynamics and deposit patterns in evaporating sessile drop containing microparticles: substrate heating and wettability effects, NAGESH D. PATIL¹, RAJNEESH BHARDWAJ², ATUL SHARMA³, Department of Mechanical Engineering, Indian Institute of Technology Bombay — The evaporation of sessile water drops containing colloidal microparticles is investigated on non-heated and heated hydrophilic and hydrophobic substrates. Time-varying drop shapes and temperatures of liquid-gas interface are recorded using high-speed and infrared camera, respectively. In heated case, infrared-thermography shows larger temperature gradient across the liquid-gas interface and recorded motion of the particles confirm Marangoni flow from the contact line to apex inside the drop. On non-heated hydrophilic substrates, a ring-like pattern forms, as reported extensively in the literature; while on heated hydrophilic substrates, a thin ring with an inner-deposit forms. On non-heated hydrophobic substrates, the contact line depins to form inner-deposit without ring; while on heated hydrophobic substrates, the contact line pins to form inner-deposit with thin ring. This pinning transition occurs due to the particles self-pinning in a stagnation region developed by the Marangoni flow near the contact line. This work gives fundamental insights on the thermal and wettability effects on internal fluid dynamics of the evaporating sessile drop and associated deposit shape, with applications in ink-jet printing and biosensors.

¹PhD Scholar

²Associate Professor

³Professor

Sunday, November 20, 2016 2:57PM - 5:07PM –
Session D17 Reacting Flows: Theory and Analysis D131 - Peter Hamlington, University of Colorado, Boulder

2:57PM D17.00001 Analysis of Turbulent Scales of Motion in Premixed Flames Using Structure Functions, PETER HAMILTON, SAMUEL WHITMAN, COLIN TOWERY, University of Colorado, Boulder, ALEXEI POLUDNENKO, Texas A&M University — Recently, multiscale turbulence-flame interactions in premixed reacting flows have been examined using both physical space and spectral approaches. However, there remains relatively little understanding of how turbulent scales of motion vary through the internal structure of the flame itself (i.e., through premixed flamelets). Such an analysis is made difficult by the inhomogeneity, small scale, and spatial locality of many premixed flames, particularly at high Damköhler and low Karlovitz numbers. Conditional structure functions provide a possible solution to this analysis challenge, and in this talk we present results from the calculation of structure functions using data from highly-resolved direct numerical simulations (DNS) of turbulent premixed flames. The high resolution of the DNS allows structure functions to be calculated normally and tangentially to the local flame surface, revealing the specific effects of the flame on turbulent scales of motion near the scale of the local flame width. Moreover, the conditional nature of the analysis allows the effects of different flame regions (e.g., the preheat and reaction zones) on turbulence to be isolated. The implications of these results for the theory and modeling of turbulent flame physics are outlined.

3:10PM D17.00002 Wavelet multi-resolution analysis of energy transfer in turbulent premixed flames, JEONGLAE KIM, MAXIME BASSENNE, Center for Turbulence Research, Stanford University, COLIN TOWERY, Department of Mechanical Engineering, University of Colorado Boulder, ALEXEI POLUDNENKO, Laboratories for Computational Physics and Fluid Dynamics, Naval Research Laboratory, PETER HAMILTON, Department of Mechanical Engineering, University of Colorado Boulder, MATTHIAS IHME, JAVIER URZAY, Center for Turbulence Research, Stanford University — Direct numerical simulations of turbulent premixed flames are examined using wavelet multi-resolution analyses (WMRA) as a diagnostics tool to evaluate the spatially localized inter-scale energy transfer in reacting flows. In non-reacting homogeneous-isotropic turbulence, the net energy transfer occurs from large to small scales on average, thus following the classical Kolmogorov energy cascade. However, in turbulent flames, our prior work suggests that thermal expansion leads to a small-scale pressure-work contribution that transfers energy in an inverse cascade on average, which has important consequences for LES modeling of reacting flows. The current study employs WMRA to investigate, simultaneously in physical and spectral spaces, the characteristics of this combustion-induced backscatter effect. The WMRA diagnostics provide spatial statistics of the spectra, scale-conditioned intermittency of velocity and vorticity, along with energy-transfer fluxes conditioned on the local progress variable.

3:23PM D17.00003 Three dimensional dynamic mode decomposition of premixed turbulent jet flames, TEMISTOCLE GRENGA, JONATHAN MACART, MICHAEL MUELLER, Princeton University — Analysis of turbulent combustion DNS data largely focuses on statistical analyses. However, turbulent combustion is highly unsteady and dynamic. In this work, Dynamic Mode Decomposition (DMD) will be explored as a tool for dynamic analysis of turbulent combustion DNS data, specifically a series of low Mach number spatially-evolving turbulent planar premixed hydrogen/air jet flames. DMD decomposes data into coherent modes with corresponding growth rates and oscillatory frequencies. The method identifies structures unbiased by energy so is particularly well suited to exploring dynamic processes at scales smaller than the largest, energy-containing scales of the flow and that may not be co-located in space and time. The focus of this work will be on both the physical insights that can potentially be derived from DMD modes and the computational issues associated with applying DMD to large three-dimensional DNS datasets.

3:36PM D17.00004 Lagrangian analysis of premixed turbulent combustion in hydrogen-air flames¹, RYAN DARRAGH, University of Colorado - Boulder, ALEXEI POLUDNENKO, Texas A&M University, PETER HAMILTON, University of Colorado - Boulder — Lagrangian analysis has long been a tool used to analyze non-reacting turbulent flows, and has recently gained attention in the reacting flow and combustion communities. The approach itself allows one to separate local molecular effects, such as those due to reactions or diffusion, from turbulent advective effects along fluid pathlines, or trajectories. Accurate calculation of these trajectories can, however, be rather difficult due to the chaotic nature of turbulent flows and the added complexity of reactions. In order to determine resolution requirements and verify the numerical algorithm, extensive tests are described in this talk for prescribed steady, unsteady, and chaotic flows, as well as for direct numerical simulations (DNS) of non-reacting homogeneous isotropic turbulence. The Lagrangian analysis is then applied to DNS of premixed hydrogen-air flames at two different turbulence intensities for both single- and multi-step chemical mechanisms. Non-monotonic temperature and fuel-mass fraction evolutions are found to exist along trajectories passing through the flame brush. Such non-monotonicity is shown to be due to molecular diffusion resulting from large spatial gradients created by turbulent advection.

¹This work was supported by the Air Force Office of Scientific Research (AFOSR) under Award No. FA9550-14-1-0273, and the Department of Defense (DoD) High Performance Computing Modernization Program (HPCMP) under a Frontier project award.

3:49PM D17.00005 Lyapunov spectrum in turbulent combustion, MALIK HASSANALY, VENKAT RAMAN, Univ of Michigan - Ann Arbor — Transient flame evolution is an important flow problem for many practical applications (for example high-altitude relight, ignition in internal combustion engines, unstart in scramjets). Current approaches to combustion modeling utilize assumptions that are valid mainly for statistically stationary processes. In order to understand the transient problem, a dynamic systems approach is followed here. The propagation of a flame in a turbulent channel flow is used as a canonical turbulent combustion system and is analyzed with the Lyapunov theory. In particular, the Lyapunov spectrum for this flow is computed using multiple coordinated simulations. For a range of flow conditions, dimensionality of the state-space is determined. It is shown that the internal structure of the flame plays a critical role in determining the response of the system to perturbations in the flow.

4:02PM D17.00006 Optimal stretching of fluid for enhancing reaction growth, THOMAS NEVINS, DOUGLAS KELLEY, Univ of Rochester — When a biological or chemical scalar grows in flowing fluid, the resulting reacted region is dependent on both the details of the flow, and the reaction kinetics. We simultaneously film reaction state and flow in a laboratory model of reactive mixing in order to examine reactive mixing in physical, time-dependent flows. Using the excitable Belousov-Zhabotinsky (BZ) reaction, we find an optimal stretching range in which the flow enhances reaction, but larger stretching causes reaction blowout. We observe the transition from flow mostly helping to mostly blowout is not associated with the transition to turbulence, and that stretching fields (closely related to finite-time Lyapunov exponents) inside the optimal range appear to have a large effect on reaction growth rate locally. We also present estimates of the optimal stretching for BZ, and hypothesize that it is a feature exclusive to excitable reactions.

4:15PM D17.00007 Dissipation Element Analysis of Reacting- and Non-Reacting

Flows. , DOMINIK DENKER, JONAS BOSCHUNG, FABIAN HENNIG, HEINZ PITSCHE, RWTH Aachen University — Dissipation element analysis is a tried and tested method for analyzing scalar field in turbulent flows. Dissipation elements are defined as an ensemble of grid point whose gradient trajectories reach the same extremal points. Therefore, the scalar field can be compartmentalized in monotonous space filling regions. Dissipation elements can be described by two parameters, namely the Euclidean distance between their extremal points and their scalar difference in these points. The joint probability density function of these two parameters is expected to suffice for a statistical reconstruction of the scalar field. In addition, normalized dissipation element statistics show a remarkable invariance towards changes in Reynolds numbers. Dissipation element statistics of the passive scalar and the turbulent kinetic energy are compared for different flow configurations including reacting and non-reacting turbulent flows. Furthermore, the Reynolds number scaling of the dissipation element parameters is investigated.

4:28PM D17.00008 Reaction front barriers in time aperiodic fluid flows. , RORY LOCKE,

KEVIN MITCHELL, University of California-Merced — Many chemical and biological systems can be characterized by the propagation of a front that separates different phases or species. One approach to formalizing a general theory is to apply frameworks developed in nonlinear dynamics. It has been shown that invariant manifolds form barriers to passive transport in time-dependent or time-periodic fluid flows. More recently, analogous manifolds termed burning-invariant-manifolds (BIMs), have been shown to form one-sided barriers to reaction fronts in advection-reaction-diffusion (ARD) systems. To model more realistic time-aperiodic systems, recent theoretical work has suggested that similar one-sided barriers, termed burning Lagrangian coherent structures (bLCSs), exist for fluid velocity data prescribed over a finite time interval. In this presentation, we use a stochastic wind to generate time dependence in a double-vortex channel flow and demonstrate the (locally) most attracting or repelling curves are the bLCSs.

4:41PM D17.00009 Attached and lifted diffusion flames in a mixing layer , MOSHE

MATALON, Univ of Illinois at Urbana-Champaign, ZHANBIN LU, Shanghai University — Many practical combustion devices are concerned with the stabilization of diffusion flames that are formed by injecting gaseous fuels into a co-flowing stream containing an oxidizer. A primary concern of these configurations is the attachment and lift-off characteristics of the diffusion flame relative to the rim of the injector. In such circumstances, the edge of the flame, which possesses a distinct structure that combines characteristics of both premixed and diffusion flames, is found to play a crucial role in determining the stabilization of the diffusion flame. In this study, we examine the effect of streams of unequal flow rates on the structural and dynamical properties of the edge flame. We show that, depending on the stoichiometric conditions and the diffusive properties of the fuel and oxidizer, the diffusion flame may either be attached to the rim of the injector, lifted and stabilized at a downstream equilibrium position, or blown off by the flow. Under certain conditions the diffusion flame may undergo spontaneous oscillations, whereby the edge of the flame exhibits a back and forth motion along a direction that coincides with the diffusion flame surface.

4:54PM D17.00010 Experimental Vortex Identification and Characterization in Reacting Jets in Crossflow , VEDANTH NAIR, BEN EMERSON, TIMOTHY LIEUWEN, Georgia Inst of Tech — Reacting jets in crossflow (JICF) is an important canonical flow field in combustion problems where there is strong coupling between heat release and the evolution of vortical structures. We use vortex identification studies to experimentally characterize the spatial evolution of vortex dynamics in a reacting JICF. A vortex identification algorithm was designed to operate on particle image velocimetry (PIV) data and its raw Mie scattering images. The algorithm uses the velocity fields to obtain comparisons between the strain rate and the rotation rate. Additionally, the algorithm uses the raw Mie scattering data to identify regions where the high acceleration at vortex cores has centrifuged seeding particles out of the vortex cores. Together, these methods are used to estimate the vortex location and circulation. Analysis was done on 10 kHz PIV data from a reacting JICF experiment, and the resulting vortex trajectory, and growth rate statistics are presented. Results are compared between non-reacting JICF and reacting studies performed with different jet density ratios and different levels of acoustic forcing. We observed how the density ratio, the frequency and amplitude of the acoustic forcing affected the vortex characteristics and growth rate.

Sunday, November 20, 2016 2:57PM - 4:54PM –

Session D18 Flow Instability: Multiphase Flow D135 - Khaled Sallam, Oklahoma State University

2:57PM D18.00001 Supersonic Injection of Aerated Liquid Jet. , ABHIJIT CHOURDHARI, KHALED

SALLAM, Oklahoma State University — A computational study of the exit flow of an aerated two-dimensional jet from an under-expanded supersonic nozzle is presented. The liquid sheet is operating within the annular flow regime and the study is motivated by the application of supersonic nozzles in air-breathing propulsion systems, e.g. scramjet engines, ramjet engines and afterburners. The simulation was conducted using VOF model and SST $k-\omega$ turbulence model. The test conditions included: jet exit of 1 mm and mass flow rate of 1.8 kg/s. The results show that air reaches transonic condition at the injector exit due to the Fanno flow effects in the injector passage. The aerated liquid jet is alternately expanded by Prandtl-Meyer expansion fan and compressed by oblique shock waves due to the difference between the back (chamber) pressure and the flow pressure. The process then repeats itself and shock (Mach) diamonds are formed at downstream of injector exit similar to those typical of exhaust plumes of propulsion system. The present results, however, indicate that the flow field of supersonic aerated liquid jet is different from supersonic gas jets due to the effects of water evaporation from the liquid sheet. The contours of the Mach number, static pressure of both cases are compared to the theory of gas dynamics.

3:10PM D18.00002 Stability of two-layer Couette flow with application to drag reduction.¹ , ALIREZA MOHAMMADI, Princeton University, ALEXANDER J. SMITS, Princeton University, Monash University —

We consider the linear stability of flows composed of two superposed fluids in Couette flow in order to improve our understanding of the longevity and performance of superhydrophobic surfaces (SHS) or liquid-infused surfaces (LIS) which are important for drag reduction. Here, we assume that the fluids are immiscible, incompressible, and Newtonian with constant properties. Single-fluid Couette flow is known to be linearly stable for any Reynolds number. However, inclusion of the second layer of fluid enriches the problem and introduces five new parameters: viscosity ratio, density ratio, thickness ratio, Froude number and Weber number. Two kinds of instability can appear: an unstable interfacial mode, and a Tollmein-Schlichting mode. In this work we parametrically study the flow stability with specific emphasis on the effects of viscosity ratio, interfacial tension, and thickness ratio.

¹Supported by the Natural Sciences and Engineering Research Council of Canada (NSERC) and the Office of Naval Research (ONR) through MURI Grants N00014-12-1-0875 and N00014-12-1-0962 (Program Manager Dr. Ki-Han Kim).

3:23PM D18.00003 Mechanisms, role of vorticity, and time scales for planar liquid sheet breakup, ARASH ZANDIAN, WILLIAM SIRIGNANO, University of California, Irvine, FAZLE HUSSAIN, Texas Tech University

— The 3D, temporal instabilities on a planar liquid sheet are studied using DNS with level-set and VoF surface tracking methods. λ_2 contours relate the vorticity to the surface dynamics. The breakup character depends on the Ohnesorge number (Oh). At high Oh , hairpin vortices form on the braid and overlap with the lobe hairpins, thinning the lobes, which puncture creating holes and bridges. The bridges break, creating ligaments that stretch and break into droplets by capillary action. At low Oh , lobe stretching and thinning is hindered by high surface tension and splitting of the original Kelvin-Helmholtz vortex, preventing early hole formation. Corrugations form on the lobe edges, influenced by the split vortices, and stretch to form ligaments. Both mechanisms are present in a transition region that shifts in Oh values based on the liquid/gas density ratio. Different characteristic times exist for the hole formation and the lobe and ligament stretching, related to surface tension and liquid viscosity, respectively. In the transition region, both times are of the same order. Streamwise vorticity triggers the 3D instabilities. Vorticity stretching and baroclinicity dominate, while the spanwise and cross-flow vorticity tilting are less important early in the breakup.

3:36PM D18.00004 Temporal length-scale cascade and expansion rate on planar liquid jet instability, WILLIAM SIRIGNANO, ARASH ZANDIAN, University of California, Irvine, FAZLE HUSSAIN, Texas Tech University

— Using the local radius of curvature of the surface and the local transverse dimension of the two-phase (i.e., spray) domain as length scales, we obtained two PDFs over a wide range of length-scales at different times and for different Reynolds and Weber (We) numbers. The PDFs were developed via post-processing of DNS Navier-Stokes results for a 3D planar liquid sheet segment with level-set and Volume-of-Fluid surface tracking, giving better statistical data for the length scales compared to the former methods. The radius PDF shows that, with increasing We , the average radius of curvature decreases, number of small droplets increases, and cascade occurs at a faster rate. In time, the mean of the radius PDF decreases while the rms increases. The other PDF represents the spray expansion in a more realistic and meaningful form, showing that the spray angle is larger at higher We and density-ratios. Both the mean and the rms of the spray-size PDF increase with time. The PDFs also track the transitions between symmetric and anti-symmetric modes.

3:49PM D18.00005 Stability of three-layered core-annular flow¹, DIPIN PILLAI, Univ of Florida

- Gainesville, SUBRAMANIAM PUSHPAVANAM, T SUNDARARAJAN, Indian Institute of Technology Madras, India — Stability of a three-layered core-annular flow is analyzed using the method of modal linear stability analysis. A temporal analysis shows that the flow becomes unstable to two modes of instability when inertial effects are negligible. An energy budget analysis reveals that these two modes correspond to capillary instability associated with each fluid-fluid interface. With an increase in Reynolds number, the system exhibits additional Reynolds stress modes of instabilities. These modes correspond to the Tollmien-Schlichting type of waves associated with high Reynolds number shear flows, and are considered precursor to transition to turbulence. An investigation of the parameter space reveals that the system may simultaneously show up to 5 distinct modes of instability, viz., the two capillary modes at each interface and three Reynolds stress modes in the bulk of each phase. In addition, a spatio-temporal analysis shows that the Reynolds stress modes are always convectively unstable whereas the capillary modes may undergo a transition from convective to absolute instability with decrease in Weber number. To obtain encapsulated droplets in experiments, the operating parameters must be chosen such that the system lies in the regime of convective instability.

¹MHRD-Govt of India, NSF 0968313

4:02PM D18.00006 A 3D Computational Study on the Air-Blast Atomization of a Planar Liquid Layer, ROBERT CHIODI, OLIVIER DESJARDINS, Cornell Univ — The air-blast atomization of a planar liquid layer is a complex fluid phenomenon involving the destabilization of a low speed liquid layer by a high speed gas coflow. While progress has been made in recent years on understanding the instability of the liquid surface, it remains difficult to accurately predict using stability analysis and requires special expertise and equipment to perform thorough experiments. Simulations provide an excellent way to conduct parametric studies to determine the effect of splitter plate geometry and momentum flux ratio on the frequency and wavelengths of instability, however, they are extremely difficult due to the high density ratio and large range of length and time scales present in the flow. Using an accurate conservative level set method in conjunction with a newly reformulated reinitialization equation, we perform 3D simulations of the air-blast atomization of a planar liquid layer and compare them to experiments. We then go on to explore the role momentum flux ratio plays in the longitudinal and transverse wavelengths of instability.

4:15PM D18.00007 Taylor-Couette flow instabilities in neutrally-buoyant suspensions, MADHU MAJJI, SANJOY BANERJEE, JEFFREY F. MORRIS, City College of CUNY — Experimentally-determined instabilities and flow states of a neutrally-buoyant suspension are described. The flow is studied in a concentric-cylinder device with inner-to-outer cylinder ratio of 0.877 with inner cylinder rotating and outer stationary. The cylinder length to annular gap ratio is 20, while the gap to particle size ratio is approximately 30, for spherical particles of 250 μm diameter. Using a slowly increasing or decreasing Re ramp, the flow agrees with all expectations for the pure fluid, while a slowly decreasing (quasi-static) ramp is used for the suspension flow, which is found to be unstable at lower Reynolds number Re (based on the effective viscosity) than pure fluid, and exhibits spiraling and ribbon states not found for a pure fluid with only inner cylinder rotating. Strikingly, the suspension at solid fraction $\phi \geq 0.05$ goes unstable first to a nonaxisymmetric state rather than axisymmetric Taylor vortices. At $0.1 \leq \phi \leq 0.2$, the flow exhibits numerous states during quasi-static ramping of Re , while for $\phi = 0.3$, the base state Couette flow gives way to wavy spirals (WS) at $Re \approx 80$ and exhibits only the WS state up to $Re = 150$. Transient behavior on sudden change of Re and particle tracking will also be presented.

4:28PM D18.00008 Liquid-liquid flow past a bluff body¹, KYEONG H. PARK, M.I.I. ZAINAL ABIDIN, PANAGIOTA ANGELI², Department of Chemical Engineering, University College London, Torrington Place, London WC1E 7JE, UK, LYSE KAHOUADJI, ZHIHUA XIE, OMAR K. MATAR, Department of Chemical Engineering, Imperial College London, South Kensington Campus, London SW7 2AZ, UK, CHRISTOPHER C. PAIN, Applied Modelling and Computation Group, Department of Earth Science and Engineering, Imperial College London, South Kensington Campus, London SW7 2AZ — The generation of instabilities behind a bluff body bounded by a pipe wall and its effects on flow pattern transitions from separated to dispersed oil-water flows are studied. A cylindrical bluff body is located in the water phase and the transverse direction of the flow. Investigations are conducted for flow rates that result in stratified flow in the absence of the bluff body. A high-speed camera is used to track the interfacial waves while the velocity profile in the water phase is determined by PIV. Numerical studies on single-phase flow assist in designing new bluff bodies. The results showed that the choice of the bluff body and its location generated vortices with frequencies similar to unbounded flows that corresponded to Strouhal number of 0.2. In two-phase flows, the bluff body generates waves with frequencies similar to the von Kármán vortices in the water phase behind the cylinder. The formation of the waves depended on the distance of the bluff body from the oil-water interface.

¹Project funded under the UK Engineering and Physical Sciences Research Council (EPSRC) Programme Grant MEMPHIS

²corresponding author

4:41PM D18.00009 Flow behaviour and transitions in surfactant-laden gas-liquid vertical flows¹ , IVAN ZADRAZIL, SOUROJEET CHAKRABORTY, OMAR MATAR, CHRISTOS MARKIDES, Imperial College London

— The aim of this work is to elucidate the effect of surfactant additives on vertical gas-liquid counter-current pipe flows. Two experimental campaigns were undertaken, one with water and one with a light oil (Exxsol D80) as the liquid phase; in both cases air was used as the gaseous phase. Suitable surfactants were added to the liquid phase up to the critical micelle concentration (CMC); measurements in the absence of additives were also taken, for benchmarking. The experiments were performed in a 32-mm bore and 5-m long vertical pipe, over a range of superficial velocities (liquid: 1 to 7 m/s, gas: 1 to 44 m/s). High-speed axial- and side-view imaging was performed at different lengths along the pipe, together with pressure drop measurements. Flow regime maps were then obtained describing the observed flow behaviour and related phenomena, i.e., downwards/upwards annular flow, flooding, bridging, gas/liquid entrainment, oscillatory film flow, standing waves, climbing films, churn flow and dryout. Comparisons of the air-water and oil-water results will be presented and discussed, along with the role of the surfactants in affecting overall and detailed flow behaviour and transitions; in particular, a possible mechanism underlying the phenomenon of flooding will be presented.

¹EPSRC UK Programme Grant EP/K003976/1

Sunday, November 20, 2016 2:57PM - 5:07PM – Session D19 Bio: Swimming D136 - David Daily, Naval Undersea Warfare Centre

2:57PM D19.00001 Imaging Techniques for Dense 3D reconstruction of Swimming Aquatic Life using Multi-view Stereo , DAVID DAILY, JILLIAN KISER, SARAH MCQUEEN, Naval Undersea Warfare Center

— Understanding the movement characteristics of how various species of fish swim is an important step to uncovering how they propel themselves through the water. Previous methods have focused on profile capture methods or sparse 3D manual feature point tracking. This research uses an array of 30 cameras to automatically track hundreds of points on a fish as they swim in 3D using multi-view stereo. Blacktip sharks, sting rays, puffer fish, turtles and more were imaged in collaboration with the National Aquarium in Baltimore, Maryland using the multi-view stereo technique. The processes for data collection, camera synchronization, feature point extraction, 3D reconstruction, 3D alignment, biological considerations, and lessons learned will be presented. Preliminary results of the 3D reconstructions will be shown and future research into mathematically characterizing various bio-locomotive maneuvers will be discussed.

3:10PM D19.00002 Fin-Body Interaction and its Hydrodynamic Benefits in Fish's Steady Swimming¹ , GENG LIU, YAN REN, HAIBO DONG, University of Virginia, GEORGE LAUDER, Harvard University — In many past studies on fish swimming, the hydrodynamics of fish caudal fins were investigated separately. However, fish body inevitably interacts with the caudal fin since the fin flaps in the wake of the body during swimming. In this work, an integrated experimental and computational approach has been used to investigate hydrodynamic performance improvement and the vortex dynamics associated with the fin-body interactions of a jack fish in steady swimming. Realistic 3D jack fish geometry and the undulatory kinematics are reconstructed based on the output of a high-speed photogrammetry system. Hydrodynamic performance and wake structures are simulated by an in-house immersed-boundary-method flow solver. It is found that the body-fin interactions enhance the thrust production of the caudal fin by more than 30% compared to that produced by an isolated caudal fin. Further analysis on the vortex dynamics has shown that the vortices shed from the posterior part of the fish body are captured by the leading edge portion of the caudal fin. This further enhances the strength of the leading-edge vortex attaching to the caudal fin and results in larger thrust production. This work reveals a potential performance enhancement mechanism in fish's steady swimming.

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

3:23PM D19.00003 Simulating underwater propulsion using an immersed boundary method based open-source solver¹ , UTKU SENTURK, Ege University, ARMAN HEMMATI, ALEXANDER J. SMITS, Princeton University — The performance of a newly developed Immersed Boundary Method (IBM) incorporated into a finite volume solver is examined using foam-extend-3.2. IBM uses a discrete forcing approach based on the weighted least squares interpolation to preserve the sharpness of the boundary, which decreases the computational complexity of the problem. Initially, four case studies with gradually increasing complexities are considered to verify the accuracy of the IBM approach. These include the flow past 2D stationary and transversely oscillating cylinders and 3D wake of stationary and pitching flat plates with aspect ratio 1.0 at Re=2000. The primary objective of this study, which is pursued by an ongoing simulation of the wake formed behind a pitching deformable 3D flat plate, is to investigate the underwater locomotion of a fish at Re=10000. The results of the IBM based solver are compared to the experimental results, which suggest that the force computations are accurate in general. Spurious oscillations in the forces are observed for problems with moving bodies which change based on spatial and temporal grid resolutions. Although it still has the full advantage of the main code features, the IBM-based solver in foam-extend-3.2 requires further development to be exploited for complex grids.

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

3:36PM D19.00004 Wave number effect on the neuromechanical phase characteristics of fish undulatory locomotion , JIALEI SONG, YANG DING, Beijing Computational Science Research Center, YONG ZHONG, RUXU DU, The Chinese University of Hong Kong — For animals with undulatory locomotion, it has been discovered that “neuromechanical phase lags” (NPL) is commonly utilized. That is, the wave of the muscle activation propagates faster than the wave of body bending, leading to an advancing phase of activation relative to the curvature forward the tail. Even though several multi parameter neuromechanical models have reproduced this phenomenon, but due to the simplification of the model, the origin of the NPL is difficult to identify. By incorporating accurate hydrodynamic and inertial effect, we tried to build a model of high fidelity to describe the dynamics of undulatory fish swimming. The hydrodynamic torque is obtained by the accurate DNS simulation. Meanwhile, the inertial torque is introduced by incorporating the reasonable density distribution and detailed undulatory motion. In our study, we studied cases with three different wave numbers on the fish body, with the swimming pattern ranges from anguilliform to carangiform. The results show different muscle actuation patterns with different wave number on body. This study might provide a beneficial guidance on the future fish-like robot design.

3:49PM D19.00005 Evolutionary Optimization of Non-Continuous and Non-Sinusoidal Gaits of a Self-Propelled Swimmer¹ , FATMA AYANCIK, EMRE AKOZ, KEITH MOORED, Lehigh University

— Animals propel themselves through the oceans with a wide variety of swimming gaits. However, it is typically assumed that biological propulsion is achieved by using continuous, sinusoidal motions. Yet, animals have been observed using non-continuous or intermittent swimming gaits and at many times non-sinusoidal motions. Through the use of an evolutionary algorithm, optimal swimming gaits that can be both nonsinusoidal and intermittent are determined. Both the non-dimensional cost of transport and swimming speed are optimized for a virtual body combined with a two-dimensional self-propelled pitching and heaving foil within a boundary element method numerical framework. Nonsinusoidal motions are varied from a triangle-wave to a square-wave motion and the intermittency of the gait is varied by changing the duty cycle of the active phase to the coasting phase during swimming. Both pure pitching, and combined heaving and pitching motions are examined. The Pareto front of optimal solutions is investigated for trends in the optimally efficient swimming gait as the swimming speed is increased. The variation in the wake structures produced by optimally efficient swimmers is probed.

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

4:02PM D19.00006 Vortical structures responsible for delayed stall in an idealized humpback whale flipper model¹ , HEESU KIM, Seoul National University, JOOHA KIM, UNIST, HAECHON CHOI, Seoul National University

— In this study, we investigate how the tubercles on the leading edge of an idealized humpback whale flipper model delay the stall. Oil-surface visualization is performed to see the surface flow pattern on the suction surface, and PIV is conducted in several streamwise and crossflow planes at different attack angles (α). Without tubercles, leading edge separation first occurs near the tip region and progresses inboard with increasing α . With tubercles, however, two types of vortical motions are observed at the mid-span. The first is streamwise vortex arrays which are dominant at $\alpha \leq 9^\circ$, and they are observed downstream of small separation bubbles near the leading edge. The second is asymmetric counter-rotating streamwise vortex pairs that become dominant at $\alpha > 9^\circ$, and these structures appear near the trailing edge. These two types of vortical motions delay flow separation at the peak regions of the mid-span, eliminating the spanwise stall progression and resulting in delayed stall. At $\alpha = 16^\circ$ at which the tubercle model stalls, a large-scale streamwise vortex is originated from flow separation near the root region. This structure delays flow separation at the mid-span, leading to higher lift coefficient.

¹Supported by NRF-2014M3C1B1033848

4:15PM D19.00007 Cetacean Swimming with Prosthetic Limbs¹ , AYODEJI BODE-OKE, YAN REN, HAIBO DONG, University of Virginia, FRANK FISH, West Chester University

— During entanglement in fishing gear, dolphins can suffer abrasions and amputations of flukes and fins. As a result, if the dolphin survives the ordeal, swimming performance is altered. Current rehabilitation techniques is the use of prosthesis to regain swimming ability. In this work, analyses are focused on two dolphins with locomotive impairment; Winter (currently living in Clearwater Marine Aquarium in Florida) and Fuji (lived in Okinawa Churaumi Aquarium in Japan). Fuji lost about 75% of its fluke surface to necrosis (death of cells) and Winter lost its tail due to amputation. Both dolphins are aided by prosthetic tails that mimic the shape of a real dolphin tail. Using 3D surface reconstruction techniques and a high fidelity Computational Fluid Dynamics (CFD) flow solver, we were able to elucidate the kinematics and hydrodynamics and fluke deformation of these swimmers to clarify the effectiveness of prostheses in helping the dolphins regain their swimming ability. Associated with the performance, we identified distinct features in the wake structures that can explain this gap in the performance compared to a healthy dolphin.

¹This work was supported by ONR MURI Grant Number N00014-14-1-0533

4:28PM D19.00008 Sensing and exploitation of vortices for a schooling fish , AMY GAO, MIT, AUDREY MAERTENS, EPFL, MICHAEL TRIANTAFYLLOU, MIT

— The question of whether fish are capable of actively sensing and using individual vortices while schooling has long been debated. Prior research has shown that fish can gain a hydrodynamic benefit when swimming in the wake of another fish. However, it remains unclear if lateral line feedback is necessary, and if so, how a fish may adjust its motion to maximize its energy savings. To begin to address this, we study though numerical simulations the hydrodynamic interactions between two fish swimming in tandem, focusing on the interaction of individual vortices with the following fish. Using a potential flow model, we show that the pressure sensed by the following fish can be captured with a low number of states, which provide information that allows the fish to locate near-field vortices and phase its undulating motion accordingly. We will discuss how vortex interactions along the fish can be beneficial, the signals they induce, and which strategies a fish may use to save the most energy.

4:41PM D19.00009 Individual nectophore kinematics during multi-jet swimming by the siphonophore *Nanomia bijuga*¹ , KELLY SUTHERLAND, University of Oregon, BRAD GEMMELL, University of South Florida, SEAN COLIN, JOHN COSTELLO, Marine Biological Laboratory

— The siphonophore *N. bijuga* is a colonial marine organism comprised of multiple swimming units that coordinate forward and reverse swimming as well as maneuvering. Though colonies can be cms long, individual swimming units (nectophores) are mms in length. To better understand swimming kinematics and jet-wake properties at the scale of individual nectophores, we collected high speed microvideography and micro particle image velocimetry at the nectophore scale. Nectophores exhibited high pulse frequencies (3 Hz) and a rapid refill time that was roughly equal to the jet time. Forward and reverse swimming were achieved using a maneuverable velum with a triangular opening (jet nozzle) that directed flow forward or backward. Detailed velum kinematics can be applied to the design of multijet underwater vehicles with varying nozzle geometries and cross sectional areas for control of exit flow properties.

¹Sloan Foundation

4:54PM D19.00010 Effects of varying inter-limb spacing to limb length ratio in metachronal swimming , HONG KUAN LAI, RACHAEL MERKEL, ARVIND SANTHANAKRISHNAN, Oklahoma State University

— Crustaceans such as shrimp, krill and crayfish swim by rhythmic paddling of four to five pairs of closely spaced limbs. Each pair is phase-shifted in time relative to the neighboring pair, resulting in a metachronal wave that travels in the direction of animal motion. The broad goal of this study is to investigate how the mechanical design of the swimming limbs affect scalability of metachronal swimming in terms of limb-based Reynolds number (Re). A scaled robotic model of metachronal paddling was developed, consisting of four pairs of hinged acrylic plates actuated using stepper motors that were immersed in a rectangular tank containing water-glycerin fluid medium. 2D PIV measurements show that the propulsive jets transition from being primarily horizontal (thrust-producing direction) at Re of order 10 to angled vertically at Re of order 100. The ratio of inter-limb spacing to limb length among metachronal swimming organisms ranges between 0.2 to 0.65 (Murphy et al., Mar. Biol. 158, 2011). 2D PIV will be used to examine the jets generated between adjacent limbs for varying inter-limb spacing to limb length ratios. The effect of increasing this ratio to beyond the biologically observed range will be discussed.

Sunday, November 20, 2016 2:57PM - 5:07PM –

Session D20 Bio: Bacteria Swimming D137-138 - Kenneth Breuer, Brown University

2:57PM D20.00001 The swimming behavior of flagellated bacteria in viscous and viscoelastic media¹

ZIJIE QU, Brown University, RENE HENDERIKX, Eindhoven University of Technology, KENNETH BREUER, Brown University — The motility of bacteria *E.coli* in viscous and viscoelastic fluids has been widely studied although full understanding remains elusive. The swimming mode of wild-type *E.coli* is well-described by a run-and-tumble sequence in which periods of straight swimming at a constant speed are randomly interrupted by a tumble, defined as a sudden change of direction with a very low speed. Using a tracking microscope, we follow cells for extended periods of time and find that the swimming behavior can be more complex, and can include a wider variety of behaviors including a “slow random walk” in which the cells move at relatively low speed without the characteristic run. Significant variation between individual cells is observed, and furthermore, a single cell can change its motility during the course of a tracking event. Changing the viscosity and viscoelasticity of the swimming media also has profound effects on the average swimming speed and run-tumble nature of the cell motility, including changing the distribution, duration of tumbling and slow random walk events. The reasons for these changes are explained using a Purcell-style resistive force model for the cell and flagellar behavior as well as model for the changes in flagellar bundling in different fluid viscosities.

¹National Science Foundation

3:10PM D20.00002 Instability of hooks during bacterial flagellar swimming, MEHDI

JABBARZADEH, HENRY C. FU, Department of Mechanical Engineering, University of Utah, HENRY FU TEAM — In bacteria, a flexible hook transmits torque from the rotary motor at the cell body to the flagellum. Previously, the hook has been modeled as a Kirchhoff rod between the cell body and rotating flagellum. To study effects of the hook's flexibility on the bacteria's swimming speed and trajectory for wide range hook stiffnesses and flagellum configurations, we develop an efficient simplified spring model for the hook by linearizing the Kirchhoff rod. We treat the hydrodynamics of the cell body and helical flagellum using resistance matrices calculated by the method of regularized Stokeslets. We investigate flagellar and swimming dynamics for a range of hook flexibilities and flagellar orientations relative to the cell body and compare the results to models without hook flexibility. We investigate in detail parameters corresponding to *E. coli* and *Vibrio alginolyticus*. Generally, the flagellum changes orientation relative to the cell body, undergoing an orbit with the period of the motor rotation. We find that as the hook stiffness decreases, steady-state orbits of the flagellum first become unstable before the hook buckles, which may suggest a new mechanism of flick initiation in run-reverse-flick motility. We also find that for some parameter ranges, there are multiple stable steady state orbits, which may have implications for the tumbling and turning of bacteria.

3:23PM D20.00003 Polymorphic transformation of helical flagella of bacteria¹

SOOKKYUNG LIM, University of Cincinnati, HOWARD BERG COLLABORATION, WILLIAM KO COLLABORATION, YONGSAM KIM COLLABORATION, WANHO LEE COLLABORATION, CHARLES PESKIN COLLABORATION — Bacteria such as *E. coli* swim in an aqueous environment by utilizing the rotation of flagellar motors and alternate two modes of motility, *runs* and *tumbles*. Runs are steady forward swimming driven by bundles of flagellar filaments whose motors are turning CCW; tumbles involve a reorientation of the direction of swimming triggered by motor reversals. During tumbling, the helical flagellum undergoes polymorphic transformations, which is a local change in helical pitch, helical radius, and handedness. In this work, we investigate the underlying mechanism of structural conformation and how this polymorphic transition plays a role in bacterial swimming.

¹National Science Foundation

3:36PM D20.00004 Visualization of bacterial flagella dynamics in a viscous shear flow¹

JAMEL ALI, Drexel University, MINJUN KIM, Southern Methodist University — We report on the dynamics of tethered bacterial flagella in an applied viscous shear flow and analyze their behavior using image processing. Flagellin proteins were repolymerized into flagellar filaments functionalized with biotin at their proximal end, and allowed to self-assemble within a micro channel coated with streptavidin. It was observed that all attached flagellar filaments aligned with the steady shear flow of various polymeric solutions. Furthermore it was observed that many of the filaments were stretched, and at elevated flow rates began to undergo polymorphic transformations, which were initiated at one end of the flagellum. When undergoing a change to a different helical form the flagellum was observed to transform to an oppositely handed helix, as to counteract the viscous torque imparted by the shear flow. It was also observed that some flagellar filaments did not undergo polymorphic transformations, but rotated about their helical axis. The rate of this rotation appears to be a function of the applied flow rate. These results expand on previous experimental work and aid in the development of a novel platform that harnesses the autonomic response of a ‘forest’ of bacterial flagella for engineering applications.

¹This work was funded by NSF Grant CMMI-1000255, KEIT MOTIE Grant No. 10052980, and with Government support under and awarded by DoD, Air Force Office of Scientific Research, National Defense Science and Engineering Graduate (NDSEG) Fellowship, 32 CFR 168a

3:49PM D20.00005 Barriers for active transport of bacteria in a microfluidic flow¹

PAYTON JOHNSON, MINH DOAN, Bucknell University, KEVIN MITCHELL, UC-Merced, TOM SOLOMON, Bucknell University — We present experiments on the motion of swimming bacteria in a laminar, hyperbolic flow in a microfluidic cross channel. The bacteria used are a genetically-mutated “smooth swimming”² *bacillus subtilis*. The movement of bacteria in the flow is bounded by *swimming invariant manifolds* (SWIMs) that act as one-way barriers. The SWIMs are similar to ‘burning invariant manifolds’³ that act as one-way barriers that impede the motion of reaction fronts in a fluid flow. We explore the structure and bounding behavior of the SWIMs and how their separation from the passive manifolds depends on the bacteria swimming speed, normalized by the characteristic fluid speeds.

¹Supported by NSF Grant DMR-1361881.

²R. Rusconi, J.S. Guasto and R. Stocker, Nature Physics **10**, 212 (2014).

³J. Mahoney, D. Bargteil, M. Kingsbury, K. Mitchell and T. Solomon, Europhys. Lett. **98**, 44005 (2012).

4:02PM D20.00006 Bacterial Trapping in Porous Media Flows¹, AMIN DEHKHARGHANI, NICOLAS WAISBORD, Tufts University, JÖRN DUNKEL, MIT, JEFFREY GUASTO, Tufts University — Swimming bacteria inhabit heterogeneous, microstructured environments that are often characterized by complex, ambient flows. Understanding the physical mechanisms underlying cell transport in these systems is key to controlling important processes such as bioremediation in porous soils and infections in human tissues.

We study the transport of swimming bacteria (*Bacillus subtilis*) in quasi-two-dimensional porous microfluidic channels with a range of periodic microstructures and flow strengths. Measured cell trajectories and the local cell number density reveal the formation of filamentous cell concentration patterns within the porous structures. The local cell densification is maximized at shear rates in the range $1\text{--}10 \text{ s}^{-1}$, but widely varies with pore geometry and flow topology. Experimental observations are complemented by Langevin simulations to demonstrate that the filamentous patterns result from a coupling of bacterial motility to the complex flow fields via Jeffery orbits, which effectively 'trap' the bacteria on streamlines. The resulting microscopic heterogeneity observed here suppresses bacterial transport and likely has implications for both mixing and cell nutrient uptake in porous media flows.

¹NSF CBET-1511340

4:15PM D20.00007 Bacteria rolling: motilities of rosette colonies in *Caulobacter crescentus*, YU ZENG, BIN LIU, Univ of California - Merced — The aquatic bacterium *Caulobacter crescentus* has two life cycle stages with distinct motilities: freely swimming swarmer cells and immotile stalked cells. Here, we show a new type of movement performed by freely suspended rosettes, spontaneous aggregates of stalked cells aligned radially relative to each other. Reproductive rosette members generate predivisional daughter cells with flagella, inducing rotations of the rosette as a whole. Such rotations exhibit dynamic angular velocities and lead to intermittent linear movements along liquid-solid interfaces, resembling rolling movements. We reconstructed the translational and rotational dynamics of the rosette movements from high-speed filming and long-term tracking. A mechanical model was developed to explain the hydrodynamic mechanism underlying such motilities. Our study illustrated a nontrivial mechanism for clustered bacteria to achieve motilities and sheds light on the adaptive significance of the collective behaviors of microorganisms in complex fluid environments.

4:28PM D20.00008 Motility and peristaltic flow in maintaining microbiome populations., SEYED AMIR MIRBAGHERI, HENRY C. FU, The University of Utah — Bacteria are an important component of the microbiome in the digestive tract, and must be able to maintain their population despite the fact that the contents of the intestines are constantly flowing towards evacuation. Many bacteria accomplish this by colonizing the surfaces of the intestines where flows diminish, but some species live in the lumen. We attempt to address whether swimming motility of these species plays an important role in maintaining bacterial population in the face of peristaltic pumping out of the intestine. Using a two-dimensional model of peristaltic flows induced by small-amplitude traveling waves we examine the Lagrangian trajectories of passive bacteria as well as motile bacteria, which are treated as Brownian particles undergoing enhanced diffusion due to the bacteria's run-and-tumble motility. We examine how the densities of growing populations of bacteria depend on the combination of motility and peristaltic flow.

4:41PM D20.00009 Instabilities in the Swimming of Bacteria, EMILY RILEY, ERIC LAUGA, University of Cambridge — Peritrichously flagellated bacteria, such as *E. coli* and *B. subtilis*, have flagella randomly distributed over their body. These flagella rotate to generate a pushing force that causes the cell to swim body first. For changes in direction these flagella return to their randomly distributed state where the flagella point in many different directions. The main observed state of swimming peritrichously flagellated bacteria however is one where all their flagella gathered or bundled at one end of the body. In this work we address this problem from the point of view of fluid-structure interactions and show theoretically and numerically how the conformation of flagella depends on the mechanics of the cell.

4:54PM D20.00010 Bacteria dispersion in microchanel containing random obstacles¹, ADAMA CREPPY, HAROLD AURADOU, FAST, Université Paris-Sud, CARINE DOUARCHE, LPS, Université Paris-Sud, VERONICA D'ANGELO, FIUBA, Argentina, JACKY NGUYEN, FAST, Université Paris-Sud, GROUPO DE MEDIOS POROSOS, FIUBA COLLABORATION, LABORATOIRE DE PHYSIQUE DU SOLIDE COLLABORATION, FLUIDE AUTOMATIQUE ET SYSTEMES THERMIQUES COLLABORATION — Dispersion of particles in porous media is a classical problem well studied where physical laws are well established and show good agreement with experimental observations. Recently, contrary to what is thought, observations revealed that self-propelled particles under flow, orient their swimming, what is designated by the term of rheotaxis. But less is known about what happen for self-propelled particles under flow in presence of obstacles. For this purpose, we developed a specific experimental setup in order to show the coupling of bacteria *E. Coli* RP437 strain swimming with the presence of obstacles in the dispersion process. We chose to develop a micro-fluidic device of rectangular section of $0.05 \mu\text{m}^2$ containing obstacles of different sizes($10\text{--}150 \mu\text{m}$) when a bacteria size is about $1 \mu\text{m}$. Thanks to the transparency of the flow we can track hundreds of trajectories of bacteria, the analysis of which revealed that their swimming influences the dispersion when the flow velocity is of the order of their swimming velocity ($10 \mu\text{m/s}$).

¹Agence Nationale de la Recherche

Sunday, November 20, 2016 2:57PM - 5:07PM – Session D21 Bubbles: Growth, Heat transfer and Boiling D139-140 - Emilio Baglietto, MIT

2:57PM D21.00001 A more general Force Balance Model to predict Bubble Departure and Lift-off Diameters in flow boiling¹, RAVIKISHORE KOMMAJOSYULA, Massachusetts Inst of Tech-MIT, THOMAS MAZZOCCO, WALTER AMBROSINI, University of Pisa, EMILIO BAGLIETTO, Massachusetts Inst of Tech-MIT — Accurate prediction of Bubble Departure and Lift-off Diameters is key for development of closures in two-phase Eulerian CFD simulation of Flow Boiling, owing to its sensitivity in the Heat Flux partitioning approach. Several models ranging from simple correlations to solving complex force balance models have been proposed in literature; however, they rely on data-fitting for specific databases, and have shown to be inapplicable for general flow applications. The aim of this study is to extend the approach by proposing a more consistent and general formulation that accounts for relevant forces acting on the Bubble at the point of Departure and Lift-off. Among the key features of the model, the Bubble Inclination angle is treated as an unknown to be inferred along with the Departure Diameter, and the relative velocity of the bubble sliding on the surface, is modeled to determine the Lift-off Diameter. A novel expression is developed for the bubble growth force in terms of flow quantities, based on extensive data analysis. The model has been validated using 6 different experimental databases with varying flow conditions and 3 fluids. Results show high accuracy of predictions over a broad range, outperforming existing models both in terms of accuracy and generality.

¹CASL - The Consortium for Advanced Simulation of LWRs

3:10PM D21.00002 Counter-current thermocapillary migration of bubbles in self-rewetting liquids.¹, R. NAZARETH, University of Edinburgh, P. SAENZ, Massachusetts Institute of Technology, K. SEFIANE, University of Edinburgh, J. KIM, University of Maryland, P. VALLURI, University of Edinburgh — In this work, we study the counter-current thermocapillary propulsion of a suspended bubble in the fluid flowing inside a channel subject to an axial temperature gradient when the surface tension dependence on temperature is non-monotonic. We use direct numerical simulations to address the two-phase conservation of mass, momentum and energy with a volume-of-fluid method to resolve the deformable interface. Two distinct regimes of counter-current bubble migration are characterized: i) “exponential decay” where the bubble decelerates rapidly until it comes to a halt at the spatial position corresponding to the minimum surface tension and ii) “sustained oscillations” where the bubble oscillates about the point of minimum surface tension. We illustrate how these sustained oscillations arise at low capillary number $O(10^{-5})$ and moderate Reynolds number $O(10)$ and, they are damped by viscosity at lower Reynolds number. These results are in agreement with the experiments by Shanahan and Sefiane (Sci. Rep. 4, 2014).

¹The work was supported by the Science without Borders program from CAPES agency of Brazilian Ministry of Education and the European Commissions Thermapower Project (294905).

3:23PM D21.00003 The effect of flow pattern around a bubble rising near a vertical wall, on the wall to liquid heat transfer., PRAMOD BHUVANKAR, SADEGH DABIRI, Purdue Univ — Two-phase flow is an effective means for heat removal due to the enhanced convective effect caused by bubbly flow and the usually high latent heat of vaporization of the liquid phase. We present a numerical study of the effect of flow patterns around a single bubble rising in shear flow near a vertical wall, on the wall-to-liquid heat transfer. The Navier-Stokes equations are solved in a frame of reference moving with the bubble, by using the front tracking method for interface tracking. Our simulations reveal an enhancement of heat transfer downstream of the bubble, and a less pronounced diminishment of heat transfer upstream of the bubble. We observe that in the range of $5 \leq Re \leq 40$ for Reynolds number based on shear and bubble diameter, heat transfer first increases, attains a maximum and decreases as Re increases. The optimum Re depends on the Archimedes number. The heat transfer enhancement is attributed to flow reversal happening in a confined region of the shear flow, in the presence of a bubble. The analytical solution of $2 - D$ inviscid shear flow over a cylinder near a wall is used to identify two parameters of flow reversal namely ‘reversal height’ and ‘reversal width’. These parameters are then used to qualitatively explain what we observe in $3 - D$ simulations.

3:36PM D21.00004 ABSTRACT WITHDRAWN —

3:49PM D21.00005 Simulating Heat Flux and Bubble Nucleation using Molecular Dynamics¹, TASSOS KARAYIANNIS, Brunel University, EDWARD SMITH, Imperial College London, KHELLIL SEFIANE, University of Edinburgh , OMAR MATAR, Imperial College London — Modelling the heat flux in multiphase flow situations must account for nucleation of bubbles, non-linear heat transfer coefficients, complex molecular interaction at the surface, detailed surface textures as well as build up of material on the surface. These complex factors combine to define the well known boiling curve, which characterises the heat flux for a given temperature gradient. Understanding and optimisation of this boiling curve, and its critical heat flux (CHF), is a problem of great importance. Molecular dynamics (MD), by modelling the motion of the individual molecules, can replicate the bubble nucleation and heat flux. Details of the wall-fluid interaction are represented with complex textures and the surface materials can be explicitly reproduced. In this talk, MD simulation results are presented for bubble nucleation and heat flux. The heat flux is matched to experimental results and the process of nucleation explored for both fractal and textured surfaces. The unique insights from the molecular scale are discussed and potential applications including surface design and coupled molecular to continuum simulation are presented.

¹EPSRC UK platform grant MACIPh (EP/L020564/1)

4:02PM D21.00006 Numerical Simulation of Bubble Formation in a Microchannel Using a Micro-Pillar, LUZ AMAYA-BOWER, Central Conn State Univ — A three dimensional numerical simulation of bubble formation in a microchannel with a micro-pillar is investigated. Simulation results are validated against experimental data, where the working fluids are water and nitrogen. The gas enters the microchannel through a single slit located at 0, along the pillar’s depth. The bubble formation process has two main regimes, namely discrete bubble and attached ligament. The transformation from one regime to another is dictated by the capillary number Ca and the volumetric flow ratio Q . An analysis is performed to evaluate the critical values at which the transformation takes place. In addition, for the discrete bubble regime, the simulation results provide a proportional correlation between Q and the size of bubbles, and an inversely proportional relationship between Q and formation time, for each Ca . The computations are performed in the range of $10^{-4} < Ca < 10^{-2}$ and $0.5 < Q < 10^{-2}$.

4:15PM D21.00007 Gas depletion through single gas bubble diffusive growth and its effect on subsequent bubbles¹, ALVARO MORENO SOTO, Physics of Fluids Group, University of Twente, ANDREA PROSPERETTI, Department of Mechanical Engineering, John Hopkins University, DETLEF LOHSE, DEVARAJ VAN DER MEER, Physics of Fluids Group, University of Twente, PHYSICS OF FLUID GROUP COLLABORATION, MCEC NETHERLANDS CENTER FOR MULTISCALE CATALYTIC ENERGY CONVERSION COLLABORATION — In weakly supersaturated mixtures, bubbles are known to grow quasi-statically as diffusion-driven mass transfer governs the process. In the final stage of the evolution, before detachment, there is an enhancement of mass transfer, which changes from diffusion to natural convection [O.R. Enríquez et al., The quasi-static growth of CO₂ bubbles, *Journal of Fluid Mechanics* 741, R1 (2014)]. Once the bubble detaches, it leaves behind a gas-depleted area. The diffusive mass transfer towards that region cannot compensate for the amount of gas which is taken away by the bubble. Consequently, the consecutive bubble will grow in an environment which contains less gas than for the previous one. This reduces the local supersaturation of the mixture around the nucleation site, leading to a reduced bubble growth rate. We present quantitative experimental data on this effect and the theoretical model for depletion during the bubble growth rate.

¹This work was supported by the Netherlands Center for Multiscale Catalytic Energy Conversion (MCEC), an NWO Gravitation programme funded by the Ministry of Education, Culture and Science of the government of the Netherlands.

4:28PM D21.00008 Mass transfer effects on the transmission of bubble screens¹, DANIEL FUSTER, Institut d'Alembert UPMC-CNRS, LUCA BERGAMASCO, Institut d'Alembert UPMC — In this work we investigate, theoretically and numerically, the reflection and transmission properties of bubble screens excited by pressure wave pulses. We use modified expressions for the bubble resonance frequency and the damping factor in order to capture the influence of mass transfer on the reflection-transmission coefficients. In addition to the influence of variables such as the bubble radius and the averaged inter-bubble distance, the analysis reveals that in conditions close to the saturation line there exists a regime where the heat transport surrounding the bubble plays an important role on the bubble's response also influencing the reflection properties of the bubble screen. The linear analysis allows us to predict the critical vapor content beyond which liquid heat's transport controls the dynamic response of the bubbles. Numerical simulations show that these effects become especially relevant in the nonlinear regime.

¹ANR Cachmap

4:41PM D21.00009 Parametric study of cross shaped hydrophobic dot for pool boiling¹, JUNG SHIN LEE, JOON SANG LEE, Yonsei Univ — In this work we applied the shape of hydrophobic dots as a new variable of pool boiling with patterned wettability. We investigated the effect of dot shapes on heat transfer rate and buoyancy of bubbles. The shape of dot is set to be cross-shaped with the aspect ratios of the branches were varied in four cases: 0.173, 0.444, 1.074, and 2.000. In this research, multiphase single component lattice Boltzmann model was used for the simulation. The shapes of contact lines were similar to the shape of border lines of hydrophobic dots, but the surface tension to make the contact line in circular shape also existed. For dots with larger aspect ratio, the shape of contact line was too distorted. Therefore large portion of the contact line invaded to the hydrophilic surface via surface tension. The fluid near contact line on the hydrophilic surface has shown large buoyancy force by capillary flow toward contact line. This overall large buoyancy force caused the quick departure of the bubble. Therefore, with large aspect ratio, the heat transfer dropping period was reduced, while heat transfer rate increased in total nucleation cycle.

¹National Research Foundation of Korea (NRF) (grant number: NRF-2015R1A2A1A15056182, NRF-2015R1A5A1037668, NRF-21A20151313219)

4:54PM D21.00010 Transition process leading to microbubble emission boiling on horizontal circular heated surface in subcooled pool¹, ICHIRO UENO, JUN ANDO, KAZUNA HORIUCHI, TAKAHITO SAIKI, TOSHIHIRO KANEKO, Tokyo University of Science — Microbubble emission boiling (MEB) produces a higher heat flux than critical heat flux (CHF) and therefore has been investigated in terms of its heat transfer characteristics as well as the conditions under which MEB occurs. Its physical mechanism, however, is not yet clearly understood. We carried out a series of experiments to examine boiling on horizontal circular heated surfaces of 5 mm and of 10 mm in diameter, in a subcooled pool, paying close attention to the transition process to MEB. High-speed observation results show that, in the MEB regime, the growth, condensation, and collapse of the vapor bubbles occur within a very short time. In addition, a number of fine bubbles are emitted from the collapse of the vapor bubbles. By tracking these tiny bubbles, we clearly visualize that the collapse of the vapor bubbles drives the liquid near the bubbles towards the heated surface, such that the convection field around the vapor bubbles under MEB significantly differs from that under nucleate boiling. Moreover, the axial temperature gradient in a heated block (quasi-heat flux) indicates a clear difference between nucleate boiling and MEB. A combination of quasi-heat flux and the measurement of the behavior of the vapor bubbles allows us to discuss the transition to MEB.

¹This work was financially supported by the 45th Research Grant in Natural Sciences from The Mitsubishi Foundation (2014 - 2015), and by Research Grant for Boiler and Pressurized Vessels from The Japan Boiler Association (2016).

Sunday, November 20, 2016 2:57PM - 5:07PM –
Session D22 Nano Flows: Computations and Modeling E141/142 - Dimitrios Papavassiliou,
University of Oklahoma

2:57PM D22.00001 A Langevin model for the Dynamic Contact Angle Parameterised Using Molecular Dynamics¹, EDWARD SMITH, ERICH MULLER, RICHARD CRASTER, OMAR MATAR, Imperial College London — An understanding of droplet spreading is essential in a diverse range of applications, including coating processes, dip feed reactors, crop spraying and biomedical treatments such as surfactant replacement theory. The default modelling tools for engineering fluid dynamics assume that the continuum hypothesis is valid. The contact line motion is very difficult to capture in this paradigm and requires some form of closure model, often tuned *a priori* to experiments. Molecular dynamics (MD), by assuming only an inter-molecular potential, reproduces the full detail of the three-phase contact line with no additional modelling assumptions. This provides an ideal test-bed to understand contact line motion. In this talk, MD results for a sheared liquid bridge are presented. The evolution and fluctuations of the dynamic contact angle are parameterised over a range of wall sliding speeds and temperatures. A Langevin model is proposed to reproduce the fluctuations and evolution of the contact angle. Results from this model are compared to molecular simulation data showing excellent agreement. The potential applications of this model, as well as limitation and possible extensions, are discussed.

¹EPSRC UK platform grant MACIPh (EP/L020564/1)

3:10PM D22.00002 A Second Order Temporal Integrator for Brownian Dynamics of Rigid Bodies¹, BRENNAN SPRINKLE, Northwestern University, FLORENCIO USABIAGA, New York University, NEELESH PATANKAR, Northwestern University, ALEKSANDER DONEV, New York University — Simulating Brownian motion of passive or active rigid bodies, with arbitrary shape, suspended in a viscous solvent is examined. Existing numerical techniques which capture the correct stochastic drift term require the solution of two saddle point problems per time step and only achieve first order accuracy deterministically. The saddle point systems are required to enforce the rigidity constraint and can be quite expensive to solve, for a large number of rigid bodies. We propose a novel method which requires the solution of two saddle point problems per time step but achieves second order accuracy in time.

¹This work was supported in part by NSF grant CBET-1418672

3:23PM D22.00003 Molecular dynamics simulations of the rotational and translational diffusion of a Janus rod-shaped nanoparticle., ALI KHARAZMI, Michigan State Univ, NIKOLAI PRIEZJEV, Wright State University — We investigate the diffusion of a Janus nanoparticle immersed in a dense Lennard-Jones fluid using molecular dynamic simulations. In particular we consider a rod-shaped particle with different surface wettability on each half-side of the particle and analyze the mean square displacement and the translational and rotational velocity autocorrelation functions. It is found that diffusion is enhanced when the wettability contrast is high and the local slip length on the nonwetting side is relatively large. We also examine the time evolution of the orientation tensor and correlate it with the particle displacement. These results are compared with our previously published results on diffusive dynamics of a Janus sphere with two hemispheres of different wettability.

3:36PM D22.00004 Investigating the gas cushion model for nano-structured superhydrophobic surfaces¹, JASON REESE, SRINIVASA RAMISETTI, MATTHEW BORG, University of Edinburgh, DUNCAN LOCKERBY, University of Warwick — We investigate the water slip properties of different nano-structured surfaces using non-equilibrium molecular dynamics (NEMD) simulations. We predict the fluid slip lengths of surface coatings comprising carbon nanotubes on platinum substrates, with nitrogen gas trapped in the interstitial gaps. Our NEMD results do not support the gas-cushion model proposed by Vinogradova (Langmuir 11:2213-2220, 1995) as this does not account for the rarefied gas effects present in nano/micro gas layers. We therefore propose a slip gas-cushion model which incorporates some of the rarefied gas effects and agrees better with our NEMD slip length calculations.

¹EPSRC grant nos. EP/N016602/1, EP/K038621/1, EP/K038664/1, EP/K038427/1

3:49PM D22.00005 A multiscale transport model for binary Lennard Jones mixtures in slit nanopores., RAVI BHADAURIA, N. R. ALURU, University of Illinois at Urbana-Champaign — We present a quasi-continuum multiscale hydrodynamic transport model for one dimensional isothermal, non-reacting binary mixture confined in slit shaped nanochannels. We focus on species transport equation that includes the viscous dissipation and interspecies diffusion term of the Maxwell-Stefan form. Partial viscosity variation is modeled by van der Waals one fluid approximation and the Local Average Density Method. We use friction boundary conditions where the wall-species friction parameter is computed using a novel species specific Generalized Langevin Equation model. The transport model accuracy is tested by predicting the velocity profiles of Lennard-Jones (LJ) methane-hydrogen and LJ methane-argon mixtures in graphene slit channels of different width. The resultant slip length from the continuum model is found to be invariant of channel width for a fixed mixture molar concentration. The mixtures considered are observed to behave as single species pseudo fluid, with the friction parameter displaying a linear dependence on the molar composition. The proposed model yields atomistic level accuracy with continuum scale efficiency.

4:02PM D22.00006 Behavior of a nano-particle and a polymer molecule in a nano-scale four-roll mill, MINH VO, DIMITRIOS PAPAVASSILIOU, The University of Oklahoma — The four-roll mill device could be used to create a mixed flow from purely extensional stresses to completely rotational through the proper selection of speed and direction of each of the four cylindrical rollers. Considerable research has been done with this device for macroscale rheological studies.. In our study, the dissipative particle dynamics (DPD) method was employed to investigate the behavior of a nano-sphere and a polymer molecule in different conditions within a four-roll mill device. Hydrophilic properties of each roll were generated by adjusting interaction parameters and using bounce back boundary condition at the solid surface. All simulations were run up to 4×10^6 time steps at room temperature using the open source LAMMPS package. After the flow in the system reached equilibrium, a nano-sphere and then a polymer chain were released at the center of the simulation box. Their trajectories were recorded at different shear rate conditions. The propagation of nanosphere in different rotational flow will be discussed. Additionally, the deformation of polymer chains will be compared to that in a simple shear flow.

replacing DFD16-2016-000883.

4:15PM D22.00007 Spontaneous Ion Depletion and Accumulation Phenomena Induced by Imbibition through Permselective Medium¹, HYOMIN LEE, Seoul Natl Univ (SNU), YEONSU JUNG, SUNGMIN PARK, HO-YOUNG KIM, SUNG JAE KIM, SNU — Generally, an ion depletion region near a permselective medium is induced by predominant ion flux through the medium. External electric field or hydraulic pressure has been reported as the driving forces. Among these driving forces, an imbibition through the nanoporous medium was chosen as the mechanism to spontaneously generate the ion depletion region. The water-absorbing process leads to the predominant ion flux so that the spontaneous formation of the ion depletion zone is expected even if there are no additional driving forces except for the inherent capillary action. In this presentation, we derived the analytical solutions using perturbation method and asymptotic analysis for the spontaneous phenomenon. Using the analysis, we found that there is also spontaneous accumulation regime depending on the mobility of dissolved electrolytic species. Therefore, the rigorous analysis of the spontaneous ion depletion and accumulation phenomena would provide a key perspective for the control of ion transportation in nanofluidic system such as desalinator, preconcentrator, and energy harvesting device, etc.

¹Samsung Research Funding Center of Samsung Electronics (SRFC-MA1301-02) and BK21 plus program of Creative Research Engineer Development IT, Seoul National University

4:28PM D22.00008 Analysis of micro-fluidic tweezers in the Stokes regime¹, LONGHUA ZHAO, Case Western Reserve University, YANG DING, Beijing Computational Science Research Center — Nanowire fluidic tweezers have been developed to capture and manipulate micro objects. The fluidic trapping force and the fluid field are important to achieve accurate control, but have not been fully understood yet. Utilizing singularity method, we construct the exact velocity field to analyze flows induced by a spheroid nanowire tumbling in the Stokes regime. To further explore the trapping, we analyze the trajectories of rigid or deformable microspheres near the tumbling nanowire using regularized Stokeslet method. The fluid structure, the trapping phenomenon and mechanism, and precise relation about trapping with the geometry will be presented.

¹YD is sponsored by the Recruitment Program of Global Young Experts (China)

4:41PM D22.00009 The filtration of colloidal gold nanoparticles with carbon nanotubes, FRANS JAN DE JONG, Hamburg University of Technology (Hamburg, Germany), ADELINE BUFFET, German Electron Synchrotron (Hamburg, Germany) — Understanding the local filtering of nanoparticles (NPs) is essential for the development and optimization of medical and industrial applications. Microfocus small-angle X-ray scattering (μ SAXS) was used to determine the local filtration kinetics of 100 nm sized colloidal gold nanoparticles (Au NPs) within a multi-walled carbon nanotube (MWCNT) forest. To get a physical insight into the Au NP filtration process within the MWCNT forest a novel model based on the well-known DLVO theory was developed. The DLVO theory is commonly used to describe the interaction between colloidal particles. In addition to the attractive Van de Waals force and the electrostatic double-layer force, a non-DLVO force is added to account for hydration and hydrophobic effects. The model presented here shows that the Au NPs are mainly unfavorably deposited into the so-called secondary energy minimum. This latter finding is in good agreement with the experimental observations and the literature, in which unfavorable particle deposition is related to deposition into the secondary energy minimum. The use of μ SAXS to get a physical insight into the local deposition kinetics of submicrometer particles opens up new pathways to optimize the preparation of MWCNT forests for filtration purposes.

4:54PM D22.00010 DSMC Simulation of Radiometric Flow in a Ratchet-Channel Nano-Pump, EHSAN ROOHI, VAHID SHAHBI, Ferdowsi University of Mashhad, TOBIAS BAIER, STEFFEN HARDT, Technical University Darmstadt — The current paper presents a description of the fluid and pumping characteristics of a ratchet-type nano-scale pump suggested by Donkov et al. [1] and presents the optimum geometry and working conditions of the pump. The pump consists of a ratchet channel with the temperature gradient applied between the opposing walls. Here, we report the physical mechanism of flow induction within the pump and show that the combination of configuration and boundary temperature of this pump induces a radiometric-type flow. Benefiting from the DSMC simulations, we suggest the optimum working condition/geometrical size of the pump. Comparison of DSMC simulation with analytical relations is reported.

Sunday, November 20, 2016 2:57PM - 5:07PM –
Session D25 Microscale Flows: Particles E145 - Sascha Hilgenfeldt, University of Illinois at Urbana-Champaign

2:57PM D25.00001 Attractive and Repulsive Forces on Particles in Oscillatory Flow, SIDDHANS AGARWAL, Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, BHARGAV RALLABANDI, Mechanical and Aerospace Engineering, Princeton University, DAVID RAJU, RAQEEB THAMEEM, SASCHA HILGENFELDT, Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — A large class of oscillating flows gives rise to rectified streaming motion of the fluid. It has recently been shown that particle transport in such flows, excited by bubbles oscillating at ultrasound frequencies, leads to differential displacement and efficient sorting of microparticles by size. We derive a general expression for the instantaneous radial force experienced by a small spherical particle in the vicinity of an oscillating interface, and generalize the radial projection of the Maxey-Riley equation to include this effect. Varying relevant system parameters, we show that the net effect on the particle can be either an attraction to or a repulsion from the bubble surface, depending in particular on the particle size and the particle/fluid density contrast. We demonstrate that these predictions are in agreement with a variety of experiments.

3:10PM D25.00002 Rectified Motion of Microparticles: Generalizing Streaming and Radiation Forces, DAVID RAJU, SIDDHANS AGARWAL, Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, BHARGAV RALLABANDI, Mechanical and Aerospace Engineering, Princeton University, SASCHA HILGENFELDT, Mechanical Science and Engineering, University of Illinois at Urbana-Champaign — It is well known that a wide variety of oscillating flows gives rise to steady streaming, i.e., rectified motion of fluid elements. Small spherical particles introduced into such a flow have been shown to experience an additional lift force that ultimately leads to particle trajectories that differ systematically from the fluid element pathlines. We demonstrate a systematic derivation of this differential particle motion on the steady streaming time scale, so that time-averaged particle trajectories can be directly predicted without computation on the fast, oscillatory time scale. The resulting dynamics can be interpreted as a generalization of streaming flow, while the closed-form lift force provides a generalization of the secondary radiation force, to which it reduces in appropriate limiting cases. These very general results are validated by comparison with experiments in the context of bubble streaming, but apply to a large class of other flows as well.

3:23PM D25.00003 3D deterministic lateral displacement separation systems, SIQI DU, GERMAN DRAZER, Rutgers, The State University of New Jersey — We present a simple modification to enhance the separation ability of deterministic lateral displacement (DLD) systems by expanding the two-dimensional nature of these devices and driving the particles into size-dependent, fully three-dimensional trajectories. Specifically, we drive the particles through an array of long cylindrical posts, such that they not only move parallel to the basal plane of the posts as in traditional two-dimensional DLD systems (*in-plane motion*), but also along the axial direction of the solid posts (*out-of-plane motion*). We show that the (projected) in-plane motion of the particles is completely analogous to that observed in 2D-DLD systems and the observed trajectories can be predicted based on a model developed in the 2D case. More importantly, we analyze the particles out-of-plane motion and observe significant differences in the net displacement depending on particle size. Therefore, taking advantage of both the in-plane and out-of-plane motion of the particles, it is possible to achieve the simultaneous fractionation of a polydisperse suspension into multiple streams. We also discuss other modifications to the obstacle array and driving forces that could enhance separation in microfluidic devices.

3:36PM D25.00004 Inertial Particle Migration in the Presence of a Permeate Flow, MIKE GARCIA, AMANDA SINGELTON, SUMITA PENNATHUR, University of California Santa Barbara — Tangential Flow Filtration (TFF) is a rapid and efficient method for the filtration and separation of suspensions of particles such as viruses, bacteria or cellular material. Enhancing the efficacy of TFF not only requires a detailed understanding of particle transport mechanisms, but also the interactions between these mechanisms and a porous wall. In this work, we numerically and experimentally explore the mechanisms of inertial particle migration in the presence of a permeate flow through the porous walls of a microchannel. Numerically, we develop a force balance model to understand the competition between permeate and inertial forces and the resultant consequences on the particle equilibrium location. Experimentally, we fabricated MEMS TFF devices to study the migration of 5, 10 and 15 μ m fluorescent polystyrene beads in straight channels with perpendicular permeate flow rates up to 90% of the inlet flow rate. We find that the permeate flow directly influences the inertial focusing position of the particles, both as a function of downstream channel position and ratio of inlet to outlet flow rate. Comparing experiments to our model, we can identify inertial, viscous and a co-dominant regimes.

3:49PM D25.00005 Two-dimensional confocal laser scanning microscopy image correlation for nanoparticle flow velocimetry, BRIAN JUN, Purdue University, MATTHEW GIARRA, Virginia Tech, BRIAN GOLZ, RUSSELL MAIN, PAVLOS VLACHOS, Purdue University — We present a methodology to mitigate the major sources of error associated with two-dimensional confocal laser scanning microscopy (CLSM) images of nanoparticles flowing through a microfluidic channel. The correlation-based velocity measurements from CLSM images are subject to random error due to the Brownian motion of nanometer-sized tracer particles, and a bias error due to the formation of images by raster scanning. Here, we develop a novel ensemble phase correlation with dynamic optimal filter that maximizes the correlation strength, which diminishes the random error. In addition, we introduce an analytical model of CLSM measurement bias error correction due to two-dimensional image scanning of tracer particles. We tested our technique using both synthetic and experimental images of nanoparticles flowing through a microfluidic channel. We observed that our technique reduced the error by up to a factor of ten compared to ensemble standard cross correlation (SCC) for the images tested in the present work. Subsequently, we will assess our framework further, by interrogating nanoscale flow in the cell culture environment (transport within the lacunar-canalicular system) to demonstrate our ability to accurately resolve flow measurements in a biological system.

4:02PM D25.00006 Extension of Kirkwood-Riseman Theory across the Entire Range of Knudsen Numbers, JAMES CORSON, MICHAEL ZACHARIAH, GEORGE MULHOLLAND, HOWARD BAUM, University of Maryland — Aggregates of small, spherical particles form in many high temperature processes (e.g. soot formation). We consider the drag force on a fractal aggregate using Kirkwood-Riseman (KR) theory, in which the force exerted on each particle in the aggregate can be obtained from the hydrodynamic interaction tensor \mathbf{T} and the friction coefficient f for flow around an isolated sphere. The force on the aggregate is the vector sum of the force on each particle. Meakin and Deutch (1987) demonstrated that this approach yields a reasonable estimate of the drag force for an aggregate in continuum flow, where \mathbf{T} is the modified Oseen tensor of Rotne and Prager. We have extended this approach across the entire Knudsen range by calculating \mathbf{T} and f using the BGK model in the linearized Boltzmann equation. Our results for f agree with Millikan's data for the entire Knudsen range, and the free molecular drag force on the aggregate calculated with our extended KR theory is within a few percent of the drag computed using Monte Carlo methods. These results suggest that we can obtain a reasonable estimate of the drag in the transition regime in seconds once we have obtained \mathbf{T} and f for a given Knudsen number.

4:15PM D25.00007 Sheath-Free Elasto-Inertia Separation of Particles Based on Shape in Straight Rectangular Microchannels, XIANGCHUN XUAN, XINYU LU, Clemson University — We demonstrate the use of straight rectangular microchannels to obtain a shape-based separation of equal-volumed spherical and peanut-shaped particles in viscoelastic fluids. This continuous sheath-free separation arises from the shape-dependent equilibrium particle positions as a result of the flow-induced elasto-inertial lift. A continuous transition from single to dual and to triple equilibrium positions is observed for both types of particles with the increase of flow rate. However, the flow rate at which the transition occurs differs with the particle shape, which is thought to correlate the rotational effects of non-spherical particles.

4:28PM D25.00008 Dynamical Density Functional Theory and Hydrodynamic Interactions in Confined Systems¹, BENJAMIN GODDARD, University of Edinburgh, ANDREAS NOLD, SERAFIM KALLIADASIS, Imperial College London — Colloidal systems consist of nano- to micrometer-sized particles suspended in a bath of many more, much smaller and much lighter particles. Motion of the colloidal particles through the bath, e.g. when driven by external forces such as gravity, induces flows in the bath. These flows in turn impart forces on the colloid particles. These bath-mediated forces, known as Hydrodynamic Interactions (HIs) strongly influence the dynamics of the colloid particles. This is particularly true in confined systems, in which the presence of walls substantially modifies the HIs compared to unbounded geometries. For many-particle systems, the many of degrees of freedom prohibit a direct solution of the underlying stochastic equations and a reduced model is necessary. We employ elements from the statistical mechanics of classical fluids, namely Dynamical Density Functional Theory (DDFT) [1,2], the computational complexity of which is independent of the number of particles to include both inter-particle and particle-wall HI and demonstrate the physical importance of using the correct description of HIs in confined systems. In addition, DDFT allows us to isolate and investigate different components of HIs.

[1] Phys. Rev. Lett. 109, 120603 (2012)

[2] J. Phys.: Condens. Matter 25, 035101 (2013)

¹Supported by EPSRC grant EP/L025159

4:41PM D25.00009 Three-dimensional particle migration in a bubble-driven acoustic streaming flow, ANDREAS VOLK, MASSIMILIANO ROSSI, Bundeswehr University Munich, BHARGAV RALLABANDI, Princeton University, SASCHA HILGENFELDT, University of Illinois at Urbana-Champaign, CHRISTIAN J. KAEHLER, ALVARO MARIN, Bundeswehr University Munich — Oscillations of hemicylindrical bubbles in microchannels generate streaming flows with characteristic toroidal structures. Over long times, a passive tracer in such a flow typically explores a large fluid volume extending several bubble radii away from the bubble center and covering the whole height of the microchannel parallel to the bubble axis. In contrast, finite-sized particles are observed to migrate to specific confined locations along the axial direction while being confined to orbits of much smaller radial extent. The size of the orbits and the axial location not only depend on the particle size, but also on the relative particle density with the surrounding fluid. In this work we will show three-dimensional measurements that reveal the size- and density-sensitive migration of the particles. A simple way to emulate the migration is to solve numerically the trajectory of a particle including only steric interactions with the bubble and the walls due to its finite size (no penetration). By comparing the experimental results with this simplistic numerical model, we will show that additional forces are necessary to explain the particle dynamics. Finally, we will discuss the effect of hydrodynamic and acoustic forces experienced by the particle in the vicinity of the bubble.

4:54PM D25.00010 Experimental analysis on viscoelasticity-induced migration of RBCs using digital holographic microscopy¹, TAESIK GO, HYEOKJUN BYEON, SANG JOON LEE, Pohang Univ of Sci & Tech — Migration of particles in viscoelastic fluids has recently received large attention, because the generated elastic forces in viscoelastic fluids give rise to a simple focusing pattern over a wide range of flow rates. In this study, the vertical focusing and alignment of rigid spherical particles, normal and hardened RBCs in a viscoelastic fluid were experimentally investigated by employing a digital in-line holographic microscopy (DIHM). By the elastic forces, the three different particles are pushed away from the walls and concentrated in the midplane of the rectangular microchannel. Furthermore, most of both RBCs maintain face-on orientation in the microchannel. The effects of deformability of RBC on the viscoelasticity-induced migration and orientation in the channel were also examined. In contrary to non-deformable particles, normal RBCs are dispersed as flow rate increases. In the region near side wall of the microchannel, normal RBCs have edge-on orientation with a large angle of inclination, compared to hardened RBCs. These findings have a strong potential in the design of microfluidic devices for deformability-based separation of cells in viscoelastic fluid flows and label-free diagnoses of certain hematological diseases.

¹This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP) (No. 2008-0061991).

Sunday, November 20, 2016 2:57PM - 5:07PM –

Session D26 Reacting Flows: Instability E146 - Peter Schmid, Imperial College - London

2:57PM D26.00001 The effect of mean flow swirl on the transfer function of an M-flame, CALUM SKENE, PETER SCHMID, Imperial College London — Direct numerical simulations of the compressible, reactive Navier-Stokes equations are used to probe the influence of mean flow swirl on the frequency response of an axisymmetric M-flame. Using linearized governing equations, coupled with its adjoint analogue, the optimal gain with respect to harmonic forcing is computed using an iterative direct-adjoint looping technique. The discrete adjoint equations are determined by modular automatic differentiation of the linearized code. The direct and adjoint information are further used to provide sensitivity information with respect to the forcing frequency, as well as to changes in the governing parameters (swirl number, Reynolds number, etc.). Special emphasis will be put on the influence of mean flow swirl on amplification and frequency shifts in the flame transfer function (FTF).

3:10PM D26.00002 Two-dimensional corrugated flames - a consequence of the Darrieus-Landau instability, ADVITYA PATYAL, MOSHE MATALON, University of Illinois at Urbana Champaign — In this study we present for the first time the development of corrugated flame surfaces resulting from gas expansion in a three-dimensional flow as a consequence of the Darrieus-Landau instability. The computations are carried out within the context of the hydrodynamic theory where the flame is treated as a surface of density discontinuity separating burned gas from the fresh combustible mixture, and its movement is tracked via a level-set method with a propagation speed that depends on the local curvature and hydrodynamic strain. To this end, a surface parameterization method is used to accurately capture the velocity jump across the flame and the strain rate along the flame interface. The numerical scheme is shown to accurately recover the exact pole-solutions predicted by the nonlinear Michelson-Sivashinsky equation in the weak gas expansion limit, and corroborates the bifurcation results from linear stability analysis. It is shown that, in accord with experimental observations, the new conformations that evolve beyond the instability threshold have a sharp crest pointing towards the burned gas with ridges along the troughs, and that these structures propagate steadily, nearly 50% faster than planar flames.

3:23PM D26.00003 Numerical simulations of Rayleigh-Taylor instability in non-premixed flames using detailed chemistry, NITESH ATTAL, PRAVEEN RAMAPRABHU, University of North Carolina at Charlotte — The Rayleigh-Taylor (RT) instability occurs at a perturbed interface separating fluids of different densities, when the lighter fluid accelerates the heavier fluid. We examine the occurrence of the RT instability, when the perturbed interface demarcates a light, fuel stream from a heavier air stream at elevated temperatures. The study is conducted using the FLASH code with modifications that include detailed chemistry, temperature-dependent EOS, and diffusive transport. The fuel-air interface is initialized at thermal equilibrium ($T_{fuel} = T_{air} = 1000K$) in a constant background acceleration (g). We vary the density difference across the interface by diluting the H_2 fuel stream with inert N_2 . The non-premixed flame formed across a burning interface alters the underlying density (ρ) stratification, so that an initially RT unstable (stable) interface can be locally stabilized (destabilized). We observe this change in local stability for both single-wavelength and multimode perturbations, and draw some conclusions on the implications of these findings to applications such as ultra-compact combustors. We also make some comparisons of the reacting, non-premixed RT problem with the corresponding inert flow.

3:36PM D26.00004 Turbulent non-premixed combustion driven by the Richtmyer-Meshkov instability, HILDA VARSHOCHI, PRAVEEN RAMAPRABHU, NITESH ATTAL, University of North Carolina at Charlotte — We report on 3D high resolution numerical simulations of a non-premixed, reacting Richtmyer-Meshkov (RM) instability performed using the FLASH code. In the simulations, a Mach 1.6 shock traverses a diffuse, corrugated material interface separating Hydrogen at 1000 K and Oxygen at 300 K, so that local misalignments between pressure and density gradients induce baroclinic vorticity at the contact line. The vorticity deposition drives the RM instability, which in turn results in combustion and flame formation. We study the evolution of the interface and the flame as the resulting RM instability grows through linear, nonlinear and turbulent stages. We develop a detailed understanding of the effects of heat release and combustion on the underlying flow properties by comparing our results with a baseline non-reacting RM flow. We document the properties of the instability (growth rates, pdfs, spectra) and the flame (scalar dissipation rate, flame surface area, heat release rate) as well as the nature of the coupling between the two. Our findings are relevant to supernovae detonation, knocking in IC engines and scramjet performance, while the underlying flow problem defined here represents a novel canonical framework to understand the broader class of non-premixed turbulent flames.

3:49PM D26.00005 Implementation of Thermal Diffusion in Chemistry Tabulation for Unstable Premixed Flames, JASON SCHLUP, GUILLAUME BLANQUART, Caltech — The inclusion of thermal diffusion, by means of multicomponent diffusion transport models, has been shown to affect the results of numerical simulations of thermo-diffusively unstable lean hydrogen flames. However, the multicomponent diffusion model involves costly matrix inversion operations, leading it to be useful in only simplified flame configurations and computational domains. In this work, a mixture-averaged thermal diffusion model is implemented into a tabulated chemistry framework. The resulting reacting flows are compared to one- and two-dimensional detailed chemistry simulations of lean hydrogen-air flames with multicomponent diffusion. The configurations used to validate the mixture-averaged thermal diffusion model with tabulated chemistry include flat and cellular tubular flames. Three-dimensional flames, both laminar and turbulent, are also considered as an application of the mixture-averaged thermal diffusion model using tabulated chemistry. These flames are compared to cases neglecting thermal diffusion and cases using detailed chemistry with the mixture-averaged thermal diffusion model.

4:02PM D26.00006 An investigation of streaklike instabilities in laminar boundary layer flames, COLIN MILLER, Univ of Maryland-College Park, MARK FINNEY, JASON FORTHOFER, SARA MCALLISTER, Missoula Fire Sciences Laboratory, MICHAEL GOLLNER, Univ of Maryland-College Park — Observations of coherent structures in boundary layer flames, particularly wildland fires, motivated an investigation on flame instabilities within a boundary layer. This experimental study examined streaklike structures in a stationary diffusion flame stabilized within a laminar boundary layer. Flame streaks were found to align with pre-existing velocity perturbations, enabling stabilization of these coherent structures. Thermocouple measurements were used to quantify streamwise amplification of flame streaks. Temperature mapping indicated a temperature rise in the flame streaks, while the region in between these streaks, the trough, decreased in temperature. The heat flux to the surface was measured with a total heat flux gauge, and the heat flux below the troughs was found to be higher at all measurement locations. This was likely a function of the flame standoff distance, and indicated that the flame streaks were acting to modify the spanwise distribution of heat flux. Instabilities in boundary layer combustion can have an effect on the spanwise distribution of heat transfer. This finding has significant implications for boundary layer combustion, indicating that instantaneous properties can vary significantly in a three-dimensional flow field.

4:15PM D26.00007 Hydrodynamic Stability Analysis of Multi-jet Effects in Swirling Jet Combustors

BENJAMIN EMERSON, TIM LIEUWEN, Georgia Inst of Tech — Many practical combustion devices use multiple swirling jets to stabilize flames. However, much of the understanding of swirling jet dynamics has been generated from experimental and computational studies of single reacting, swirling jets. A smaller body of literature has begun to explore the effects of multi-jet systems and the role of jet-jet interactions on the macro-system dynamics. This work uses local temporal and spatio-temporal stability analyses to isolate the hydrodynamic interactions of multiple reacting, swirling jets, characterized by jet diameter, D , and spacing, L . The results first identify the familiar helical modes in the single jet. Comparison to the multi-jet configuration reveals these same familiar modes simultaneously oscillating in each of the jets. Jet-jet interaction is mostly limited to a spatial synchronization of each jet's oscillations at the jet spacing values analyzed here ($L/D=3.5$). The presence of multiple jets vs a single jet has little influence on the temporal and absolute growth rates. The biggest difference between the single and multi-jet configurations is the presence of nearly degenerate pairs of hydrodynamic modes in the multi-jet case, with one mode dominated by oscillations in the inner jet, and the other in the outer jets. The close similarity between the single and multi-jet hydrodynamics lends insight into experiments from our group (Aguilar, M., Malanoski, M., Adhitya, G., Emerson, B., Acharya, V., Noble, D. and Lieuwen, T., 2015. J. Engr. Gas Turbines and Power, 137(9); Smith T., Emerson B., Chterev, I., Noble D., Lieuwen T., 2016. ASME Paper GT2016-57755).

4:28PM D26.00008 Linear stability analysis of Clarke-Riley diffusion flames , DANIEL GOMEZ-LENDINEZ, Universidad Carlos III de Madrid, WILFRIED COENEN, ANTONIO L SANCHEZ, University of California San Diego — The buoyancy-driven laminar flow associated with the Burke-Schumann diffusion flame developing from the edge of a semi-infinite horizontal fuel surface burning in a quiescent oxidizing atmosphere displays a self-similar structure, first described by Clarke and Riley (Journal of Fluid Mechanics, 74:415–431). Their analysis was performed for unity reactant Lewis numbers, with the viscosity and thermal conductivity taken to be linearly proportional to the temperature. Our work extends this seminal work by considering fuels with non-unity Lewis numbers and gas mixtures with a realistic power-law dependence of the different transport properties. The problem is formulated in terms of chemistry-free, Shvab-Zel'dovich, linear combinations of the temperature and reactant mass fractions, not changed directly by the reactions, as conserved scalars. The resulting self-similar base-flow solution is used in a linear stability analysis to determine the critical value of the boundary-layer thickness—measured by the local Grashof number—at which the flow becomes unstable, leading to the development of Görtler-like streamwise vortices. The analysis provides the dependence of the critical Grashof number on the relevant flame parameters.

4:41PM D26.00009 Spatio-temporal Linear Stability Analysis of Multiple Reacting Wakes

JACOB KUNNUMPURAM SEBASTIAN, BENJAMIN EMERSON, TIM LIEUWEN, Georgia Institute of Technology — Hydrodynamic stability of reacting shear flows plays a key role in controlling a variety of combustor behaviors, such as combustion instability, mixing and entrainment, and blowoff. A significant literature exists on the hydrodynamics of single bluff body flows, but not the multi-bluff body flows that are found in applications. The objective of this work was to compare the spatio-temporal stability of multiple reacting wakes and single reacting wakes, within the framework of linear stability theory. Spatio-temporal stability analyses are conducted on model velocity and density profiles, with key parameters being the density ratio across the flame, bluff body spacing, dimensionless shear, and asymmetry parameters (if the two wakes are dissimilar). The introduction of the additional bluff body can exert both a stabilizing and destabilizing effect on the combined two-wake system, depending on the spatial separation between the bluff bodies. Furthermore, while the most rapidly amplified mode of the single wake mode is the sinuous (asymmetric) one, in the two wake system, the most rapidly amplified mode can be either sinuous or varicose (symmetric), also depending on spatial separation.

4:54PM D26.00010 Direct numerical computation of linear stability of gaseous detonations , DMITRY KABANOV, ASLAN KASIMOV, King Abdullah Univ of Sci & Tech (KAUST) — We develop an algorithm for the computation of linear stability of gaseous detonations that combines the elements of normal-mode analysis and direct simulation. A shock-fitting method is applied to governing equations which are linearized assuming the general time dependence. The computed time series of the shock perturbation is postprocessed to determine the growth rate of instability and neutral boundaries. The method is applied to the reactive Euler equations and its simplified analogs.

Sunday, November 20, 2016 2:57PM - 5:07PM – Session D27 Surface Tension: Experimental Methods and Measurements E147-148 - Hyoungsoo Kim, Princeton University

2:57PM D27.00001 Alcohol drops on miscible liquid: mixing or spreading? , HYOUNGSOO KIM, Princeton University, KOEN MULLER, Delft University of Technology, OREST SHARDT, Princeton University, SHAHRIAR AFKHAMI, New Jersey Institute of Technology, HOWARD STONE, Princeton University — —*abstract*—We studied how a sessile drop of alcohol behaves when placed on a fully miscible liquid. The dynamics of the subsequent mixing and spreading were captured by using a high-speed camera and investigated by varying parameters (e.g., surface tension, density, and viscosity). We observed that a deposited alcohol drop on a liquid bath remains as a floating lens shape, the alcohol liquid leaks out along the rim of the droplet, and it spreads axi-symmetrically along the bottom liquid interface. To visualize spreading and mixing features, we used time-resolved Particle Tracking Velocimetry and a Schlieren method. We observed a localized mixing flow at the rim of the floating droplet where the maximum flow speed is obtained, driven by a solutal Marangoni effect. Underneath the interface of the bath liquid, a viscous boundary layer develops while the alcohol liquid spreads along the radial direction. We also observed a finite quasi-steady interfacial flow velocity regime after the alcohol droplet touched the bottom liquid surface. In this regime, the flow speed linearly increases inside the floating lens, and outside the lens the flow speed decays along the r -direction with a power-law slope, $U_r \sim r^{-1/2}$. Physical arguments to support the observations will be discussed.

3:10PM D27.00002 Elasto-capillary torsion at a liquid interface , ALEXANDROS ORATIS, TIMOTHY FARMER, JAMES BIRD, Boston University — When a liquid drop wets a solid, the droplet typically spreads over the solid. By contrast, for sufficiently compliant solids, the solid can instead spread around the drop. This wrapping phenomenon has been exploited to assemble 3-dimensional structures from 2-dimensional sheets, a process often referred to as capillary origami. Although existing studies of this self-assembly have demonstrated bending and folding, methods of inducing spontaneous twisting by means of capillarity are less clear. Here we demonstrate that spontaneous twist can be initiated in a compliant solid through a combination of surface chemistry and capillarity. Experimentally, we measure the angle of twist on a surface with binary patterns of surface wettability as we vary the solid's geometric and material properties. We develop a scaling law to relate this angle of twist to the elastic and interfacial properties, which compares well with our experimental results.

3:23PM D27.00003 A Comparative Study of the Effect of Surfactant and Temperature in Fluid Interfaces.¹, ALDO H. CORTES-ESTRADA, LAURA A. IBARRA-BRACAMONTES, ALICIA AGUILAR-CORONA, GONZALO VIRAMONTES-GAMBOA, Universidad Michoacana De San Nicolas De Hidalgo — A fluid interface is the boundary region formed when two immiscible fluids come into contact. One of the most important properties of fluid interfaces is the interfacial tension. The interfacial tension between two fluids can be modified by the presence of surfactant. In addition, the temperature is a relevant factor that can also modify the interfacial properties. In this work the behavior of the interface formed by oil and water in the presence of surfactant at different temperatures is presented. Interfacial tension measurements were obtained by the Pendant Drop technique. Two types of surfactant were tested, Sodium Dodecyl Sulfate (SDS) as a hydrophilic surfactant, and Sorbitan Monooleate (Span 80) as a lipophilic surfactant. The range of variations in temperature was from 25 to 60 Celsius degree. Hexane or Dodecane was used as the oil phase. The main results showed that the lipophilic surfactant showed a greater efficiency with respect to the hydrophilic surfactant used. As the temperature increased in the range considered an exponential decay for the interfacial tension was observed. This decay was dominated by the surfactant concentration.

¹This study was supported by the Mexican Council of Science and Technology (CONACyT) and by the Scientific Research Coordination of the University of Michoacan in Mexico.

3:36PM D27.00004 The spontaneous puncture of thick liquid films, BAPTISTE NEL, EM-MANUEL VILLERMAUX, Aix Marseille Université, CNRS, Centrale Marseille, IRPHE UMR 7342, 13384 Marseille, France — We call thick those films for which the disjoining pressure is ineffective. Water films with thickness h in the 1-10 μm range are thick, but it is also known that, paradoxically, they nucleate holes spontaneously. We have uncovered a mechanism solving the paradox. Most natural films are dirty to some extent, and we show that if a spot of dissolved substance lowers locally the surface tension of the liquid, the corresponding Marangoni stress may lead to a self-sustained instability triggering film rupture. When deposited with size a , the spot dissipates by molecular diffusion (coefficient D) along the film in a time a^2/D . Before doing so, the surface tension gradient $\Delta\sigma/a$ between the spot center (tension $\sigma - \Delta\sigma$) and the rest of the film (tension σ) induces an inhomogeneous outward interstitial flow which digs the spot, and reinforces the tension gradient. Hence the instability, which occurs within a timescale $\tau \sim \sqrt{\rho a^2 h / \Delta\sigma}$, with ρ the liquid density. When the Peclet number $Pe = a^2/D\tau$ is small, diffusion regularizes the film, which remains flat: clean films don't break, while for $Pe > 1$, the film punctures. This new scenario will be illustrated by several experiments.

3:49PM D27.00005 Evaporation of water between two microspheres: how wetting affects drying.¹, KUN CHO, School of Advanced Materials Science and Engineering, Sungkyunkwan University, YESEUL KIM, SKKU Advanced Institute of Nanotechnology (SAINT), Sungkyunkwan University, JUN LIM, Beamline Division, Pohang Light Source, JOON HEON KIM, Advanced Photonics Research Institute (APRI), Gwangju Institute of Science and Technology (GIST), BYUNG MOOK WEON², School of Advanced Materials Science and Engineering, SKKU Advanced Institute of Nanotechnology (SAINT), Sungkyunkwan University — When a small volume of water is confined between microparticles or nanoparticles, its evaporation behavior can be influenced by wettability of particles. This situation frequently appears in coating or printing of colloidal drops in which colloidal particles are uniformly dispersed into a liquid. To explore water evaporation between particles, here we study on evaporation dynamics of water between two microspheres by utilizing high-resolution X-ray microscopy for side views and optical microscopy for bottom views. We find that evaporating water gets pinned on microsphere surfaces, due to a force balance among air, water, and microspheres. Side and bottom views of evaporating water enable us to evaluate water curvature evolution around microspheres before and after pinning. Interestingly curvature evolution is controlled by cooperation of evaporation and wetting dynamics. This study would be useful in identifying and controlling of coating or printing for colloidal drops.

¹This research was supported by Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Education (NRF-2016R1D1A1B01007133).

²E-mail: bmweon@skku.edu

4:02PM D27.00006 Elasticity of Flowing Soap films, ILDOO KIM, SHREYAS MANDRE, Brown University — The robustness of soap films and bubbles manifests their mechanical stability. The single most important factor underlying the mechanical stability of soap films is its elasticity. Non-destructive measurement of the elasticity in these films has been cumbersome, because of its flowing nature. Here we provide a convenient, reproducible, and non-destructive method for measuring the elasticity by generating and inspecting Marangoni waves. Our method is based on generating an oblique shock by inserting a thin cylindrical obstacle in the flowing film, and converting the measured shock angle to elasticity. Using this method, we find a constant value for the elasticity of 22 dyne/cm in the commonly used range of film widths, thicknesses or flow rates, implying that the surface of the film is chemically saturated with soap molecules.

4:15PM D27.00007 Large-scale fluid circulation induced by asymmetric surface under vertical vibration¹, YI-CHENG HUANG, Tongji University, JUN ZHANG, New York University and NYU Shanghai, JIN-QIANG ZHONG, Tongji University — A thick layer of fluid, water, contained in a rectangular box of a few centimeters in each direction, is deformed asymmetrically on its free surface by the imposed boundary wettability. As the fluid layer is subject to harmonic excitation at a level below the threshold of Faraday instability, an unidirectional, large-scale circulation emerges in the fluid bulk. We experimentally study its pattern, timescale needed to evolve and the underlying mechanisms.

¹Supported by the NSF of China through grant no. 11572230.

4:28PM D27.00008 Capillary deposition of advected floating particles, EMILIE DRESSAIRE, New York University, AYMERIC DEBAISIEUX, ENSTA, FEDERICO GREGORI, New York University — The deposition and aggregation of particles flowing through a confined environment can dramatically hinder the transport of suspensions. Yet, the mechanisms responsible for the deposition of particles in shear flow are not fully understood. Here, we use an experimental model system in which floating particles are advected on the surface of a water channel and deposited on fixed obstacles through attractive capillary effects. By varying the flow rate of the liquid, the wetting properties and size of the particles and obstacles, we can tune the magnitude of the capillary and hydrodynamic forces that determine the probability of deposition and the equilibrium position on the substrate. We show that arrays of obstacles can be designed to efficiently capture the floating particles advected by the flow.

4:41PM D27.00009 Writing on water with permanent markers¹ , SEPIDEH KHODAPARAST,

FRANÇOIS BOULOGNE, HOWARD A. STONE, Complex Fluids Group, Princeton University — Permanent markers create a continuous thin stain on a surface, which, after drying, can only be removed by high pressure cleaning or organic solvents. The stains of the markers are hydrophobic and thus effectively resist rinsing by water. We introduce a peeling technique based on surface tension, which benefits from this hydrophobicity, to transfer complex two-dimensional marks onto the air-water interface. As an air-water meniscus reaches the stain edge, the surface tension applies a detachment force to the thin layer. If larger than the adhesion of the stain on the substrate, the surface tension can peel off the entire layer. We examine the efficiency of this peeling method for elastic thin films in an experimental model made of thin polystyrene films of well-controlled geometrical properties adhering on clean glass substrates. We investigate the effect of film thickness and interface velocity. At low interface velocities $U < 1 \text{ mm/s}$, films of thicknesses down to 50 nm are peeled and transferred to the air-water interface with no defects. Peeling with the meniscus can be used in a large variety of applications such as water-assisted transfer printing, peel-and-stick technologies, cleaning the water proof stains without solvent and fabrication of flexible wearable electronics.

¹This research is supported by grant from Swiss National Science Foundation (P2ELP2-158896).

4:54PM D27.00010 Dynamics of surface tension driven mixing of an alcohol droplet with water. , RAJ DANDEKAR, ANURAG PANT, BABURAJ PUTHENVEETTIL, Indian Institute of Technology Madras — We study

the flow induced by the surface tension driven spreading of an ethanol droplet of radius r_d on the surface of a 5mm water layer, visualizing the flow using aluminium flakes on the surface of the water layer with backlighting and high speed imaging. The concentration of tracer aluminium particles was found to have no effect on the scaling law for spreading. The drop, when brought in contact with the water surface causes a local depression in surface tension, resulting in a thin circular region to expand radially outwards. We observe that the dimensionless radius of the expanding front ($r^* = r/r_d$) scales with the dimensionless time ($t^* = \mu r_d / \Delta\gamma$), as $r^* \sim t^{*1/4}$, where μ is the viscosity of water and $\Delta\gamma$ is the surface tension difference between water and the ethanol droplet. A scaling analysis taking the viscous and the marangoni forces into account explains the observed scaling law. Our observations differ from that in the case of continuous alcohol supply (Sánchez et al., Phys. Fluids 27, 032003, 2015) where the observed scaling law is $r^* \sim t^{*1/2}$. The expanding front radius reaches a maximum value and then decreases with time.

Sunday, November 20, 2016 2:57PM - 5:07PM –

Session D28 Particle-laden Flows: Simulations F149 - Olivier Desjardins, Cornell University

2:57PM D28.00001 A numerical study of bidisperse particles in cluster-induced

turbulence¹ , RAVI PATEL, Cornell University, BO KONG, Iowa State University, JESSE CAPECELATRO, University of Michigan, RODNEY FOX, Iowa State University, OLIVIER DESJARDINS, Cornell University — Particle-laden turbulent flow is an important feature of many diverse environmental and industrial systems. To elucidate the mechanics of these types of flows, we study cluster-induced turbulence (CIT), wherein momentum coupling between a carrier fluid and setting particles leads to turbulent-like fluctuations in various quantities of interest. In this work, simulations of CIT with bidisperse particles are presented. The flow of kinetic energy is tracked from its generation due to drag until its dissipation due to fluid viscosity and particle collisions. As suggested by Fox (2014), the particle kinetic energy is separated into a correlated turbulent kinetic energy and an uncorrelated granular energy. An overall energy balance is computed for various exchange terms to determine their relative importance and to understand the underlying physical mechanisms in bidisperse CIT. Additionally, volume fraction and velocity statistics for both particle types and the fluid are presented. From these results, the consequences on closures for Reynolds-averaged stress models of particle-laden flows are discussed.

¹National Science Foundation

3:10PM D28.00002 Dynamic subgrid-scale modeling for LES of particle-laden tur-

bulent flows¹ , MAXIME BASSENNE, GEORGE ILHWAN PARK, JAVIER URZAY, PARVIZ MOIN, Center for Turbulence Research, Stanford University — A new dynamic model is proposed for large-eddy simulations of small inertial particles in turbulent flows. The model is simple, involves no significant computational overhead, and is flexible enough to be deployed in any type of flow solvers and grids, including unstructured setups. The approach does not require any tunable parameters and is based on the use of elliptic differential filters. Particle laden isotropic turbulence and turbulent channel flow are considered. Improved agreement with direct numerical simulation results are observed in the dispersed-phase statistics. The comparisons include analyses of particle acceleration, local carrier-phase velocity, turbophoresis, and preferential-concentration metrics.

¹PSAAP-II Center at Stanford (DoE Grant 107908)

3:23PM D28.00003 Eulerian-Lagrangian study of particle resuspension by a periodically-forced impinging jet¹ , WEN WU, Queens Univ, GIOVANNI SOLIGO, CRISTIAN MARCHIOLI, Dipartimento Politecnico di Ingegneria e Architettura, Università degli Studi di Udine, ALFREDO SOLDATI, Institut für Strömungsmechanik und Wärmeübertragung, TU Wien, UGO PIOMELLI, Queens Univ — In this work, we investigate the mechanisms that govern particle resuspension in an impinging flow over surfaces covered with mobile sediments. An Eulerian-Lagrangian approach based on large-eddy simulation of turbulence, and one-way coupling Lagrangian-tracking of particles, is used to model a vertical impinging jet, to which a sequence of periodically-forced azimuthal vortices is superposed. We show how the dynamics of sediments is governed by their interaction with the turbulent structures (including the large-scale vortices) and the separated flow. After initial lift-up from the impingement surface, particles are accumulated in regions where near-wall vortices roll around the impinging azimuthal vortex, forming rib-like structures that either propel particles away from the azimuthal vortex or entrap them in the shear layer between the azimuthal and secondary vortices. We demonstrate that these trapped particles are more likely to reach the outer flow region and generate a persistent cloud of airborne particles.

¹WW thanks the ACRI Young Investigator Training Program (YITP). GS gratefully acknowledges the University of Udine for grant Mobilità Europea ed extra Europa per ricerche per tesi.

3:36PM D28.00004 A Comparative Study of Euler-Euler and Euler-Lagrange Mesoscale Simulations of Moderately Dense Cluster-induced Gas-Particle Turbulence, BO KONG, Ames Laboratory - USDOE, Iowa State Univ., RAVI PATEL, Cornell University, JESSE CAPECELATRO, University of Michigan, OLIVIER DESJARDINS, Cornell University, RODNEY FOX, Iowa State University — Recently Euler-Lagrange (EL) approaches have gained considerable popularity, but the computational cost of resolved EL simulations is often prohibitively high. Therefore, Euler-Euler (EE) approaches, such as kinetic-theory-based two-fluid models (TFM), remain the major workhorse in this area. However, the hydrodynamic assumption in TFM has been proven invalid by many experiments and simulations, especially for dilute particles. Previously, the EE Anisotropic Gaussian approach (EE-AG) has been shown to produce good agreement for key statistic results with EL simulations when particles are dilute. In this work, a novel EE-AG/TFM hybrid solution algorithm, based on different contributions to the particle-phase spatial fluxes, is proposed and implemented in an open-source CFD package. Fully resolved mesoscale simulations of cluster-induced turbulence with moderately dense particles are performed. The detailed comparisons with EL simulations demonstrate that this new EE method can accurately capture the dynamics of the gas-solid flows and produce results comparable to the EL simulations.

3:49PM D28.00005 Direct Numerical Simulation of Poly-dispersed Solid-Fluid Systems, ERICH ESSMANN, PEI SHUI, University of Edinburgh, RAMA GOVINDARAJAN, TIFR-Hyderabad, STEPHANE POPINET, Universit Pierre et Marie Curie, PRASHANT VALLURI, University of Edinburgh — The fluid dynamics of poly-dispersed solid – fluid systems are of great importance, particularly is the behaviour of methane clathrates slurries. In this work, a framework is being developed for the direct numerical simulation of these systems. We have extended the Gerris software package of (Popinet et al, 2003). In our solid solver, Gerris Immersed Solid Solver (GISS), to account for collisions we have implemented a novel contact model (Ness & Sun, 2016) for solid-solid interactions. A composite contact model is being used, in which each solid in the domain is divided into two regions. The outer region uses a Hookean repulsive and a lubrication force model to simulate contact. The inner region uses a constraint based contact model to ensure that the numerical overlap of the solids is not excessive. We have validated our methodology against published experimental data. Particularly, we compared the chaotic motion of an ellipsoidal solid in an ideal fluid (Aref, 1993) to that predicted by GISS and the settling behaviour of two colliding spheres of different densities (Zhao, 2003). The validated extensions will allow us to compare previous results from GISS to regimes in which solid-solid contact is important.

4:02PM D28.00006 Large-eddy simulation of charged particle flows to model sandstorms¹, MUSTAFA RAHMAN, WAN CHENG, RAVI SAMTANEY, King Abdullah University of Science and Technology — Intense electric fields and lightning have been observed in sandstorms. It is proposed to investigate the physical mechanisms essential for production and sustenance of large-scale electric fields in sandstorms. Our central hypothesis is that the turbulent transport of charged sand particles is a necessary condition to attain sustained large-scale electric fields in sandstorms. Our investigation relies on simulating turbulent two-phase (air and suspended sand particles) flows in which the flow of air is governed by the filtered Navier-Stokes equations with a subgrid-scale model in a Large-Eddy-Simulation setting, while dust particles are modeled using the Eulerian approach using a version of the Direct Quadrature Method of Moments. For the fluid phase, the LES of incompressible turbulent boundary layer employs stretched spiral vortex subgrid-scale model and a virtual wall model similar to the work of Cheng, Pullin & Samtaney (J. Fluid Mech. 2015). We will quantify the effects of different sand particle distributions, and turbulent intensities on the root-mean-square of the generated electric fields.

¹Supported by KAUST OCRF under award number URF/1/1704-01-01. The supercomputer Shaheen at KAUST is used for all simulations.

4:15PM D28.00007 The preferential erosion and deposition of heavy particles over erodible beds, SCOTT SALESKY, MARCO GIOMETTO, University of British Columbia, MICHAEL LEHNING, Swiss Federal Institute of Technology, MARC PARLANGE, University of British Columbia — The erosion, transport, and deposition of heavy particles over erodible beds by turbulent flow is a significant process in the context of sediment transport, aeolian processes, and snow transport in alpine and polar regions. While it is well-known that terrain features can lead to spatially inhomogeneous deposition velocities, a systematic study considering the effects of terrain and particle properties has not been conducted to date using large eddy simulation (LES). Using a recently developed Eulerian finite-volume model for the transport of heavy particles over complex terrain in LES, we perform simulations of the transport, erosion, and deposition of heavy particles over idealized surface topography. A new model for particle ejection in the saltation layer subject to the constraints of energy and momentum conservation is adapted for use in an Eulerian framework. A suite of simulations is conducted in order to explore the governing parameters relevant for erosion and deposition (e.g. Stokes number, Rouse number, Shields number, surface cohesion) and to investigate the influence of the mean flow vs. turbulent fluxes for the observed erosion and deposition patterns. Implications for model development will be highlighted, and numerical considerations will be discussed.

4:28PM D28.00008 High resolution simulations of down-slope turbidity currents into stratified saline ambient, RAPHAEL OUILLON, SENTHIL RADHAKRISHNAN, ECKART MEIBURG, UC Santa Barbara, BRUCE SUTHERLAND, University of Alberta, Edmonton — In this work we explore the properties of turbidity currents moving down a slope into a stratified saline ambient through highly resolved 3D Navier-Stokes simulations. Turbidity events are difficult to measure and to replicate experimentally for a wide range of parameters, but they play a key role in ocean, lake or river sediment transport. Our objectives are to improve on previous numerical studies, obtain quantitative data in a more controlled environment than current experimental set-ups, and combine results with analytical arguments to build physics-based scaling laws. We validate our results and propose a simple scaling law to predict the velocity of the front down a slope for any stratification. We also compute a time and space dependent entrainment of ambient fluid and highlight its strong variability. We then introduce a predictable scaling law for the intrusion depth that does not depend on an averaged entrainment and uses it as a verification tool instead. Finally, we show that the ratio of Stokes losses in the local flow around individual particles to dissipative losses of the large scale flow determines the ability of the flow to convert potential energy into kinetic energy. For different parameters, either mechanism can dominate the dynamics of the flow.

4:41PM D28.00009 Solids mixing in bubbling fluidized beds: CFD-based analysis of Bubble Dynamics and Time Scales, AKHILESH BAKSHI, Massachusetts Institute of Technology, CHRISTOS ALTANTZIS, Massachusetts Institute of Technology, National Energy Technology Laboratory, AHMED GHONIEM, Massachusetts Institute of Technology — In bubbling fluidized bed reactors, solids mixing is critical because it directly affects fuel segregation and residence time. However, there continues to be a lack of understanding because (a) most diagnostic techniques are only feasible in lab-scale setups and (b) the dynamics are sensitive to the operating conditions. Thus, quantitative estimates of mixing (e.g., dispersion coefficient, mixing indices) often span orders of magnitude although it is well accepted that the micro-mixing and gross circulation of solid particles is driven by bubble motion. To quantify this dependence, solids mixing is investigated using fine-grid 3D CFD simulations of a large 50 cm diameter fluidized bed. Detailed diagnostics of the computed flow-field data are performed using MS3DATA, a tool that we developed to detect and track bubbles, and the solids motion is correlated with the spatial and size distribution of bubbles. This study will be useful for quantifying mixing at commercial scales.

4:54PM D28.00010 Simulation of Collision of Arbitrary Shape Particles with Wall in a Viscous Fluid. , FAZLOLAH MOHAGHEGH, H. S. UDAYKUMAR, University of Iowa — Collision of finite size arbitrary shape particles with wall in a viscous flow is modeled using immersed boundary method. A potential function indicating the distance from the interface is introduced for the particles and the wall. The potential can be defined by using either an analytical expression or level set method. The collision starts when the indicator potentials of the particle and wall are overlapping based on a minimum cut off. A simplified mass spring model is used in order to apply the collision forces. Instead of using a dashpot in order to damp the energy, the spring stiffness is adjusted during the bounce. The results for the case of collision of a falling sphere with the bottom wall agrees well with the experiments. Moreover, it is shown that the results are independent from the minimum collision cut off distance value. Finally, when the particle's shape is ellipsoidal, the rotation of the particle after the collision becomes important and noticeable: At low Stokes number values, the particle almost adheres to the wall in one side and rotates until it reaches the minimum gravitational potential. At high Stokes numbers, the particle bounces and loses the energy until it reaches a situation with low Stokes number.

Sunday, November 20, 2016 2:57PM - 4:54PM –

Session D29 CFD: Applications II F150 - Gerald Recktenwald, Portland State University

2:57PM D29.00001 3D simulation of liquid-gas flows with contact lines and applications to drop impact on a fiber , SHENG WANG, OLIVIER DESJARDINS, Cornell University — In this talk, we propose a numerical approach to simulate liquid-gas flows with contact lines, then, apply it to investigate drop impact on a horizontal fiber. This approach combines a conservative level set method to capture the interface, an immersed boundary method to represent the curved boundary, and a curvature boundary method to treat the contact lines. The simulation results are compared with experimental results. Two key aspects are investigated: the critical velocity for a drop to be captured by a fiber, and the topology of the impacting drop. Numerical results show good qualitative agreement with experiments in predicting the critical velocity. A regime map of drop topology based on the Weber number and drop-to-fiber diameter ratio is compared to the experimental results.

3:10PM D29.00002 High-performance sailboat hydrofoil optimization using vortex lattice methods, and the effects of free-stream turbulence , GIANLUCA MENEGHELLO, MIT, POORIYA BEYHAGHI, THOMAS BEWLEY, UCSD — The identification of an optimized hydrofoil shape depends on an accurate characterization of both its geometry and the incoming, turbulent, free-stream flow. We analyze this dependence using the computationally inexpensive vortex lattice model implemented in AVL, coupled with the recently developed global, derivative-free optimization algorithm implemented in Δ – DOGS. Particular attention will be given to the effect of the free-stream turbulence level — as modeled by a change in the viscous drag coefficients — on the optimized values of the parameters describing the three dimensional shape of the foil. Because the simplicity of AVL, when contrasted with more complex and computationally expensive LES or RANS models, may cast doubts on its usefulness, its validity and limitations will be discussed by comparison with water tank measurement, and again taking into account the effect of the uncertainty in the free-stream characterization.

3:23PM D29.00003 Application of RANS Simulations for Contact Time Predictions in Turbulent Reactor Tanks for Water Purification Process , CASSANDRA NICKLES, MATTHEW GOODMAN, JOSE SAEZ, EMIN ISSAKHANIAN, Loyola Marymount University — California's current drought has renewed public interest in recycled water from Water Reclamation Plants (WRPs). It is critical that the recycled water meets public health standards. This project consists of simulating the transport of an instantaneous conservative tracer through the WRP chlorine contact tanks. Local recycled water regulations stipulate a minimum 90-minute modal contact time during disinfection at peak dry weather design flow. In-situ testing is extremely difficult given flowrate dependence on real world sewage line supply and recycled water demand. Given as-built drawings and operation parameters, the chlorine contact tanks are modeled to simulate extreme situations, which may not meet regulatory standards. The turbulent flow solutions are used as the basis to model the transport of a turbulently diffusing conservative tracer added instantaneously to the inlet of the reactors. This tracer simulates the transport through advection and dispersion of chlorine in the WRPs. Previous work validated the models against experimental data. The current work shows the predictive value of the simulations.

3:36PM D29.00004 Statistical data generated through CFD to aid in the scale-up of shear sensitive processes. , IRFAN KHAN, SHANKHADEEP DAS, MIKE CLOETER, PAUL GILLIS, MICHAEL POINDEXTER, The Dow Chemical Company — A number of industrial processes are considered shear-sensitive, where the product quality depends on achieving the right balance between mixing energy input and the resulting strain rate distribution in the process. Examples of such industrial processes are crystallization, flocculation and suspension polymerization. Scale-up of such processes are prone to a number of challenges including the optimization of mixing and shear rate distribution in the process. Computational Fluid Dynamics (CFD) can be a valuable tool to aid in the process scale-up; however for modeling purpose, the process will often need to be simplified appropriately to reduce the computational complexity. Commercial CFD tools with appropriate Lagrangian particle tracking models can be used to gather statistical data such as maximum strain rate distribution and maximum number of passes through a specific strain rate. This presentation will discuss such statistical tools and their application to a model scale-up problem

3:49PM D29.00005 Assess and improve the sustainability of water treatment facility using Computational Fluid Dynamics. , JIE ZHANG, ANDRES TEJADA-MARTINEZ, University of South Florida, HONGXIA LEI, City of Tampa Water Department, QIONG ZHANG, University of South Florida — Fluids problems in water treatment industry are often simplified or omitted since the focus is usually on chemical process only. However hydraulics also plays an important role in determining effluent water quality. Recent studies have demonstrated that computational fluid dynamics (CFD) has the ability to simulate the physical and chemical processes in reactive flows in water treatment facilities, such as in chlorine and ozone disinfection tanks. This study presents the results from CFD simulations of reactive flow in an existing full-scale ozone disinfection tank and in potential designs. Through analysis of the simulation results, we found that baffling factor and CT10 are not optimal indicators of disinfection performance. We also found that the relationship between effluent CT (the product of disinfectant concentration and contact time) obtained from CT transport simulation and baffling factor depends on the location of ozone release. In addition, we analyzed the environmental and economic impacts of ozone disinfection tank designs and developed a composite indicator to quantify the sustainability of ozone disinfection tank in technological, environmental and economic dimensions.

4:02PM D29.00006 Transition to complex dynamics in the cubic lid-driven cavity , KE

WU, JUAN M. LOPEZ, BRUNO D. WELFERT, JASON YALIM, Arizona State Univ — The cubic lid-drive cavity flow is simulated numerically by a Chebyshev collocation method, focusing on the onset of unsteadiness. The onset is directly to intermittent chaos. This has been reported by others in 2014, but they were unable to explain why, in such a simple geometry at quite modest Reynolds numbers ($Re \approx 1930$), the flow should go from being steady directly to intermittent chaos. In this presentation, we show that the reason has three components: instability of the steady flow is subcritical, breaking of the reflection symmetry about the spanwise midplane, and spanwise confinement. These ingredients lead to there being no locally attracting states for Re beyond which the steady state loses stability, and the intermittent bursts are excursions shadowing unstable and stable manifolds of the (unstable) saddle local states. We show that this comes about because the instability of the steady state is close to a subcritical double Hopf bifurcation, and note that very recent theory on such bifurcations shows that they have associated dynamics that captures all the observed complexity in the cubic lid-driven cavity in the neighborhood of its primary instability.

4:15PM D29.00007 Numerical study and validation on a two-phase ejector flow using R134a refrigerant¹ , SUNGHOON BAEK, SIMON SONG, Hanyang University

— An ejector is a pumping device that uses a low pressure jet flow to entrain a low-momentum secondary flow, and the two flows are mixed and pressurized in a mixing tube and a diffuser. When the ejector replaces an expansion valve in a standard refrigeration cycle, a compression work can be saved by the pumping effect and the efficiency of the cycle is known to be improved. However, the details of flow characteristics in the ejector are still unknown due to difficulties in experiments and complex flow phenomena. We numerically studied a supersonic ejector flow of R134a refrigerant, and validated the results against experimental data. As a results, we found that combinations of mixture, realizable k-epsilon, evaporation-condensation models, and energy equation are suitable to predict the ejector performance in a design point of view.

¹This work was supported by the National Research Foundation of Korea(NRF) grant funded by the Korea government(MSIP) (No. 2016R1A2B3009541)

4:28PM D29.00008 ABSTRACT WITHDRAWN —

4:41PM D29.00009 Flow characteristics of infinite-span wings with wavy leading edges¹ , RAFAEL PEREZ-TORRO, JAE-WOOK KIM, Univ of Southampton

— Implicit LES computations are performed for an infinite-span wing based on the NACA0021 aerofoil section with a sinusoidal wavy leading edge (WLE). At $Re_\infty = 1.2 \times 10^5$ and $M_\infty = 0.3$, the computations performed in this study show that three-dimensional laminar separation bubbles (LSBs) form at troughs of the undulated wing. Prior to stall, LSBs can be found in all troughs. However, past the stall angle, LSBs tend to group together in a collocated fashion, leaving regions of complete separation in between groups where a separated shear layer (SSL) is formed. It is found that the size of the LSB group is highly dependent on the number of WLE wavelengths used in the spanwise-periodic domain. The LSB group formation process is investigated by means of simulations where the geometry is slowly pitched from an angle of attack of $\alpha = 10^\circ$ to $\alpha = 20^\circ$. The study also includes the analysis of instantaneous flow fields using Proper Orthogonal Decomposition (POD) and Dynamic Mode Decomposition (DMD) techniques.

¹The authors acknowledge the HPC facilities of the UK National Supercomputer Archer via the support of the UK Turbulence Consortium (EP/L000261/1) and the local Iridis4 at the University of Southampton.

Sunday, November 20, 2016 2:57PM - 5:07PM —

Session D30 Granular Flows: Computation and Modeling F151 - Joe Goddard, University of

California, San Diego

9500 Gilman Dr.

La Jolla, CA 92093-0411 USA

100, Inha-ro, Nam-gu, Incheon 22212, Korea

2:57PM D30.00001 Cahn-Hilliard Regularization of the "mu(I)" Rheology , JOE GODDARD

DARD, University of California, San Diego, JAESUNG LEE, INHA Technical College — Recently Barker et al. [J. Fluid Mech. 779 (2015) 794-818] have shown that the popular $\mu(I)$ model for the viscoplasticity of granular media is ill-posed, exhibiting short wave-length instabilities of the Hadamard variety. As one possible regularization of the model, we employ the dissipative analog of the classical Cahn-Hilliard (CH) model, with dissipation potential given by: $\psi(\nabla v, \nabla \nabla v) = \psi_0(\mathbf{D}) + k|\nabla \nabla v|^2$, with $\mathbf{D} = \text{Sym}(\nabla v)$ and $k > 0$, with stress for the standard $\mu(I)$ model given by $\partial \psi_0 / \partial \mathbf{D}$, and with hyperstress given by $\partial \psi / \partial \nabla \nabla v$. Following the linear-stability analysis of Barker et al. of the momentum balance and continuity equation, we obtain a modification of their dispersion relation giving growth rate in terms of spatial wave number. It is found that the higher-gradient terms in the CH model lead to a large wave number cut-off of the instability, so that the model provides a possibly useful regularization of the $\mu(I)$ model.

3:10PM D30.00002 Embryo as an active granular fluid: stress-coordinated cellular constriction chains , MICHAEL HOLCOMB, Department of Physics, Texas Tech University , GUO-JIE GAO, Department of Mathematical and Systems Engineering, Shizuoka University, JEFFREY THOMAS, Department of Cell Biology and Biochemistry, Texas Tech University Health Sciences Center, JERZY BLAWZDZIEWICZ, Department of Mechanical Engineering, Texas Tech University

— Mechanical stress plays an intricate role in gene expression in individual cells and sculpting of developing tissues. Motivated by our observation of the cellular constriction chains (CCCs) during the initial phase of ventral furrow formation in the *Drosophila melanogaster* embryo, we propose an active granular fluid (AGF) model that provides valuable insights into cellular coordination in the apical constriction process. In our model, cells are treated as circular particles connected by a predefined force network, and they undergo a random constriction process in which the particle constriction probability P is a function of the stress exerted on the particle by its neighbors. We find that when P favors tensile stress, constricted particles tend to form chain-like structures. In contrast, constricted particles tend to form compact clusters when P favors compression. A remarkable similarity of constricted-particle chains and CCCs observed *in vivo* provides indirect evidence that tensile-stress feedback coordinates the apical constriction activity.

3:23PM D30.00003 Theory for Indirect Conduction in Dense, Gas-Solid Systems

AARON LATTANZI, CHRISTINE HRENYA, University of Colorado — Heat transfer in dense gas-solid systems is dominated by conduction, and critical to the operation of rotary-kilns, catalytic cracking, and heat exchangers with solid particles as the heat transfer fluid. In particular, the indirect conduction occurring between two bodies separated by a thin layer of fluid can significantly impact the heat transfer within gas-solid systems. Current state-of-the-art models for indirect conduction assume that particles are surrounded by a static "fluid lens" and that one-dimensional conduction occurs through the fluid lens when the lens overlaps another body. However, attempts to evaluate the effect of surface roughness and fluid lens thickness (theoretical inputs) on indirect conduction have been restricted to static, single-particle cases. By contrast, here we quantify these effects for dynamic, multi-particle systems. This analysis is compared to outputs from computational fluid dynamics and discrete element method (CFD-DEM) simulations of heat transfer in a packed bed and flow down a heated ramp. Analytical predictions for model sensitivity are found to be in agreement with simulation results and differ greatly from the static, single-particle analysis. Namely, indirect conduction in static systems is found to be most sensitive to surface roughness, while dynamic systems are sensitive to the fluid lens thickness.

3:36PM D30.00004 Discrete particle modelling of granular roll waves¹

JONATHAN TSANG, STUART DALZIEL, NATHALIE VRIEND, University of Cambridge — A granular current flowing down an inclined chute or plane can undergo an instability that leads to the formation of surface waves, known as roll waves. Examples of roll waves are found in avalanches and debris flows in landslides, and in many industrial processes. Although related to the Kapitza instability of viscous fluid films, granular roll waves are not yet as well understood. Laboratory experiments typically measure the surface height and velocity of a current as functions of position and time, but they do not give insight into the processes below the surface: in particular, the possible formation of a boundary layer at the free surface as well as the base. To overcome this, we are running discrete particle model (DPM) simulations. Simulations are validated against our laboratory experiments, but they also allow us to examine a much larger range of parameters, such as material properties, chute geometry and particle size dispersity, than that which is possible in the lab. We shall present results from simulations in which we vary particle size and dispersity, and examine the implications on roll wave formation and propagation. Future work will include simulations in which the shape of the chute is varied, both cross-sectionally and in the downstream direction.

¹EPSRC studentship (Tsang) and Royal Society Research Fellowship (Vriend)

3:49PM D30.00005 A nonlinear description of the viscosity and dilatancy of granular suspensions

DAVIDE MONSORNO, CHRISTOS VARSAKELIS, MILTIADIS PAPALEXANDRIS, Université catholique de Louvain — In the first part of this talk we present a rheology law for granular suspensions based on the representation theorem of isotropic tensors. The proposed law has a number of desirable properties, namely, it is free of singularities, it vanishes at equilibrium, and it predicts non-zero bulk viscosity as well as shear-rate dependent normal viscous stresses. Next, we present an evolution equation for the volume fraction of the granular phase that can describe the dilatancy of granular suspensions in a consistent manner. Its derivation is based upon the introduction of the volume fraction and its gradient as internal degrees of freedom. The resulting model has been applied to a number of well-known test cases, such as plane-shear and pressure-driven flows, and its predictions are presented and compared with experimental data. In particular, we show that this model can successfully predict important features of granular suspensions such as normal stress differences and particle migration.

4:02PM D30.00006 Numerical study of cavitation and pinning effects due to gas injection through a bed of particles: application to a radial-flow moving-bed reactor.

GUILLAUME VINAY, FELAURYS VASQUEZ, FLORENCE RICHARD, IFP Energies nouvelles, APPLIED MECHANICS TEAM — In the petroleum and chemical industries, radial-flow moving-bed reactors are used to carry out chemical reactions such as catalytic reforming. Radial-flow reactors provide high capacity without increased pressure drop or greatly increased vessel dimensions. This is done by holding the catalyst in a basket forming an annular bed, and causing the gas to flow radially between the outer annulus and the central tube. Catalyst enter the top of the reactor, move through the vessel by gravity to the bottom where it is removed and then regenerated. Within the catalytic bed, the combined effects of particles motion and radial injection of the gas may lead to cavitation and pinning phenomenon that may clearly damage the reactor. We study both cavitation and pinning effects using an in-house numerical software, named Peligriff (www.peligriff.com/), designed to simulate particulate flows at different scales; from the particle scale, where fluid/particle interactions are directly solved, to the particles suspension scale where the fluid/solid interactions are modeled. In the past, theoretical and experimental studies have already been conducted in order to understand the way cavitation and pinning occur. Here, we performed simulations involving a few thousands of particles aiming at reproducing experimental experiments. We will present comparisons between our numerical results and experimental results in terms of pressure drop, velocity, porosity.

4:15PM D30.00007 ABSTRACT WITHDRAWN —

4:28PM D30.00008 A thermodynamically consistent model for granular-fluid mixtures considering pore pressure evolution and hypoplastic behavior¹

JULIAN HESS, YONGQI WANG, Technische Universität Darmstadt — A new mixture model for granular-fluid flows, which is thermodynamically consistent with the entropy principle, is presented. The extra pore pressure described by a pressure diffusion equation and the hypoplastic material behavior obeying a transport equation are taken into account. The model is applied to granular-fluid flows, using a closing assumption in conjunction with the dynamic fluid pressure to describe the pressure-like residual unknowns, hereby overcoming previous uncertainties in the modeling process. Besides the thermodynamically consistent modeling, numerical simulations are carried out and demonstrate physically reasonable results, including simple shear flow in order to investigate the vertical distribution of the physical quantities, and a mixture flow down an inclined plane by means of the depth-integrated model. Results presented give insight in the ability of the deduced model to capture the key characteristics of granular-fluid flows.

¹We acknowledge the support of the Deutsche Forschungsgemeinschaft (DFG) for this work within the project number WA 2610/3-1.

4:41PM D30.00009 A non-local plasticity theory for slow granular flows

PRABHU R NOTT, Indian Institute of Science — Recent studies on dense granular materials have shown evidence of non-locality in the mechanical response, wherein the motion of an intruder is aided by shearing the material far from it. This behaviour is not explained by classical plasticity theories, which also have other serious failings. Non-local theories proposed earlier are either of phenomenological origin, or based on the introduction of an additional field variable whose mechanical origin is debatable. Here we present a non-local plasticity theory whose mechanical origin is easy to comprehend, involves no additional field variables, and captures rather simply the physical picture of plastic events in a spatial point influencing its neighbourhood. Most crucially, the theory is able to predict the kinematics of simple shear flows, in particular the exponentially decaying velocity profile in simple shear, and shear-induced dilatancy. Finally, our non-local theory plasticity theory is Hadamard well-posed, a significant improvement over the local theories.

4:54PM D30.00010 ABSTRACT WITHDRAWN —

Sunday, November 20, 2016 2:57PM - 4:54PM —

Session D31 Experimental Techniques - 3D Particle Velocimetry F152 - Alex Techet, MIT

2:57PM D31.00001 Robust 4 Camera 3D Synthetic Aperture PIV , ABHISHEK BAJPAYEE,

ALEXANDRA TECHET, MIT — We present novel processing techniques which allow for robust 4 camera 3D synthetic aperture (SA) PIV. These pre and post processing techniques, applied to raw images and reconstructed volumes, significantly improve SA reconstruction SNR values and consequently allow for accurate SAPIV velocity fields. SA, or light field, PIV has typically required 8 or 9 cameras in order to achieve high reconstruction quality and velocity field reconstruction quality values, Q and Q_v , respectively. This is primarily because the effective signal to noise ratio (SNR) of refocused images, when using traditional multiplicative or additive refocusing techniques, increases with the number of cameras being used. However, tomographic reconstruction (used with TomoPIV), is able to achieve relatively high SNR reconstructions using 4 or 5 cameras owing to its iterative but significantly more computationally expensive algorithm. Our processing techniques facilitate better recovery of relevant information in SA reconstructions using only 4 views. As a result, we no longer have to trade setup cost and complexity (number of cameras) for computational speed of the reconstruction algorithm.

3:10PM D31.00002 Tomographic Aperture-Encoded Particle Tracking Velocimetry:

A New Approach to Volumetric PIV , DAN TROOLIN, AARON BOOMSMA, WING LAI, STAMATIOS POTHOS, TSI Incorporated, FLUID MECHANICS RESEARCH INSTRUMENTS TEAM — Volumetric velocity fields are useful in a wide variety of fluid mechanics applications. Several types of three-dimensional imaging methods have been used in the past to varying degrees of success, for example, 3D PTV (Maas et al., 1993), DDPIV (Peireira et al., 2006), Tomographic PIV (Elsinga, 2006), and V3V (Troolin and Longmire, 2009), among others. Each of these techniques has shown advantages and disadvantages in different areas. With the advent of higher resolution and lower noise cameras with higher stability levels, new techniques are emerging that combine the advantages of the existing techniques. This talk describes a new technique called Tomographic Aperture-Encoded Particle Tracking Velocimetry (TAPTV), in which segmented triangulation and diameter tolerance are used to achieve three-dimensional particle tracking with extremely high particle densities (on the order of $\text{ppp} = 0.2$ or higher) without the drawbacks normally associated with ghost particles (for example in TomoPIV). The results are highly spatially-resolved data with very fast processing times. A detailed explanation of the technique as well as plots, movies, and experimental considerations will be discussed.

3:23PM D31.00003 High fidelity digital inline holographic PTV for 3D flow measurements: from microfluidics to wall-bounded turbulence , JIARONG HONG, MOSTAFA TOLOUI, KEVIN MALLERY, University of Minnesota — Three-dimensional PIV and PTV provides the most comprehensive flow information for unraveling the physical phenomena in a wide range of fluid problems, from microfluidics to wall-bounded turbulent flows. Compared with other commercialized 3D PIV techniques, such as tomographic PIV and defocusing PIV, the digital inline holographic PTV (namely DIH-PTV) provides 3D flow measurement solution with high spatial resolution, low cost optical setup, and easy alignment and calibration. Despite these advantages, DIH-PTV 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, small sampling volume and expensive computations, limiting its broad use for 3D flow measurements. Here we will report our latest work on improving DIH-PTV method through an integration of deconvolution algorithm, iterative removal method and GPU computation to overcome some of abovementioned limitations. We will also present the application of our DIH-PTV for measurements in the following sample cases: (i) flows in bio-filmed microchannel with $50\text{-}60 \mu\text{m}$ vector spacing within sampling volumes of $1 \text{ mm (streamwise)} \times 1 \text{ mm (wall-normal)} \times 1 \text{ mm (spanwise)}$; (ii) turbulent flows over smooth and rough surfaces (1.1 mm vector spacing within $15 \text{ mm} \times 50 \text{ mm} \times 15 \text{ mm}$); (iii) 3D distribution and kinematics of inertial particles in turbulent air duct flow.

3:36PM D31.00004 Smartphone based Tomographic PIV using colored shadows , ANDRES A. AGUIRRE-PABLO, MESHAL K. ALARFAJ, ER QIANG LI, SIGURDUR T. THORODDSEN, King Abdullah University of Science and Technology — We use low-cost smartphones and Tomo-PIV, to reconstruct the 3D-3C velocity field of a vortex ring. The experiment is carried out in an octagonal tank of water with a vortex ring generator consisting of a flexible membrane enclosed by a cylindrical chamber. This chamber is pre-seeded with black polyethylene microparticles. The membrane is driven by an adjustable impulsive air-pressure to produce the vortex ring. Four synchronized smartphone cameras, of 40 Mpx each, are used to capture the location of particles from different viewing angles. We use red, green and blue LED's as backlighting sources, to capture particle locations at different times. The exposure time on the smartphone cameras are set to 2 seconds, while exposing each LED color for about $80 \mu\text{s}$ with different time steps that can go below $300 \mu\text{s}$. The timing of these light pulses is controlled with a digital delay generator. The backlight is blocked by the instantaneous location of the particles in motion, leaving a shadow of the corresponding color for each time step. The image then is preprocessed to separate the 3 different color fields, before using the MART reconstruction and cross-correlation of the time steps to obtain the 3D-3C velocity field. This proof of concept experiment represents a possible low-cost Tomo-PIV setup.

3:49PM D31.00005 3D-PTV around Operational Wind Turbines , IAN BROWNSTEIN, JOHN DABIRI, Stanford — Laboratory studies and numerical simulations of wind turbines are typically constrained in how they can inform operational turbine behavior. Laboratory experiments are usually unable to match both pertinent parameters of full-scale wind turbines, the Reynolds number (Re) and tip speed ratio, using scaled-down models. Additionally, numerical simulations of the flow around wind turbines are constrained by the large domain size and high Re that need to be simulated. When these simulations are preformed, turbine geometry is typically simplified resulting in flow structures near the rotor not being well resolved. In order to bypass these limitations, a quantitative flow visualization method was developed to take in situ measurements of the flow around wind turbines at the Field Laboratory for Optimized Wind Energy (FLOWE) in Lancaster, CA. The apparatus constructed was able to seed an approximately $9\text{m} \times 9\text{m} \times 5\text{m}$ volume in the wake of the turbine using artificial snow. Quantitative measurements were obtained by tracking the evolution of the artificial snow using a four camera setup. The methodology for calibrating and collecting data, as well as preliminary results detailing the flow around a 2kW vertical-axis wind turbine (VAWT), will be presented.

4:02PM D31.00006 Uncertainty quantification in volumetric Particle Image Velocimetry., SAYANTAN BHATTACHARYA, Purdue University, JOHN CHARONKO, Los Alamos National Laboratory, PAVLOS VLA-CHOS, Purdue University — Particle Image Velocimetry (PIV) uncertainty quantification is challenging due to coupled sources of elemental uncertainty and complex data reduction procedures in the measurement chain. Recent developments in this field have led to uncertainty estimation methods for planar PIV. However, no framework exists for three-dimensional volumetric PIV. In volumetric PIV the measurement uncertainty is a function of reconstructed three-dimensional particle location that in turn is very sensitive to the accuracy of the calibration mapping function. Furthermore, the iterative correction to the camera mapping function using triangulated particle locations in space (volumetric self-calibration) has its own associated uncertainty due to image noise and ghost particle reconstructions. Here we first quantify the uncertainty in the triangulated particle position which is a function of particle detection and mapping function uncertainty. The location uncertainty is then combined with the three-dimensional cross-correlation uncertainty that is estimated as an extension of the 2D PIV uncertainty framework. Finally the overall measurement uncertainty is quantified using an uncertainty propagation equation. The framework is tested with both simulated and experimental cases. For the simulated cases the variation of estimated uncertainty with the elemental volumetric PIV error sources are also evaluated. The results show reasonable prediction of standard uncertainty with good coverage.

4:15PM D31.00007 Studying Vortex Dynamics of Rotating Convection with High-resolution PIV Measurement¹, HAO FU, Nanjing University & Chinese Academy of Sciences, SHIWEI SUN, YU WANG, BOWEN ZHOU, YUAN WANG, Nanjing University — A novel experimental setup for studying vortex dynamics in rotating Rayleigh-Benard convection has been made in School of Atmospheric Sciences, Nanjing University. With water as the working fluid, three lasers with different frequencies and the corresponding three CCDs have been placed to complete 2D2C (two dimensions, two components) PIV measurement. The lasers are fixed on two crossing guiding ways and can move up and down to scan the flow field. An algorithm has been made to reconstruct 3D velocity field based on multiple 2D2C PIV data. This time, we are going to present the details of this new machine and algorithm, as well as some scientific understanding of vortex dynamics owing to this high-resolution velocity measurement system.

¹This work was supported by LMSWE Lab Funding No.14380001”

4:28PM D31.00008 A Laser Sheet Self-Calibration Method for Scanning PIV, ANNA N. KNUTSEN, JAMES R. DAWSON, Norwegian University of Science and Technology (NTNU), JOHN M. LAWSON, Max Planck Institute for Dynamics and Self-Organisation, NICHOLAS A. WORTH, Norwegian University of Science and Technology (NTNU) — A laser sheet self-calibration method for scanning PIV has been developed to replace the current laser sheet calibration, which is complex, time consuming and very sensitive to misalignment of the optics or cameras during experiments. The new calibration method is simpler, faster and crucially more robust. The concept behind the method is to traverse a laser sheet through the measurement volume, take a series of images from two different views, and calculate the global 3D particle locations. This information is used to find the real space coordinates of the measurement volume and the orientation and width of the laser sheets. The spatial location of the particles is found by object matching and triangulation. The light intensity in the laser sheet has an approximately Gaussian shape, and the illumination of one particle which will be illuminated multiple times during the scan will thus vary as the sheet is scanned across the measurement volume. The thickness of the laser sheet is calculated by identifying the variation of illumination of the particles during a scan and fitting this to a Gaussian shaped curve, while the orientation is found using a least square fit. The accuracy of the new method will be presented with respect to both synthetic and experimental data.

4:41PM D31.00009 Tracer Particle Response in High-Gradient Flow, JOSHUA HERZOG, DAVID ROTHAMER, Univ of Wisconsin, Madison — Many laser-based fluid velocity measurements depend on the motion of tracer particles seeded into the flow. In most cases, the tracers are assumed to follow the flow exactly. However, this is not always the case. The actual motion of a tracer particle is dependent on the properties of both the particle and the fluid surrounding it. Previous analysis for spherical particles in the Stokes regime (assumes $Re \ll 1$) shows that the absolute difference between the particle and fluid velocity exponentially decays in time, with the relaxation time constant dependent on particle diameter, free stream velocity, Reynolds number, and both particle and fluid mass density. For all cases, it is necessary to accurately describe the physics of the tracer particle motion to perform rigorous quantitative studies with particle-based techniques. This study aims to measure and describe particle response to a step change in velocity in a uniform flow. Velocity profiles of solid tracer particles ranging from 300 to 3800 nm in diameter, with initial particle Reynolds numbers up to 100, were measured in a shock tube using particle image velocimetry. The goal of this study is to assess velocity relaxation estimates and assumptions for particle-based velocimetry techniques.

Sunday, November 20, 2016 2:57PM - 5:07PM – Session D32 Rough Wall Turbulent Boundary Layers - II Oregon Ballroom 201 - Corey Markfort, University of Iowa

2:57PM D32.00001 Surface fluxes in atmospheric boundary layer flows over complex terrain, WEI ZHANG, Cleveland State University, COREY MARKFORT, University of Iowa, FERNANDO PORT-AGEL, WIRE, EPFL — Interactions between the atmosphere and the land/water surface can be described by fluxes of momentum, heat and other scalars. While predicting the atmospheric boundary-layer (ABL) flows and modeling regional/global weather and climate, these surface fluxes need to be specified as boundary conditions. It is a common practice to use formulations based on the Monin-Obukhov similarity theory even for flows over a wide range of complex terrain, which maybe deviate significantly from the conditions of steady, fully-developed ABL flow, due to the knowledge gap for turbulent transport of fluxes across the interface. This work aims to provide insights for spatial distribution of the surface fluxes in ABL flows involving typical complex terrain cases, including surface roughness transition, steep topography and canopy patches. Results from wind-tunnel experiments will be presented to characterize the surface momentum and heat fluxes for different flow regimes and their correlation to the turbulent flow properties in thermally-stratified boundary layers. Application of the similarity theory to such cases is evaluated by comparing to the measurements. Ultimately, new knowledge of surface fluxes will help to improve parameterization of the surface-atmosphere interaction in numerical models.

3:10PM D32.00002 Measurements of turbulent boundary layer flow and surface fluxes over roughness and temperature transitions, COREY MARKFORT, IIHR-Hydroscience and Engineering, University of Iowa, WEI ZHANG, Mechanical Engineering Department, Cleveland State University, FERNANDO PORTE-AGEL, WIRE Laboratory, EPFL — Often natural and engineered surfaces have spatially heterogeneous properties at a variety of scales that affect the structure of the turbulent boundary layer, which is no longer in equilibrium with the local surface. Predicting the spatial distributions of surface momentum and scalar fluxes over heterogeneous surfaces remains a challenge. We present measurements made in a thermally stratified boundary layer wind tunnel to characterize the turbulent flow and surface fluxes for abrupt transitions in surface temperature and roughness. We compare the development of internal boundary layers for momentum and heat, and associated mean surface flux for two cases. The first is a smooth boundary layer with an abrupt change in surface temperature and the second also involves a change from a fully rough to a smooth wall. The effects of roughness change on surface heat flux and implications for prediction are examined. The data will be compared to typical models that utilize Monin-Obukhov similarity theory.

3:23PM D32.00003 Contrasts Between Momentum and Scalar Exchanges Over Very Rough Surfaces¹, ELIE BOU-ZEID, QI LI, Princeton University — Understanding of the physical processes modulating transport of momentum and scalars over very rough walls is essential in a large range of engineering and environmental applications. Since passive scalars are advected with the flow, broad similarity is expected between momentum and scalar transport. However, unlike momentum, which is dominated by form drag over very rough walls, scalar transport must occur through the viscous exchanges at the solid-fluid interface, which might result in transport dissimilarity. To examine these similarities and differences of momentum and passive scalar exchanges over large three-dimensional roughness elements, a suite of large-eddy simulations is conducted. The turbulent components of the transport of momentum and scalars within the canopy and roughness sublayers are found to be similar. However, strong dissimilarity is noted between the dispersive fluxes. The dispersive components are also found to be a significant fraction of the total fluxes within and below the roughness sublayer. Increasing frontal density induces a general transition in the flow from a rough boundary layer type to a mixed-layer-like type, which is found to have contrasting effects on momentum and scalar transport.

¹This study was funded by the US National Science Foundations Sustainability Research Network Cooperative Agreement number 1444758 and Water Sustainability and Climate program Grant number CBET-1058027.

3:36PM D32.00004 On the turbulent boundary layer over geophysical-like topographies., LEONARDO P. CHAMORRO, ALI M. HAMED, University of Illinois at Urbana-Champaign, LUCIANO CASTILLO, Texas Tech University — The developing and developed flows over 2D and 3D large-scale wavy walls were experimentally studied with high-resolution planar PIV in a refractive-index-matching channel. The 2D wall is described by a sinusoidal wave in the streamwise direction with amplitude to wavelength ratio $a/\lambda x = 0.05$, while the 3D wall has an additional wave in the spanwise direction with $a/\lambda y = 0.1$. The flow was characterized at $Re=4000$ and 40000 , based on the bulk velocity and the channel half height. The walls have amplitude to boundary layer thickness ratio $a/\delta_{99} \approx 0.1$ and resemble large-scale and geophysical-like roughnesses found in rivers and natural terrain. Instantaneous velocity fields and time-averaged turbulence quantities reveal strong coupling between large-scale topography and the turbulence dynamics near the wall, and the presence of a well-structured shear layer that enhances the turbulence for both walls. However, the 3D wall exhibits spanwise flow that is thought to be responsible for distinctive flow features, including comparatively reduced spanwise vorticity and decreased turbulence levels. Further insight is drawn in the developed and developing regions through proper orthogonal decomposition and quadrant analysis.

3:49PM D32.00005 Physical modeling of the atmospheric boundary layer in the UNH Flow Physics Facility, GREGORY TAYLOR-POWER, STEPHANIE GILOOLY, MARTIN WOSNIK, JOE KLEWICKI, JOHN TURNER, University of New Hampshire — The Flow Physics Facility (FPF) at UNH has test section dimensions $W=6.0\text{m}$, $H=2.7\text{m}$, $L=72\text{m}$. 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) using upstream roughness arrays. The American Society for Civil Engineers defines standards for simulating ABLs for different terrain types, from open sea to dense city areas (ASCE 49-12). The standards require the boundary layer to match a power law shape, roughness height, and power spectral density criteria. Each boundary layer type has a corresponding power law exponent and roughness height. The exponent and roughness height both increase with increasing roughness. A suburban boundary layer was chosen for simulation and a roughness element fetch was created. Several fetch lengths were experimented with and the resulting boundary layers were measured and compared to standards in ASCE 49-12: Wind Tunnel Testing for Buildings and Other Structures. Pitot tube and hot wire anemometers were used to measure average and fluctuating flow characteristics. Velocity profiles, turbulence intensity and velocity spectra were found to compare favorably.

4:02PM D32.00006 The flow field around a pair of cubes immersed in the inner part of a turbulent boundary layer measured using holographic microscopy¹, JIAN GAO, JOSEPH KATZ, Johns Hopkins University — The objective of this study is to characterize the interaction of a turbulent boundary layer with roughness elements. Digital holographic microscopy is applied to measure the flow structure and turbulence around a pair of cubic roughness elements aligned in the spanwise direction with height $a = 90\delta_\nu$, and embedded in the inner layer of a turbulent channel flow at $Re_\tau = 2500$. The ratio of half-channel height to cube height is 25, and the cubes are separated by $1.5a$. The field-of-view size is $385\delta_\nu \times 250\delta_\nu \times 190\delta_\nu$, and the vector spacing of the 3D 3-component velocity fields is $5.4\delta_\nu$. Ensemble statistics, which provide distributions of mean velocities and Reynolds stresses, demonstrate the boundary layer separation upstream of the obstacles and resulting formation of vortical semi-rings around each cube, dominated by vertical vorticity on the sides and spanwise vorticity on top of the cubes. The vortical "canopy" persists and hovers around the separated region behind the cubes, but then becomes scrambled in the turbulent wake further downstream. The necklace vortices form upstream of the cubes and remain concentrated near the wall as streamwise vortices between the cubes, but expand substantially in the wake region. Effects of the neighboring cube include accelerated flow channeling in the space between the obstacles and differences between the spatial distributions of vorticity at the inner and outer sides.

¹Funded by NSF and ONR

4:15PM D32.00007 Effect of truncated cone roughness element density on hydrodynamic drag¹, KRISTOFER WOMACK, Johns Hopkins University, MICHAEL SCHULTZ, U.S. Naval Academy, CHARLES MENEVEAU, Johns Hopkins University — An experimental study was conducted on rough-wall, turbulent boundary layer flow. Varying planform densities of truncated cone roughness elements were investigated. Element densities studied ranged from 10% to 57%. Detailed turbulent boundary layer velocity statistics were recorded with a two-component LDV system on a three-axis traverse. Hydrodynamic roughness length (z_0) and skin-friction coefficient (C_f) were determined and compared with the estimates from existing roughness element drag prediction models including Macdonald et al. (1998) and Yang et al. (2015). The roughness elements used in this work model idealized barnacles, so implications of this data set for ship powering are considered.

¹Office of Naval Research

4:28PM D32.00008 Skin friction measurements of mathematically generated roughness in the transitionally- to fully-rough regimes, JULIO BARROS, MICHAEL SCHULTZ, KAREN FLACK, United States Naval Academy — Engineering systems are affected by surface roughness which cause an increase in drag leading to significant performance penalties. One important question is how to predict frictional drag purely based upon surface topography. Although significant progress has been made in recent years, this has proven to be challenging. The present work takes a systematic approach by generating surface roughness in which surfaces parameters, such as rms , skewness, can be controlled. Surfaces were produced using the random Fourier modes method with enforced power-law spectral slopes. The surfaces were manufactured using high resolution 3D-printing. In this study three surfaces with constant amplitude and varying slope, P , were investigated ($P = -0.5, -1.0, -1.5$). Skin-friction measurements were conducted in a high Reynolds number turbulent channel flow facility, covering a wide range of Reynolds numbers, from hydraulic-smooth to fully-rough regimes. Results show that some long wavelength roughness scales do not contribute significantly to the frictional drag, thus highlighting the need for filtering in the calculation of surface statistics. Upon high-pass filtering, it was found that k_{rms} is highly correlated with the measured k_s .

4:41PM D32.00009 ABSTRACT WITHDRAWN —

4:54PM D32.00010 ABSTRACT WITHDRAWN —

Sunday, November 20, 2016 2:57PM - 5:07PM —

Session D33 Turbulent Boundary Layers: Structures - I Oregon Ballroom 202 - Suranga Dharmarathne, Texas Tech University

2:57PM D33.00001 Effects of vortical motions on turbulence scalar transport in a turbulent channel flow, SURANGA DHARMARATHNE, Texas Tech University, MURAT TUTKUN, Institute for Energy Technology, RONALD ADRIAN, Arizona State University, LUCIANO CASTILLO, Texas Tech University — Direct numerical simulations of a turbulent channel flow at Reynolds number, $Re_\tau = 394$, (based on friction velocity and channel half height) were carried out to investigate the relationship between coherent vortices and turbulence scalar transport. Previous observations from three-dimensional two-point correlations of streamwise velocity fluctuations (u) and scalar fluctuations (θ) suggest that three-dimensional iso-surfaces of correlations are distinctively different in all regions of the turbulent channel. Hence, it can be hypothesized that the scalars are more attracted to vortical structures of the flow and the velocity fluctuations reside in low-momentum regions induced by hairpin packets or vortex clusters. In order to test this, we first employ vortex identification methods (λ_2 , λ_{ci} , and Q -criterion) to capture the regions of high vortical activity. Then the three-dimensional correlations between scalar fluctuations and extracted vortices are computed to investigate the validity of the hypothesis. Behavior of the streamwise component of fluctuating flux term ($u\theta$) and the wall-normal component of fluctuating flux term ($v\theta$) are studied to quantify the relation between scalar fluxes and vortices.

3:10PM D33.00002 Hairpin vortices in the outer and near wall regions of the canonical turbulent boundary layer, JAMES WALLACE, University of Maryland, XIAOHUA WU, Royal Military College of Canada, PARVIZ MOIN, Stanford University — While the dominance of hairpin vortices and their significance for transport processes in the transitional and early turbulent regions of the canonical turbulent boundary layer has been widely accepted, opinion is divided about the developed flow downstream. Here we investigate the representative vortical structures in the outer and near wall regions for the momentum thickness Reynolds number, Re_θ , of up to 3000 using the DNS database described in Phys. Fl. 26, 091104. This boundary layer grows spatially from a laminar state at $Re_\theta = 80$ beneath a freestream of continuous and nearly isotropic turbulence decaying from an intensity of 3 to 0.8%. The vortical structures are visualized with the swirling strength, λ_{ci} . In the outer region hairpin vortices appear and are advected over distances corresponding to about 300 - 400 in Re_θ within the fully turbulent region, demonstrating that they are not remnants of transitional structures. The near wall vortical structures are more difficult to visualize and require careful tuning of the swirling strength and making invisible the flow above the near wall region of the flow. The hairpins in this region occur in intermittent clusters that have features remarkably similar to transitional turbulent spots.

3:23PM D33.00003 Spectral structure and linear mechanisms in a ‘rapidly’ distorted boundary layer¹, SOURABH DIVAN², Indian Inst of Science, JONATHAN MORRISON, Imperial College London — A characteristic feature of a turbulent boundary layer (TBL) at high Reynolds numbers is the presence of coherent motions such as the ‘large scale motions’ and ‘superstructures’. In this work we attempt to mimic such coherent motions and their spectral structure using a simplified experimental arrangement of a boundary layer flow over a flat plate subjected to grid-generated turbulence and/or localized patch of surface roughness. The velocity measurements done downstream of a grit roughness patch (in absence of grid turbulence) show that over a certain distance the energy spectrum of streamwise velocity fluctuations shows a bi-modal shape which resembles that found in a high-Re TBL. We also carry out experiments with both grid turbulence and grit roughness present and show that it is possible to ‘synthesize’ the structure of a TBL in the wall-normal direction, in the limited context of streamwise coherent motions, using the present experimental design. These results indicate that the predictions of the Rapid Distortion Theory (RDT) can be applied to the present case in a region close to the plate leading edge, and we examine the linearized effects of ‘blocking’ and ‘shear’ on turbulent fluctuations near the edge of the boundary layer and close to the wall in the framework of the RDT.

¹We acknowledge financial support from EPSRC (grant no. EP/1037938)

²The work was done when the author was a research associate in the Aeronautics Department at Imperial College London

3:36PM D33.00004 Relating instantaneous structures and mean flow characteristics of turbulent boundary layers, CHARITHA DE SILVA, JIMMY PHILIP, NICHOLAS HUTCHINS, IVAN MARUSIC, University of Melbourne — Recent works have highlighted the presence of thin interfacial layers of high shear that demarcate regions of relatively uniform streamwise momentum in turbulent boundary layers. Here, we aim to further our understanding of how such a zonal-like structural arrangement manifests in the averaged flow statistics. To this end, we start by identifying high shear interfaces in turbulent boundary layers employing particle image velocimetry databases that span more than an order of magnitude of friction Reynolds number ($Re_\tau = 10^3\text{--}10^4$). Inspection of these recurrent features reveal that their geometry is highly contorted and exhibits self-similarity across a wide range of scales. The Reynolds number dependence of these features is also investigated, together with their associated scaling. Based on these findings and the persistent presence of sharp changes in momentum in turbulent boundary layers, a simple model is presented towards reconstructing the mean velocity profile.

3:49PM D33.00005 Decomposition of multi-scale coherent structures in a turbulent boundary layer by variational mode decomposition¹, WENKANG WANG, CHONG PAN, JINJUN WANG, Fluid Mechanics Key Laboratory of Ministry of Education, Institute of Fluid Mechanics, Beihang University — Turbulent boundary layer (TBL) is believed to contain a wide spectrum of coherent structures, from near-wall low-speed streaks characterized by inner scale to log-layer large-scale coherent motions (LSM and VLSM) characterized by outer scale. Recent studies have evidenced the interaction between these multi-scale structures via either bottom-up or top-down mechanisms, which implies the possibility of identifying the coexistence of their footprints at medium flow layer. Here, we propose a Quasi-Bivariate Variational Mode Decomposition method (QB-VMD), which is an update of the traditional Empirical Mode Decomposition (EMD) with bandwidth limitation, for the decomposition of the PIV measured 2D flow fields with large ROI ($\Delta x \times \Delta z \sim 4\delta \times 1.5\delta$) at specified wall-normal heights ($y/\delta = 0.05 \sim 0.2$) of a turbulent boundary layer with $Re_\tau = 3460$. The empirical modes identified by QB-VMD well capture the characteristics of log-layer LSMS as well as that of near-wall streak-like structures. The lateral scales of these structures are analyzed and their respective energy contribution are evaluated.

¹Supported by both the National Natural Science Foundation of China (Grant Nos. 11372001 and 11490552) and the Fundamental Research Funds for the Central Universities of China (No. YWF-16-JCTD-A-05).

4:02PM D33.00006 Influence of large-eddy breakup device on near-wall turbulent structures in turbulent boundary layer¹, JOON-SEOK KIM, JINYUL HWANG, MIN YOON, JUNSUN AHN, HYUNG JIN SUNG², KAIST, FLOW CONTROL LAB TEAM — Direct numerical simulation of a large-eddy breakup (LEBU) device in a spatially developing turbulent boundary layer was performed to investigate the influence of outer structures on the near-wall turbulence. The thin and rectangular shaped LEBU device was placed on $y/\delta = 0.8$ and the device reduced the skin-friction coefficient (C_f) up to 17%. Decomposition of C_f showed that the contribution of the Reynolds shear stress decreased along the wall-normal direction. The reduction of the Reynolds shear stress was associated with the decrease of the ejection and sweep events, and in particular the latter was significantly reduced compared to the former in the near-wall region. The spanwise length scale of high-speed structures was more shortened than that of low-speed very near the wall ($y^+ = 20$). As a result, the dispersive motions induced by the outer sweeps were weakened leading to the reduction of C_f even the LEBU device located far from the wall.

¹This work was supported by the Creative Research Initiatives (No. 2016-004749) program of the National Research Foundation of Korea (MSIP).

²Correspondence to Hyung Jin Sung

4:15PM D33.00007 Influence of large-scale low- and high-speed structures on the near-wall vortical motions in turbulent boundary layer¹, JINYUL HWANG, HYUNG JIN SUNG, KAIST — Direct numerical simulation data of turbulent boundary layer ($Re_\tau = 1000$) are used to investigate the large-scale influences on the vortical structures in the near-wall region. The streamwise swirling strength (λ_x) depends on the strength of large-scale streamwise velocity fluctuations (u_l). The amplitude of λ_x decreases under the negative u_l rather than the positive u_l , analogous to the amplitude modulation of the velocity fluctuations. The dependence of λ_x on u_l is due to the opposite spanwise motions in the footprints of low- and high-speed structures, which are congregative and dispersive, respectively. Conditionally averaged fields conditioned on λ_x under the negative- and positive- u_l events show that the swirling motions lie within the congregative and dispersive motions. The dispersive motion is more intense than the congregative motion because the sweep of high-momentum fluid under $u_l > 0$ splats on the wall while the spanwise motions under $u_l < 0$ decrease as the flows come close to each other. Due to the strong dispersive motion, the small-scale spanwise velocity fluctuations (w_s) are strengthened, whereas w_s are relatively weak (attenuated) under $u_l < 0$. As a result, the wall-normal components are also enhanced under $u_l > 0$, which contributes to the amplification of λ_x .

¹This work was supported by the Creative Research Initiatives (No. 2016-004749) program of the National Research Foundation of Korea (MSIP) and partially supported by KISTI under the Strategic Supercomputing Support Program.

4:28PM D33.00008 Spatio-temporal characteristics of large scale motions in a turbulent boundary layer from direct wall shear stress measurement¹, ROMMEL PABON, CASEY BARNARD, LAWRENCE UKEILEY, MARK SHEPLAK, University of Florida — Particle image velocimetry (PIV) and fluctuating wall shear stress experiments were performed on a flat plate turbulent boundary layer (TBL) under zero pressure gradient conditions. The fluctuating wall shear stress was measured using a microelectromechanical 1mm \times 1mm floating element capacitive shear stress sensor (CSSS) developed at the University of Florida. The experiments elucidated the imprint of the organized motions in a TBL on the wall shear stress through its direct measurement. Spatial autocorrelation of the streamwise velocity from the PIV snapshots revealed large scale motions that scale on the order of boundary layer thickness. However, the captured inclination angle was lower than that determined using the classic method by means of wall shear stress and hot-wire anemometry (HWA) temporal cross-correlations and a frozen field hypothesis using a convection velocity. The current study suggests the large size of these motions begins to degrade the applicability of the frozen field hypothesis for the time resolved HWA experiments. The simultaneous PIV and CSSS measurements are also used for spatial reconstruction of the velocity field during conditionally sampled intense wall shear stress events.

¹This material is based upon work supported by the National Science Foundation Graduate Research Fellowship under Grant No. DGE-1315138

4:41PM D33.00009 Streak instability as an initiating mechanism of the large-scale motions in a turbulent channel flow., MATTEO DE GIOVANETTI, Department of Aeronautics, Imperial College London, HYUNG JIN SUNG, Department of Mechanical Engineering, KAIST, YONGYUN HWANG, Department of Aeronautics, Imperial College London — The large-scale motions (or bulges) have often been believed to be formed via merge and/or growth of the near-wall hairpin vortical structures. Here, we report our observation that they can be directly generated by an instability of the amplified streaky motions in the outer region (i.e. very-large-scale motions) through the self-sustaining process. We design a LES-based numerical experiment in turbulent channel flow for $Re_T = 2000$ where a body forcing is implemented to artificially drive an infinitely long streaky motion in the outer layer. As the forcing amplitude is increased, it is found that a new energetic structure emerges at $\lambda_x \approx 3 \sim 4h$ of the streamwise length (h is the half height of channel) particularly in the wall-normal and spanwise velocities. A careful statistical examination reveals that this structure is likely to be linked with the sinuous-mode streak instability of the amplified streak, consistent with previous theoretical studies. Application of dynamic mode decomposition to this instability further shows that the phase speed of this structure scales with the outer velocity and it is initiated around the critical layer of the streaky flow.

4:54PM D33.00010 On validating Quasi-Steady Quasi-Homogeneous nature of the relationship between large-scale and small-scale structures in a turbulent boundary layer, CHI ZHANG, SERGEI CHERNYSHENKO, Imperial College London — A formal definition to the two hypotheses of the quasi-steady and quasi-homogeneous (QSQH) theory was proposed. The theory is supposed to explain the phenomenon of the large-scale structures influencing the small-scale structures in a turbulent boundary layer. Multi-objective optimisations were performed to find the optimal cut-off parameters for the new large-scale filters. The new filters were proved to obtain much more clear large-scale structures than the filter suggested by the previous studies. Calculations and comparisons for a set of statistical flow properties extracted from the databases of the direct numerical simulations of a plane channel flow were conducted. The accuracy of the predictions based on the QSQH theory was observed improving when the Reynolds number increases. Extrapolations of u_{rms} and two-points correlation from medium to high Reynolds number based on the QSQH approach were preformed and about 10% accuracy was reported. The more interesting thing is that the QSQH theory implies a dependence of the mean profile log-law constants on the Reynolds number. The main overall result of the present work is the validations of the two hypotheses of the quasi-steady quasi-homogeneous theory in near-wall turbulent flows.

Sunday, November 20, 2016 2:57PM - 5:07PM –

Session D34 Turbulence: Model Analysis via Statistical and Probabilistic Techniques

Oregon Ballroom 203 - Micheal Wilczek, Max Planck Institute

2:57PM D34.00001 A probability distribution approach to synthetic turbulence time series, MICHAEL SINHUBER, Stanford University, EBERHARD BODENSCHATZ, MICHAEL WILCZEK, Max Planck Institute for Dynamics and Self-Organization — The statistical features of turbulence can be described in terms of multi-point probability density functions (PDFs). The complexity of these statistical objects increases rapidly with the number of points. This raises the question of how much information has to be incorporated into statistical models of turbulence to capture essential features such as inertial-range scaling and intermittency. Using high Reynolds number hot-wire data obtained at the Variable Density Turbulence Tunnel at the Max Planck Institute for Dynamics and Self-Organization, we establish a PDF-based approach on generating synthetic time series that reproduce those features. To do this, we measure three-point conditional PDFs from the experimental data and use an adaption-rejection method to draw random velocities from this distribution to produce synthetic time series. Analyzing these synthetic time series, we find that time series based on even low-dimensional conditional PDFs already capture some essential features of real turbulent flows.

3:10PM D34.00002 Non-Gaussian Extension of the Sparse-Direct Interaction Perturbation¹, DAVID PETTY, CARLOS PANTANO, Univ of Illinois - Urbana — An extension of the Sparse Direct-Interaction Perturbation (SDIP) technique is investigated with the objective to predict the theoretical Obukhov-Corrsin constant consistently with experimental observation. This extension involves relaxing the assumption that, in the case of a turbulent passive scalar field, third-order correlations between Non-Direct-Interaction (NDI) fields are identically zero. This is the leading order term in the traditional SDIP expansion. The theory of invariants and dimensional analysis provide a functional form of the retained third-order correlation, and integrability of its Fourier modes produces constraints on the remaining unknown parameters. To close the approximation, these unknown parameters are determined from direct numerical simulation of actively forced turbulent mixing. The resulting closure is then used to estimate the scalar spectral constant.

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

3:23PM D34.00003 Turbulence Model Discovery with Data-Driven Learning and Optimization, RYAN KING, PETER HAMLINGTON, University of Colorado, Boulder — Data-driven techniques have emerged as a useful tool for model development in applications where first-principles approaches are intractable. In this talk, data-driven multi-task learning techniques are used to discover flow-specific optimal turbulence closure models. We use the recently introduced autonomic closure technique to pose an online supervised learning problem created by test filtering turbulent flows in the self-similar inertial range. The autonomic closure is modified to solve the learning problem for all stress components simultaneously with multi-task learning techniques. The closure is further augmented with a feature extraction step that learns a set of orthogonal modes that are optimal at predicting the turbulent stresses. We demonstrate that these modes can be severely truncated to enable drastic reductions in computational costs without compromising the model accuracy. Furthermore, we discuss the potential universality of the extracted features and implications for reduced order modeling of other turbulent flows.

3:36PM D34.00004 Reducing RANS Model Error Using Random Forest, JIAN-XUN WANG, JIN-LONG WU, HENG XIAO, Virginia Tech , JULIA LING, Sandia National Labs — Reynolds-Averaged Navier-Stokes (RANS) models are still the work-horse tools in the turbulence modeling of industrial flows. However, the model discrepancy due to the inadequacy of modeled Reynolds stresses largely diminishes the reliability of simulation results. In this work we use a physics-informed machine learning approach to improve the RANS modeled Reynolds stresses and propagate them to obtain the mean velocity field. Specifically, the functional forms of Reynolds stress discrepancies with respect to mean flow features are trained based on an offline database of flows with similar characteristics. The random forest model is used to predict Reynolds stress discrepancies in new flows. Then the improved Reynolds stresses are propagated to the velocity field via RANS equations. The effects of expanding the feature space through the use of a complete basis of Galilean tensor invariants are also studied. The flow in a square duct, which is challenging for standard RANS models, is investigated to demonstrate the merit of the proposed approach. The results show that both the Reynolds stresses and the propagated velocity field are improved over the baseline RANS predictions. SAND Number: SAND2016-7437 A

3:49PM D34.00005 RANS turbulence model form uncertainty quantification for wind engineering flows, CATHERINE GORLE, Stanford University, STEPHANIE ZEOLI, LAURENT BRICTEUX, University of Mons — Reynolds-averaged Navier-Stokes simulations with linear eddy-viscosity turbulence models are commonly used for modeling wind engineering flows, but the use of the results for critical design decisions is hindered by the limited capability of the models to correctly predict bluff body flows. A turbulence model form uncertainty quantification (UQ) method to define confidence intervals for the results could remove this limitation, and promising results were obtained in a previous study of the flow in downtown Oklahoma City. The objective of the present study is to further investigate the validity of these results by considering the simplified test case of the flow around a wall-mounted cube. DNS data is used to determine: 1. whether the marker, which identifies regions that deviate from parallel shear flow, is a good indicator for the regions where the turbulence model fails, and 2. which Reynolds stress perturbations, in terms of the tensor magnitude and the eigenvalues and eigenvectors of the normalized anisotropy tensor, can capture the uncertainty in the flow field. A comparison of confidence intervals obtained with the UQ method and the DNS solution indicates that the uncertainty in the velocity field can be captured correctly in a large portion of the flow field.

4:02PM D34.00006 Galerkin POD Model Closure with Triadic Interactions by the Maximum Entropy Method¹, NICOLAS HÉROUARD, School of Engineering and Information Technology, The University of New South Wales, Northcott Drive, Canberra, ACT, 2600., ROBERT K. NIVEN, School of Engineering and Information Technology, The University of New South Wales, Northcott Drive, Canberra, ACT, 2600, Australia., BERND R. NOACK, LIMSI, CNRS, Paris, France., MARKUS W. ABEL, Ambrosys GmbH, Institute for Physics and Astrophysics, Potsdam University, Potsdam, Germany., MICHAEL SCHLEGEL, Institut für Strömungsmechanik und Technische Akustik, Technische Universität Berlin, Berlin, Germany — The maximum entropy method of Jaynes provides a method to infer the expected or most probable state of a system, by maximizing the relative entropy subject to physical constraints such as conservation of mass, energy and power. A maximum entropy closure for reduced-order models of fluid flows based on principal orthogonal decomposition (POD) is developed, to infer the probability density function for the POD modal amplitudes. This closure takes into account energy transfers by triadic interactions between modes, by extension of a theoretical model of these interactions in incompressible flow (Noack et al, JNET, 2008). The framework is applied to several incompressible flow systems including the cylinder wake, both at low and high Reynolds number (oscillatory and turbulent flow conditions), with important implications for the triadic structure and power balance (energy cascade) in the system.

¹ Australian Research Council Discovery Projects grant DP140104402

4:15PM D34.00007 A single-point model from SO(3) decomposition of the axisymmetric mean-flow coupled two-point equations, TIMOTHY CLARK, Department of Mechanical Engineering, University of New Mexico, Albuquerque NM, ROBERT RUBINSTEIN, Computational Aerosciences Branch, NASA Langley Research Center, Hampton VA, SUSAN KURIEN, Theoretical Division, Los Alamos National Laboratory, Los Alamos NM — The fluctuating-pressure-strain correlations present a significant challenge for engineering turbulence models. For incompressible flow, the pressure is an intrinsically two-point quantity (represented as Green's function, integrated over the field), and therefore representing the implied scale-dependence in a one-point model is difficult. The pioneering work of Launder, Reece and Rodi (1975) presented a model that satisfied the tensor symmetries and dimensional consistency with the underlying Green's function solution, and described the assumptions embedded in their one-point model. Among the constraints of such a model is its inability to capture scale-dependent anisotropic flow development. Restricting our attention to the case of axisymmetric mean-field strains, we present a one-point model of the mean-flow couplings, including the pressure-strain terms, starting from a directional (tensorially isotropic) and polarization (tensorially anisotropic and trace-free) representation of the two-point correlation equations, truncated to the lowest order terms. The model results are then compared to simulations performed using arbitrary orders of spherical harmonic functions from which the exact solution may be obtained to desired accuracy.

4:28PM D34.00008 Rapid Bayesian Inference for Fluid Flow Modeling and Control¹, ROBERT K. NIVEN, The University of New South Wales, EURIKA KAISER, University of Washington, BERND R. NOACK, LIMSI, CNRS, Paris, France., LOUIS N. CATTAFESTA III, Florida State University, MARKUS W. ABEL, Potsdam University, LAURENT CORDIER, CNRS - Université de Poitiers — We give a new framework for rapid Bayesian inference for flow modeling and control, based on Bayes' rule $p(\vec{\theta}|\vec{x}) = p(\vec{x}|\vec{\theta})p(\vec{\theta})/p(\vec{x})$, where p is a probability density function, \vec{x} are multivariate data and $\vec{\theta}$ is one model drawn from a continuous model space $\Omega_{\vec{\theta}}$. We thus seek the pdf of the model $\vec{\theta}$, given the data \vec{x} . Traditionally, Bayesian inference requires marginalization of the integral $p(\vec{x}) = \int d\vec{\theta} p(\vec{x}|\vec{\theta})p(\vec{\theta})$, which is highly computationally expensive and may not even be feasible. Instead, we propose initial order reduction of the data, such as by k-means clustering, to generate discretized data c_i on a reduced-order data space C , followed by Bayesian inference to infer the conditional probability $P(\gamma_m|c_k)$ of the discretized model γ_m in a reduced-order model space Γ . If needed, an inversion to infer $p(\vec{\theta}|\gamma_m)$ can be conducted. The method substantially reduces the computational complexity of Bayesian inference, enabling real-time turbulent modeling and control. We report applications to several turbulent flow and dynamical systems.

¹ Australian Research Council Discovery Projects grant DP140104402

4:41PM D34.00009 On the statistics of backscatter from sub-grid fluctuations at high Reynolds numbers¹, MICHELE BUZZICOTTI, Department of Physics & INFN, University of Rome Tor Vergata, Rome, Italy., HUSSEIN ALUIE, Department of Mechanical Engineering, University of Rochester, USA, LUCA BIFERALE, FABIO BONACCORSO, MORITZ LINKMANN, Department of Physics & INFN, University of Rome Tor Vergata, Rome, Italy. — We study the effect of different filtering strategies on the statistical properties of the subgrid-scale energy transfer of high Reynolds numbers homogeneous and isotropic turbulence. We focus on the upscale energy transfer (backscatter) from small to large scales. We discuss the extent to which the backscatter statistics depend on the filtering strategy, using either exact projectors on different subsets of Fourier modes or more traditional convolutions with analytical kernels in physical space. We also assess the backscatter contribution from different helical components of the sub-grid fluctuations.

¹ERC ADG NewTURB 2013

4:54PM D34.00010 ABSTRACT WITHDRAWN —

Sunday, November 20, 2016 2:57PM - 5:07PM —
Session D35 Stratified Turbulence and Convection Oregon Ballroom 204 - Philip Marcus, University of California, Berkeley

2:57PM D35.00001 Localized turbulent spots in a stratified shear flow, JOHN TAYLOR,

DAMTP, University of Cambridge — Despite the large Reynolds numbers involved, turbulence in geophysical flows is often highly intermittent in space and time as the stabilizing effects of density stratification inhibit vertical motions. Direct numerical simulations of stratified turbulence exhibit highly localized ‘bursting’ events. The transient nature of these bursts makes them difficult to study systematically. Here, we use a new control technique to study localized patches of turbulence in stratified shear flows. The Richardson number, controlling the ‘heaviness’ of dense fluid is adjusted in time to maintain a fixed level of turbulent kinetic energy. This process allows us to maintain localized turbulent spots and study their properties.

3:10PM D35.00002 Instabilities, exact coherent structures and layer formation in horizontally shearing body-forced stably-stratified flow¹, DAN LUCAS, DAMTP, University of Cambridge,

C.P. CAULFIELD, BPI & DAMTP, University of Cambridge, RICH KERSWELL, University of Bristol — We consider turbulence driven by a large scale horizontal shear by way of the Kolmogorov flow (sinusoidal body forcing) and a background linear stable stratification imposed in the third direction. This provides a tractable arena to investigate the formation of coherent structures, which in this case organise the flow into horizontal layers by inclining the background shear as the strength of the stratification is increased. The coherent structures can be traced back to new instabilities of the base flow which have a vertical wavelength depending on Richardson number. We investigate how the vertical lengthscales observed in the turbulence are related to the exact solutions and compare to the other well studied examples of instability driving layer formation. We also expose the chaotic motions of the stratified turbulence by locating unstable periodic orbits embedded therein.

¹Supported by EPSRC grant EP/K0345291/1 ‘Mathematical Underpinnings of Stratified Turbulence’

3:23PM D35.00003 Zombie Turbulence and More in Stratified Couette Flow, PHILIP

MARCUS, University of CA Berkeley, JOE BARRANCO, San Francisco State Univ, SUYANG PEI, Texas A & M Corpus Christi, CHUNG-HSIANG JIANG, University of CA Berkeley — Zombie turbulence occurs in rotating, shearing vertically-stratified flows such as stratified Couette flows. The turbulence is triggered by a neutrally-stable eigenmode with a critical layer receptive to finite-amplitude perturbations. Once excited, the critical layer becomes a vortex layer pair that rolls up into discrete vortices. Those vortices excite new critical layers, and the process repeats ad infinitum. When the vortex amplitudes become sufficiently large, the flow becomes turbulent. Although possessing a mid-range energy spectrum with $E(k) \propto k^{-5/3}$, the turbulence is non-Kolmogorov, highly anisotropic, and with large turbulent, but coherent, structures that retain the length scales of the spacing between the critical layers. The motivation for this study is protoplanetary disks (PPDs) where new stars form. In the PPD the Brunt-Vaisala frequency N increases as a function of distance from the midplane where it is zero. We cannot trigger the initial finite amplitude instability where N is small (close to the midplane). However, computations in PPDs and Couette flows show that zombie turbulence forms where N is large, and then a new type of turbulence, that is neither zombie nor Kolmogorov turbulence, fills in the remainder of the domain even where $N = 0$.

3:36PM D35.00004 Conditional Analysis of Dynamically Distinct Regions in Stratified Turbulence¹, GAVIN PORTWOOD, STEPHEN DE BRUYN KOPS, University of Massachusetts, JOHN TAYLOR, University of Cambridge, HASEM SALEHIPOUR, University of Toronto, COLM-CILLE CAULFIELD, University of Cambridge — Stratified flows have been shown to exhibit broadly intermittent flow dynamics at large scales. In DNS of forced homogeneous stratified turbulence, we employ a conditional averaging technique to distinguish compositional flow regions which define the entire flow domain. Here, we condition on the vertical density gradient at inertial and buoyancy length scales to subdivide homogeneous stratified turbulence into three distinct regions that may be characterised by $Gn \equiv \epsilon/\nu N^2$. We show that flows across the Fr-Re parameter space exhibit regions of (a) moderately ‘quiescent’ flow with few three-dimensional overturnings, (b) ‘layered’ turbulent regions which have constrained vertical length scales, and (c) three dimensional ‘patches’ of turbulence and that these regions may be characterised by $Gn \sim O(1)$, $Gn \sim O(10)$, and $Gn \sim O(100)$, respectively. We conjecture that treating stratified turbulence as an instantaneous assemblage of these different regions in varying proportions may explain some of the apparently highly scattered flow dynamics and statistics previously reported in the literature.

¹U.S. Office of Naval Research via grant N00014-15-1-2248; U.K. Engineering and Physical Sciences Research Council grant EP/K034529/1; U.S. DoD HPCMP Frontier Project FP-CFD-FY14-007

3:49PM D35.00005 Formation of temperature front in stably stratified turbulence

, YOSHIFUMI KIMURA, Nagoya University, PETER SULLIVAN, JACKSON HERRING, National Center for Atmospheric Research — An important feature of stably stratified turbulence is the significant influence of internal gravity waves which makes stably stratified turbulence unique compared to homogeneous isotropic turbulence. In this paper, we investigate the genesis of temperature fronts—a crucial subject both practically and fundamentally—in stably stratified turbulence using Direct Numerical Simulations (DNS) of the Navier-Stokes equation under the Boussinesq approximation with 1024^3 grid points. Vertical profiles of temperature fluctuations show almost vertically periodic sawtooth wavy structures with negative and positive layers stacked together with clear boundaries implying a sharp temperature fronts. The sawtooth waves consist of gradual decreasing temperature fluctuations with rapid recovery to a positive value as the frontal boundary is crossed vertically. This asymmetry of gradients comes from the structure that warm temperature region lies on top of cool temperature region, and can be verified in the skewed probability density function (PDF) of vertical temperature gradient. We try to extract the flow structures and mechanism for the formation and maintenance of the strong temperature front numerically.

4:02PM D35.00006 Structure and Mixing of a Turbulent Meandering Plume Part 1: Concentration and Velocity Structure, D.L. YOUNG, Georgia Tech, A.I. LARSSON, University of Gothenburg,

D.R. WEBSTER, Georgia Tech — While much is known about the dynamics and mixing of straight non-buoyant plumes of chemical tracer, comparatively little is understood about the dynamics of meandering plumes, where meander is defined as large scale movement of the plume centerline. Meandering chemical plumes occur in atmospheric and other environmental flows, such as the flow past natural obstacles. In this study, we present simultaneous PIV, PTV, and LIF measurements of a phase-locked meandering plume, the motion of which is forced by the periodic oscillation of a diverting plate. The plume evolves in a turbulent boundary layer in a moderate-Re open channel flow. Similar measurements are made for a straight plume for comparison. Analysis of the LIF data reveals that, for the meandering plume compared to the straight plume, the centerline phase-averaged concentration decreases more rapidly with distance downstream and the plume width increases more rapidly with distance downstream. This indicates a more rapid dilution of tracer. Furthermore, the concentration profiles, along transects perpendicular to the plume centerline, are not symmetric about the meandering plume centerline. Analysis of the velocity data indicates that the large-scale alternating-sign vortices induced by the diverting plate are the dominant feature of the flow. The vortices force the plume to meander and govern the spatial distribution of the phase-averaged concentration, phase-averaged vorticity, Reynolds stress, and TKE.

4:15PM D35.00007 Structure and Mixing of a Turbulent Meandering Plume Part 2:

Turbulent Mixing and Eddy-Diffusivity, D.R. WEBSTER, D.L. YOUNG, Georgia Tech, A.I. LARSSON, University of Gothenburg — Turbulent mixing in a meandering non-buoyant chemical plume is far less understood than in a straight plume – partially due to the difficulty separating the plume meander fluctuations from the turbulent fluctuations. In this study we present high resolution measurements of the covariance of the turbulent fluctuations of velocity and concentration in a phase-locked meandering plume, acquired by combining simultaneous PTV velocity and LIF concentration measurements. The effectiveness of the eddy-diffusivity model for predicting the turbulent flux is assessed. Analysis of the data reveals that the spatial distribution of the turbulent flux is governed by the large-scale alternating-sign vortices that induce the plume meander. Further, regions of high turbulent flux are co-located with areas of large phase-averaged concentration gradients. As a result, the eddy-diffusivity framework models the turbulent flux effectively. As expected from turbulent mixing theory, the eddy-diffusivity coefficient plateaus at a constant value once the plume width reaches the size of the largest eddies (i.e., the scale of the water depth in this open channel flow). However, when the plume width is less than the water depth the eddy-diffusivity coefficient scales with the plume width to the $3/4$ power. This differs from the theoretical $4/3$ scaling that results from the assumption of an inertial subrange. The extent of the inertial subrange is extremely limited in the current moderate- Re open channel flow.

4:28PM D35.00008 A comparison of the turbulent entrainment process in line plumes and wall plumes, DAVID PARKER, HENRY BURRIDGE, JAMIE PARTRIDGE, PAUL LINDEN, Department of Applied Mathematics and Theoretical Physics, University of Cambridge — Flows driven by sources of buoyancy appear in a large number of geophysical and industrial applications. The process of turbulent entrainment in these flows is key to understanding how they evolve and how one might model them. It has been observed that the entrainment is reduced when a line source of buoyancy is positioned immediately adjacent to a wall. To gain insight into the effect of the wall on the entrainment process we perform simultaneous PIV and LIF on both line plumes, in the absence of any boundary, and when the source is adjacent to a vertical boundary forming a wall plume. The experiments are designed to isolate the effect of the wall by using the same experimental setup and parameters for both flows with the addition of the wall and half the buoyancy flux used in the wall plume case. Of particular interest is the effect the large scale eddies, forming at the edge of the plume and engulfing ambient fluid, have on the entrainment process. By using velocity statistics in a coordinate system based on the instantaneous scalar edge of the plume, a technique we have recently used to analyse similar effects in an axisymmetric plume, the significance of this large scale engulfment will be quantified.

4:41PM D35.00009 Statistical parameters of thermally driven turbulent anabatic flow¹, RONI HILEL, DAN LIBERZON, Civil and Environmental Engineering Faculty, The Technion, Israel — Field measurements of thermally driven turbulent anabatic flow over a moderate slope are reported. A collocated hot-films-sonic anemometer (Combo) obtained the finer scales of the flow by implementing a Neural Networks based *in-situ* calibration technique. Eight days of continuous measurements of the wind and temperature fluctuations reviled a diurnal pattern of unstable stratification that forced development of highly turbulent unidirectional up slope flow. Empirical fits of important turbulence statistics were obtained from velocity fluctuations' time series alongside fully resolved spectra of velocity field components and characteristic length scales. TKE and TI showed linear dependence on Re , while velocity derivative skewness and dissipation rates indicated the anisotropic nature of the flow. Empirical fits of normalized velocity fluctuations power density spectra were derived as spectral shapes exhibited high level of similarity. Bursting phenomenon was detected at 15% of the total time. Frequency of occurrence, spectral characteristics and possible generation mechanism are discussed.

¹BSF Grant 2014075

4:54PM D35.00010 Phenomenology of turbulent convection¹, MAHENDRA VERMA, ANANDO CHATTERJEE, ABHISHEK KUMAR, Indian Institute of Technology Kanpur, RAVI SAMTANEY, King Abdullah University of Science and Technology — We simulate Rayleigh-Bénard convection (RBC) in which a fluid is confined between two thermally conducting plates. We report results from direct numerical simulation (DNS) of RBC turbulence on 4096^3 grid, the highest resolution hitherto reported, on 65536 cores of Cray XC40, Shaheen II, at KAUST. The non-dimensional parameters of our simulation are: the Rayleigh number $Ra = 1.1 \times 10^{11}$ (the highest ever for a pseudo-spectral simulation) and Prandtl number of unity. We present energy flux diagnostics of shell-to-shell (in wave number space) transfer. Furthermore, noting that convective flows are anisotropic due to buoyancy, we quantify anisotropy by subdividing each wavenumber shell into rings and quantify ring energy spectrum. An outstanding question in convective turbulence is the wavenumber scaling of the energy spectrum. Our pseudo-spectral simulations of turbulent thermal convection coupled with novel energy transfer diagnostics have provided a definitive answer to this question. We conclude that convective turbulence exhibits behavior similar to fluid turbulence, that is, Kolmogorov's $k^{-5/3}$ spectrum with forward and local energy transfers, along with a nearly isotropic energy distribution.

¹The supercomputer Shaheen at KAUST was utilized for the simulations.

Sunday, November 20, 2016 2:57PM - 5:07PM –

Session D36 Drops: Superhydrophobic Surfaces Portland Ballroom 251 - Pierre Chantelot, Ecole Polytechnique

2:57PM D36.00001 Ring bouncing, PIERRE CHANTELOT, ANAIS GAUTHIER, CHRISTOPHE CLANET, DAVID QUERE, Ecole Polytechnique - PMMH, ESPCI — Point like superhydrophobic macrotextures attached to a flat substrate of same repellency can modify the dynamics of impacting water droplets and lead to shorter bouncing times than on a flat substrate. We investigate the contact time reduction for centered and off-centered impacts on a single texture and show that the effect is robust. We discuss how a macrotextured substrates modifies the impact figure compared to a regular substrate and link it to the reduction of the bouncing time.

3:10PM D36.00002 Superhydrophobic immersion, MARTIN COUX, ADRIEN MATHIS, CHRISTOPHE CLANET, DAVID QUERE, PMMH, ESPCI ; LadhyX, Ecole Polytechnique — A superhydrophobic object is an object on which water doesn't spread. We can think conversely, such an object should be covered by air when immersed in water. The film of air that is formed in this case is visible at the naked eye owing to its brightness. Natural questions that arise from the observation of this phenomenon are how much air stays trapped between the liquid and the solid, *i.e.* what is the thickness of the film, and how this quantity can be modified. In this study, we describe an experimental setup that allows us to easily control the velocity of immersion of an object into a liquid bath and to access the volume of dragged air, from which we can deduce the thickness of the air film.

3:23PM D36.00003 Tuning Superhydrophobic Nanostructures to Enhance Jumping-Droplet Condensation, MEGAN MULROE, Virginia Tech, BERNADETA SRIJANTO, PATRICK COLLIER, Oak Ridge National Laboratory, JONATHAN BOREYKO, Virginia Tech — It was recently discovered that condensation growing on a nanostructured superhydrophobic surface can spontaneously jump off the surface when two or more droplets coalesce together. The minimum droplet size for jumping to occur is of order 10 microns, but it is unclear whether this is the true lower limit of jumping droplets or simply a limitation of current superhydrophobic surfaces. Here, we analyze the dynamics of jumping droplets on six different superhydrophobic surfaces where the topography of the nanopillars was systematically varied. The critical diameter for jumping to occur was observed to be highly dependent upon the height and diameter of the nanopillars; surfaces with very tall and slender nanopillars enabled jumping droplets at a smaller critical size of order 1 micron. An energetic model of the incipient growth of condensate shows that the nanostructure topology affects the rate of increase of a growing droplet's apparent contact angle, with jumping being enabled at very large angles. These findings indicate that the true upper limit to the performance of jumping-droplet condensers has not yet been reached and can be further improved using advanced nanofabrication techniques.

3:36PM D36.00004 More Puddle Jumping, BABAK ATTARI, Rheem Manufacturing, MARK WEISLOGEL, ANDREW WOLLMAN, YONGKANG CHEN, Portland State University, TREVOR SNYDER, 3D Systems — Large droplets and puddles jump spontaneously from sufficiently hydrophobic surfaces during routine drop tower tests. The simple low-cost passive mechanism can in turn be used as an experimental device to investigate dynamic droplet phenomena for drops up to 10,000 times larger than their normal terrestrial counterparts. We provide or confirm quick and qualitative design guides for such 'drop shooters' as employed in drop tower tests including relationships to predict droplet ejection durations and velocities as functions of drop volume, surface texture, surface contour, wettability pattern, drop volume, and fluid properties including contact angle. The latter are determined via profile image comparisons with numerical equilibrium interface computations. Water drop volumes of 0.04 to 400 mL at ejection speeds of -0.007 to 0.12 m/s are demonstrated. An example application of the puddle jump method is made to the classic problem of regime mapping for low-gravity phase change heat transfer for large impinging drops. Many other candidate problems might be identified.

3:49PM D36.00005 Viscous Puddle Jump, TAIF AL JUBAREE, MARK WEISLOGEL, Portland State Univ, TAN HUA, Washington state university — The phenomena of spontaneous droplet jump from hydrophobic surfaces during low-g drop tower tests was recently reviewed (Wollman et al., Experiments in Fluids, 2016). Such drops may be over 10,000 times larger than typical terrestrial drops and are more akin to puddles than drops. In this work we investigate the effect of viscosity on the puddle jump process for drop/puddle volumes up to 100 mL and dynamic viscosities up to 950 cSt. The large low-cost hydrophobic surfaces are created using PTFE-coated 320 grit sand paper. We adopt a scaling approach to evaluate the relevant terms of the momentum equation before performing an energy balance for both driving and dissipation terms. A scaling law is corroborated by the experimental data for viscous puddle jump time and puddle recoil velocity. Numerical solutions are also conducted for comparisons. We demonstrate highly damped puddle jumps which may be exploited in turn to study further drop dynamics phenomena such as vanishingly small Weber number drop-wall impacts, over-damped oblique impacts and rebounds, and viscous wall-bound droplet boiling in low-gravity environments.

4:02PM D36.00006 Capillary migration of large confined super-hydrophobic drops in wedges¹, LOGAN TORRES, MARK WEISLOGEL, SAM ARNOLD, Portland State University — When confined within an interior corner, drops and bubbles migrate to regions of minimum energy by the combined effects of surface tension, surface wetting, and corner geometry. Such capillary phenomena are exploited for passive phase separation operations in micro-fluidic devices on earth and macro-fluidic devices aboard spacecraft. Our study focuses on the migration of large inertial-capillary drops confined between two planar super-hydrophobic surfaces. In our experiments, the near weightless environment of a drop tower produces $Bo \ll 1$ for drop volumes $O(10\text{mL})$ with migration velocities up to 10 cm/s. We observe transient power law behavior as a function of drop volume, wedge angle, initial confinement, and fluid properties including contact angle. We then further demonstrate how the experiment method may be employed as a large horizontal quiescent droplet generator for studies ranging from inertial non-wetting moving contact line investigations to large geyser-free horizontal drop impacts.

¹NASA Cooperative Agreement NNX12AO47A, URMP

4:15PM D36.00007 Robust superhydrophobic PDMS/camphor based composite coatings with self-cleaning and self-healing properties, SUSHANTA MITRA, BICHITRA SAHOO, SONIL NANDA, JANUSZ KOZINSKI, York University — We report a novel process for the preparation of self-cleaning polymer composite with self-healing ability to self-repair from chemical and mechanical damages using readily available materials like Polydimethylsiloxane (PDMS) and camphor soot particles. When the camphor soot particles loading attained a critical level, the composite coating on glass and stainless steel surfaces reveals self-cleaning property with water contact angle of 171°. We also demonstrate that any degradation of its surface energy under the oxygen plasma etching can be recuperated, illustrating that the obtained superhydrophobic surface has a good self-healing ability. The fabricated PDMS/Camphor soot hybrid coating exhibited excellent retention of superhydrophobicity against impact of sand particles from a height of 10-70 cm. In addition, after being damaged chemically by strong acid treatment (2M HNO₃ solution), the coating can also restore its properties after a short thermal cycle. Such versatile superhydrophobic surfaces can have wide applications ranging from under-water marine vessels to coating for surfaces to protect them from moisture and unwanted penetration of water.

4:28PM D36.00008 Self-Cleaning Properties on Superhydrophobic Surfaces via Condensation¹, DAVID MILLER, JULIE CROCKETT, DANIEL MAYNES, Brigham Young University — Superhydrophobic (SH) surfaces have many unique capabilities, one of which is self-cleaning. When a water droplet rolls on a contaminated SH surface, particulates can adhere to the droplet and roll away with the droplet, creating a self-cleaning effect. Another unique characteristic of SH surfaces is the promotion of dropletwise condensation when cooled in a humid environment. These droplets may engulf particulates on the surface as they are generated and coalesce. This research seeks to understand the potential cleaning efficiency SH surfaces have when water vapor is condensed on a dirty SH surface and allowed to roll off. Multiple condensation cycles with common particulates deposited on SH surfaces oriented vertically are explored. Sliding and contact angles are measured to approximate the cleaning efficiency of the condensed, rolling droplets after each condensation cycle. Results are compared with the cleaning efficiency of water droplets placed on the surface to roll.

¹Moxtek, INC

4:41PM D36.00009 Any material becomes superhydrophobic, if you can make it rough enough at multiple scales.¹, DANIEL ATTINGER, CHRISTOPHE FRANKIEWICZ, Iowa State University — Superhydrophobic surfaces with the self-cleaning behavior of lotus leaves are sought for drag reduction and phase change heat transfer applications. These superrepellent surfaces have traditionally been fabricated by random or deterministic texturing of a hydrophobic material, either as the base material or as a coating on technically relevant base materials. Recently, superrepellent surfaces have also been made from hydrophilic materials, by deterministic texturing using photolithography, without low-surface energy coating. Here, we show that hydrophilic materials can also be made superrepellent [1] to water by chemical texturing, a stochastic rather than deterministic process. These metallic surfaces [2] are the first analog of lotus leaves, in terms of wettability, texture and repellency. A mechanistic model is also proposed to describe the influence of multiple scales of roughness on wettability and repellency. These superrepellent surfaces made of hydrophilic materials are also able to switch between a metastable Cassie-Baxter state and a hydrophilic wetting state. Related opportunities for controlling phase change heat transfer will be discussed. [1] S. Herminghaus, "Roughness-induced non-wetting," *EPL (Europhysics Letters)*, vol. 52, p. 165, 2000; [2] C. Frankiewicz and D. Attinger, "Texture and wettability of metallic lotus leaves," *Nanoscale*, DOI: 10.1039/c5nr04098a.

¹Financial support from the US National Science Foundation, CEBET grant 1235867, is gratefully acknowledged.

4:54PM D36.00010 Droplets on porous hydrophobic surfaces perfused with gas: An air-table for droplets, NIKOLAOS VOURDAS, VASSILIS STATHOPOULOS, Technological Educational Institute of Sterea Ellada, LABORATORY OF CHEMISTRY AND MATERIALS TECHNOLOGY TEAM — Wetting phenomena on porous hydrophobic surfaces are strongly related to the volume and the pressure of gas pockets resided at the solid-liquid interface. When the porous medium is perfused with gas by means of backpressure an inherently sessile pinned droplet undergoes various changes in its shape, contact angles and mobility. This provides an alternative method for active and controlled droplet actuation, without use of electricity, magnetism, foreign particles etc. Superhydrophobicity is not a prerequisite, electrode fabrication is not needed, the liquid is not affected thermally or chemically etc. In this work we explore this method, study the pertinent underlying mechanisms, and propose some applications. The adequate backpressure for droplet actuation has been measured for various hydrophobic porous surfaces. Backpressure for actuation may be as low as some tens of mbar for some cases, thus providing a rather low-energy demanding alternative. The droplet actuation mechanism has been followed numerically; it entails depinning of the receding contact line and movement, by means of a forward wave propagation reaching on the front of the droplet. Applications in valving water plugs inside open- or closed- channel fluidics will be provided.

Sunday, November 20, 2016 2:57PM - 5:07PM –
Session D37 Drops: Capillary Fluidics Portland Ballroom 252 - Serafim Kalliadasis, Imperial College - London

2:57PM D37.00001 Self-organization of levitating droplets over a dry heated substrate¹, VLADIMIR AJAEV, Southern Methodist University, DMITRY ZAITSEV, DMITRY KIRICHENKO, OLEG KABOV, Institute of Thermophysics, Russia — Levitating droplets of liquid condensate are known to organize themselves into ordered structures over hot liquid-gas interfaces. We report experimental observation of similar behavior over a dry heated surface. Even though the life-time of the structure is shorter in this case, its geometric characteristics are remarkably similar to the case of droplets levitating over liquid-gas interfaces. A simple model is developed that predicts the mechanisms of both droplet levitation and inter-droplet interaction leading to pattern formation over dry surface; the model is shown to be in excellent agreement with the experimental data. Using the insights from the new experiments, we are able to resolve some long-standing controversies pertaining to the mechanism of levitation of droplets over liquid-gas interfaces.

¹We gratefully acknowledge the support from the Ministry of Education and Science of Russia (Agreement No 14.613.21.0011, project identifier RFMEFI61314X0011).

3:10PM D37.00002 Analysis of pumpless liquid transport on a wettability-patterned track, ARITRA GHOSH, KEN BRENNER, SOUVICK CHATTERJEE, PALLAB SINHA MAHAPATRA, University of Illinois at Chicago, RANJAN GANGULY, Jadavpur University, CONSTANTINE MEGARIDIS, University of Illinois at Chicago — Pumpless liquid transport can be achieved by tuning curvature of liquid volumes (Laplace pressure) on a diverging superhydrophilic track surrounded by a superhydrophobic background. The liquid, which starts in the form of a deposited droplet, propagates on the track as a well-defined bulge (bulk liquid) followed by a trailing liquid film conforming to the track geometry. In this work, we present a semi-analytical model to explain the trends of observed phenomena as well as the liquid transport dynamics (velocity, acceleration, flow rate) with respect to track geometry, solid wettability contrast, and feed volume. High speed image analysis of the motion of the bulk liquid is performed using a droplet shape tracking algorithm; dominant forces are identified and model predictions are compared with the experimental data. The combination of experimental and analytical tools offers new insight on a problem that is relevant to open-surface microfluidic devices, especially in the point of care (i.e. low cost) technological domain.

3:23PM D37.00003 Liquid spreading along a nanostructured superhydrophilic microlane, SEUNGHO KIM, HO-YOUNG KIM, Seoul National University — Deposition of functional liquids on solid surfaces is an important step in electronic circuit printing and fabrication of some biochips. Here we show that a liquid drop that gently touches a nanostructured superhydrophilic microlane surrounded by hydrophobic background spreads along the pre-defined pattern, allowing for a facile venue to liquid patterning. We find that different regimes of spreading dynamics occur depending on the lane width and the driving force at the liquid source. For a hydrophilic lane narrower than a critical width, the hemiwicking flow driven by capillarity but resisted by viscosity follows the Washburn law. For relatively wider lanes, on the other hand, the spreading rate is a sensitive function of the hydrostatic pressure at the liquid source, so that different power laws for spreading distance with time are observed. We rationalize the observed power laws with scaling analysis considering the effects of liquid bulk invading the hydrophilic lane.

3:36PM D37.00004 Fabrication of Janus hydrogels with stiffness gradient using drop coalescence¹, DONGHEE LEE, KALE GOLDEN, SANGJIN RYU, University of Nebraska-Lincoln — The stiffness of the extracellular matrix (ECM) regulates cellular behaviors, and polyacrylamide (PAAM) gels with stiffness gradient have been used to simulate inhomogeneous ECM and to study the effects of the ECM stiffness on cells. Such hydrogel substrates with stiffness gradient can be fabricated with relatively complicated methods using microfluidics and moving masks. In our study, we develop a simpler method for fabricating Janus hydrogel which has a gradient of stiffness. Two prepolymer solutions were prepared for soft and stiff gel compositions, respectively, and one drop of each solution was placed on a hydrophobic patterned glass. Then, these two drops were gently squeezed by another glass being slowly lowered until coalescence, and gel polymerization was initiated after a certain time period for mixing. The motion of the drops was guided by the hydrophobic pattern. AFM nano-indentation showed that the fabricated Janus PAAM gels have a stiffness gradient which could be controlled by increasing mixing time.

¹This study was supported by Bioengineering for Human Health grant from UNL and UNMC.

3:49PM D37.00005 Optimum responses of droplets under electro-wetting actuation, TUAN TRAN, QUOC VO, Nanyang Technological University, Singapore — The electro-wetting phenomenon has been used extensively to manipulate shape and position of liquid droplets in various applications such as microfluidics, microswitches, liquid lenses, light valves, and fast response displays. One of the quantities critically affecting the performance of such applications is the actuation time, defined as the duration for a droplet to reach a new equilibrium state after an electrical field is applied. We experimentally study the dynamical response of electro-actuated droplets for a wide range of control parameters including viscosity, drop size, and electric field. We show that there exists a relation between such parameters to achieve optimum actuation time, which can be validated by experimental data.

4:02PM D37.00006 Surface nanodroplets for highly efficient liquid-liquid microextraction, MIAOSI LI, ZIYANG LU, HAITAO YU, XUEHUA ZHANG, Royal Melbourne Institute of Technology — Nanoscale droplets on a substrate are an essential element for a wide range of applications, such as laboratory-on-chip devices, simple and highly efficient miniaturized reactors for concentrating products, high-throughput single-bacteria or single-biomolecular analysis, encapsulation, and high-resolution imaging techniques. The solvent exchange process is a simple bottom-up approach for producing droplets at solid-liquid interfaces that are only several tens to hundreds of nanometers in height, or a few femtoliters in volume. Oil nanodroplets can be produced on a substrate by solvent exchange in which a good solvent of oil is displaced by a poor solvent. Our previous work has significantly advanced understanding of the principle of solvent exchange, and the droplet size can be well-controlled by several parameters, including flow rates, flow geometry, gravitational effect and composition of solutions. In this work, we studied the microextraction effect of surface nanodroplets. Oil nanodroplets have been demonstrated to provide highly-efficient liquid-liquid microextraction of hydrophobic solute in a highly diluted solution. This effect proved the feasibility of nanodroplets as a platform for preconcentrating compounds for in situ highly sensitive microanalysis without further separation. Also the long lifetime and temporal stability of surface nanodroplets allow for some long-term extraction process and extraction without addition of stabilisers.

4:15PM D37.00007 Visualization of the Cassie-Wenzel transition with X-ray microscopy¹, SU JIN LIM, YESEUL KIM, SKKU Advanced Institute of Nanotechnology (SAINT), Sungkyunkwan University, SUYEON JEONG, CHANGHYUN PANG, School of Chemical Engineering, Sungkyunkwan University, BYUNG MOOK WEON², School of Advanced Materials Science and Engineering, SKKU Advanced Institute of Nanotechnology (SAINT), Sungkyunkwan University — Water droplets on hydrophobic surfaces with micropillar usually exhibit two wetting states: (i) the Cassie state when air is trapped between water and micropillars and (ii) the Wenzel state when air is completely replaced by water. A transition from the Cassie to the Wenzel states is essential in designing stable hydrophobic surfaces. Directly visualizing the Cassie-Wenzel (C-W) transition is difficult with conventional microscopies because of no transparency from micropillars. Here we suggest a powerful technique based on high-resolution high-penetration X-ray microscopy for clearly visualizing the C-W transition. Thanks to the X-ray penetrating into the opaque micropillars, we were able to directly explore the intermediate state during the C-W transition. We study on the transition dynamics regarding how air replacement by water was gradually propagated with position and time. We believe that the replacement dynamics would be explained as a kind of phase transition kinetics.

¹This research was supported by Global Ph.D Fellowship Program and Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Education (NRF-2015H1A2A1034133) (NRF-2016R1D1A1B01007133).

²Email: bmweon@skku.edu

4:28PM D37.00008 Direct measurements of the pressure distribution along the contact area during droplet impact, THANH-VINH NGUYEN, The University of Tokyo, KIYOSHI MATSUMOTO, Toyo University, ISAO SHIMOYAMA, The University of Tokyo — We report direct measurements of the pressure distribution on the contact area during the impact of a droplet on a micropillar array. The measurements were realized using an array of MEMS-based force sensors fabricated underneath the micropillars. We show that immediately after the droplet hits the surface, the pressure becomes maximum at the center of the contact area and this maximum pressure value is more than 10 times larger than the dynamic pressure. This result emphasizes the effect of water-hammer-type pressure during the early stage of the impact. Furthermore, our measurement results demonstrate that the critical pressure associated with Cassie-Wenzel transition agrees well with the maximum capillary pressure of the micropillar array.

4:41PM D37.00009 Compound Droplet Levitation for Lab-on-a-Chip¹, JAMES BLACK, G. PAUL NEITZEL, Georgia Institute of Technology — A fluid transport mechanism utilizing thermocapillarity has been previously shown to successfully levitate and translate both microliter- and nanoliter-volume droplets of silicone oil. The surface flow required to drive levitation and transport has not been achieved for aqueous droplets, and encapsulation of samples within a layer of silicone oil is necessary. A droplet-on-demand generator capable of producing nanoliter-volume compound droplets has been developed and previously reported. The work presented here discusses efforts to demonstrate the applicability of this microfluidic transport mechanism to lab-on-a-chip systems. We elaborate on translation speeds of single-phase, nanoliter-volume, silicone-oil droplets. Compound droplets of varying compositions of oil and water are then generated, captured, levitated, and merged to explore the composition limits thereof.

¹Work supported by NSF and NASA

4:54PM D37.00010 Pore-scale modeling of moving contact line problems in immiscible two-phase flow, ALEC KUCALA, DAVID NOBLE, MARIO MARTINEZ, Sandia National Laboratories — Accurate modeling of moving contact line (MCL) problems is imperative in predicting capillary pressure vs. saturation curves, permeability, and preferential flow paths for a variety of applications, including geological carbon storage (GCS) and enhanced oil recovery (EOR). Here, we present a model for the moving contact line using pore-scale computational fluid dynamics (CFD) which solves the full, time-dependent Navier-Stokes equations using the Galerkin finite-element method. The MCL is modeled as a surface traction force proportional to the surface tension, dependent on the static properties of the immiscible fluid/solid system. We present a variety of verification test cases for simple two- and three-dimensional geometries to validate the current model, including threshold pressure predictions in flows through pore-throats for a variety of wetting angles. Simulations involving more complex geometries are also presented to be used in future simulations for GCS and EOR problems. Sandia is a multiprogram laboratory operated by Sandia Corporation, a Lockheed Martin Company, for the United States Department of Energy's National Nuclear Security Administration under contract DE-AC04-94AL85000.

Sunday, November 20, 2016 2:57PM - 5:07PM –

Session D38 Flow Instability: Interfacial and Thin Film-Fingering Portland Ballroom 255 -

Matthew Hennessy, Imperial College London

2:57PM D38.00001 A minimal model of solvent evaporation and absorption in thin films

GIULIA L. FERRETTI, MATTHEW G. HENNESSY, JOAO T. CABRAL, OMAR K. MATAR, Imperial College London — We present a minimal model of solvent evaporation and absorption in thin multicomponent films that consist of a volatile solvent and one or many non-volatile solutes. A detailed asymptotic analysis is carried out in order to (i) elucidate the key regimes that occur during evaporation and absorption and (ii) compute solutions that facilitate the extraction of physically significant model parameters from experimental data. A state diagram of the drying process is constructed and used to predict the experimental conditions that lead to the formation of a solute-rich skin below the evaporating surface. In the case of solvent absorption, the model predicts the existence of a diffuse saturation front that propagates from the film surface towards the substrate. The theoretical results are found to be in excellent agreement with data produced from dynamic vapour sorption experiments of ternary mixtures composed of an aluminum salt, glycerol, and water. Finally, we show how the model can be used to predict the drying and absorption dynamics over a wide range of experimental conditions.

3:10PM D38.00002 The effect of a blurred interface on the viscous fingering instability

THOMAS E. VIDEBAEK, Department of Physics, University of Chicago, THIBAULT GUILLET, Department of Mechanics, École Polytechnique, IRMGARD BISCHOFBERGER, Department of Mechanical Engineering, Massachusetts Institute of Technology, SIDNEY R. NAGEL, Department of Physics, University of Chicago — The viscous fingering instability in a quasi-two dimensional Hele-Shaw cell provides a simple tool for studying a complex structure formation. Concentrating on the instability between pairs of miscible fluids where the interfacial tension is nearly zero, we smooth out the discontinuity in the gap-averaged viscosity at the boundary between the fluids; in this system we discover two new features. (i) We find a delay in the instability onset when we change the curvature of the finger tip by applying an oscillatory translational shear of the top plate. (ii) We observe a sharp transition in the structure of the fingers when we decrease the injection rate of the inner fluid so that diffusion smooths the interface. At this transition, there is a jump in both the wavelength and the onset radius of fingering as well as a change in the three-dimensional structure as the fingers go from half filling to fully filling the cell. These experiments indicate that, by controlling the viscosity contrast at the interface, one can alter and perhaps even completely suppress the instability.

3:23PM D38.00003 Instabilities of an immiscible reactive micellar interface in a Hele-Shaw cell, ZAHRA NIROOBAKHSH, Dept of Materials Science and Engineering, Pennsylvania State University, MATTHEW LITMAN, Dept of Mathematics, Pennsylvania State University, ANDREW BELMONTE, Dept of Mathematics/Materials Science and Engineering, Pennsylvania State University — We present the case of a micellar reaction involving two immiscible fluids, which results in the growth of a thin viscoelastic layer between them. A Hele-Shaw cell is initially filled with different oils, including oleic acid, which acts as a cosurfactant. The oil is displaced by an aqueous solution of the surfactant cetylpyridinium chloride. A rich variety of viscous fingering patterns are observed, which are different from classic Saffman-Taylor patterns. We discuss how they change with concentration, surfactant injection rate and type of oil. We also measure the viscoelastic properties of this material using an interfacial rheometer.

3:36PM D38.00004 Control of viscous fingering by chemical reactions

CHINAR RANA, ANNE DE WIT, Univ Libre De Brussels — Viscous fingering is a hydrodynamic instability that occurs in porous media when a less viscous fluid displaces a more viscous one. The interface between both fluids then deforms into fingers, which leads to enhanced mixing. We investigate theoretically the possibility to control this instability thanks to chemical reactions changing the viscosity *in situ*. To do so, we analyze numerically the influence of different chemical kinetics on viscous fingering, looking in particular for conditions stabilizing the instability. We show that the reaction-diffusion specificities of the chemical front can be used to tune the viscous fingering pattern. The properties of this reactive fingering are obtained by computing onset times, mixing lengths and characteristics of the nonlinear fingering dynamics as a function of the parameters of the problem which are the relative viscosity ratios of reactants and products, the Damköhler number quantifying the ratio of hydrodynamic and chemical times scales as well as the Péclet number.

3:49PM D38.00005 Influence of chemical reaction decreasing interfacial tension on immiscible viscous fingering

REIKO TSUZUKI, MASANARI FUJIMURA, NAGATSU YUICHIRO, Tokyo University of Agriculture and Technology — We have experimentally investigated the effects of chemical reaction on immiscible viscous fingering (VF). In the present study, we use a chemical reaction producing a surfactant leading to a decrease in interfacial tension. In our experiment, a more viscous paraffin oil containing linoleic acid is displaced by a less viscous NaOHaq in a radial Hele-Shaw cell. We have found the influence of the reaction on the VF pattern depends on the displacement flow rate. At low flow rate, the reaction makes the fingers narrower. On the other hand, at intermediate flow rate, the reaction makes the fingers wider. At high flow rate, there is little influence of the reaction. These results can be interpreted as follows; when the reaction rate is much faster than the flow rate, interfacial tension is decreased uniformly over the interface. As a result, more finger-splitting occur and the fingers become narrower. When the reaction rate and flow rate are competing, the interfacial tension gradient is formed along the interface. As a result, Marangoni convection is produced, which leads to wider fingers. When the flow rate is much faster than the reaction rate, little reaction occurs during the formation of VF. As a result, the reaction does not influence on VF pattern.

4:02PM D38.00006 Inverse Saffman-Taylor instability in Hele-Shaw experiments using micro-particles¹, FARZAM ZOUESHTIAGH, University of Lille, IEMN-CNRS 8520, ILYESSE BIHI, University of Lille, IEMN-CNRS 8520, University of Florida, JASON BUTLER, University of Florida, CHRISTINE FAILLE, University of Lille, INRA-PIHM, MICHAL BAUDOIN, University of Lille, IEMN-CNRS 8520 — Saffman-Taylor instability can occur when a low viscosity fluid displaces one of higher viscosity. It results from the decrease of the flow resistance as the fluid of lower viscosity replaces the more viscous one. This Saffman-Taylor instability is revisited experimentally for the inverse case of a viscous fluid displacing air when partially wetting particles are lying on the walls. Though the inverse case is otherwise stable, the presence of the particles results in a fingering instability at low capillary number. This capillary-driven instability is driven by the integration of particles into the interface which results from the minimization of the interfacial energy [1]. Bihi et al., Phys. Rev. Lett., 117: 034501, 2016

¹We acknowledge the support from the Marie Curie International Research Staff Exchange Scheme Fellowship (Patterns and Surfaces No. 269207) within the 7th European Community Framework Programme.

4:15PM D38.00007 Experimental study on viscous fingering with partial miscible fluids, RYUTA SUZUKI, YUICHIRO NAGATSU, Department of Chemical Engineering, Tokyo University of Agriculture and Technology, MANORANJAN MISHRA, Department of Mathematics Indian Institute of Technology Ropar, TAKAHIKO BAN, Graduate School of Engineering Science, Osaka University — Viscous fingering (VF) instability occurs when a more viscous fluid is displaced by a less viscous one in porous media or Hele-Shaw cells. So far, studies of VF have focused on fluids that are either fully miscible or immiscible. However, little attention has been paid to VF in partially miscible fluids. Here, we have experimentally investigated VF in a radial Hele-Shaw cell using an aqueous two phase system (Ban et al. Soft Matter, 2012) which is an example of partially miscible fluids system. We have found novel instabilities that are counter-intuitive in miscible and immiscible systems. These include multiple droplets formation for low flow rate and widening of fingers at intermediate flow rate. The occurrence of the new instability patterns is induced by Korteweg effect in which convection is induced during phase separation in partially miscible systems.

4:28PM D38.00008 Spreading dynamics of superposed drops of two liquids on a spinning disk, SUBHADARSHINEE SAHOO, ASHISH ORPE, Chemical Engineering Division, CSIR-National Chemical Laboratory, Pune, India 411008, PANKAJ DOSHI, Pfizer, Inc., Groton, Connecticut, USA — We have experimentally studied simultaneous spreading of two liquid drops, one engulfing the other, when rotated atop a horizontal spinning disk using flow visualization technique. A drop of high surface tension liquid is placed centrally on a horizontal disk followed by a drop of second liquid placed exactly above the first. The second liquid, of higher volume, lower surface tension and lower density than the first, engulfs the first drop completely. The disk is then rotated at a desired speed for a range of volume ratios of two liquids. The spreading behavior of both the drops is captured using a high speed camera. Such an arrangement of two liquids drops does not affect the spreading behavior of the outer liquid, but influences that of the inner liquid significantly. The drop spreads to a larger extent and breaks into more fingers as compared to the case where the same liquid is spreading in the absence of outer liquid. The fingering instability is achieved at earlier times with decreasing volume ratios, indicative of the increasing influence of the edge of outer liquid film being closer to that of the inner liquid. Further, sustained rotation of the disk leads to emanation of drops from the spreading drop travelling outwards through the fingers of outer liquid.

4:41PM D38.00009 Suppression of viscous fingering instability by a chemical reaction producing gel, YUICHIRO NAGATSU, TOSHIRO KANBARA, MASAFUMI TANIGUCHI, Department of Chemical Engineering, Tokyo University of Agriculture and Technology — Viscous fingering (VF) is a well-known hydrodynamic instability which is observed when a more viscous fluid is displaced by another less viscous one in porous media or Hele-Shaw cell. In such a situation, the interface between two fluids formed finger-like patterns. Recently, several techniques for suppress VF instability has been developed, which include time-dependent injection (Dias et al. PRL 2012), addition of permeability gradient (Al-Housseiny Nat. Phys. 2012), and use of viscoelastic plates of Hele-Shaw cell (Pihler-Puzović PRL 2012). Here, we demonstrate our trial to suppress VF by chemical reaction producing gel. We have succeeded to find a system of solutions and reaction in which the reaction producing a gel is able to completely suppress VF. In addition, we have performed rheological measurement of the gel produced at the reactive interface. The VF experiment and the rheological measurement have been performed by varying concentrations of the reactants. We show the storage modulus (G') of the gel, which corresponds to elastic response to small amplitude oscillatory shear, is responsible for the ability of suppression of VF in the present reactive system.

4:54PM D38.00010 Stabilization of miscible viscous fingering by chemical reaction decreasing viscosity, SHUNTARO ARAI, YUICHIRO NAGATSU, Department of Chemical Engineering, Tokyo University of Agriculture and Technology, PRIYANKA SHUKLA, Department of Mathematics, Indian Institute of Technology Madras, ANNE DE WIT, Université libre de Bruxelles (ULB), Nonlinear Physical Chemistry Unit, 1050 Brussels, Belgium — Viscous fingering (VF) occurs when a more viscous fluid is displaced by a less viscous one in porous media or Hele-Shaw cells. In this study, experiment on miscible VF with chemical reaction is conducted by using a Hele-Shaw cell. Here, the chemical reaction takes place between a polymer dissolved in the more viscous solution and hydrochloric acid (HCl) dissolved in the less viscous one in the miscible interface region. The reaction decreases the viscosity of the polymer solution. The experiment shows that the reaction stabilizes VF when the flow rate is small. In the present study, the corresponding numerical simulation is also conducted. The simulation is able to reproduce the experimental results mentioned above when different diffusion coefficients are considered meaning that HCl diffuses faster than the polymer. However, the stabilization cannot be found under conditions of the same diffusivity of the reactants. These numerical results show that the different diffusivity is responsible for the stabilization of miscible VF by the chemical reaction decreasing viscosity.

Sunday, November 20, 2016 2:57PM - 5:07PM –
Session D39 Bio: Plant Mechanics Portland Ballroom 256 - Sunghwan Jung, Virginia Tech University

2:57PM D39.00001 Active flows on trees, ADEN FORROW, Department of Mathematics, MIT, FRANCIS G. WOODHOUSE, DAMTP, University of Cambridge, JRN DUNKEL, Department of Mathematics, MIT — Coherent, large scale dynamics in many nonequilibrium physical, biological, or information transport networks are driven by small-scale local energy input. We introduce and explore a generic model for compressible active flows on tree networks. In contrast to thermally-driven systems, active friction selects discrete states with only a small number of oscillation modes activated at distinct fixed amplitudes. This state selection can interact with graph topology to produce different localized dynamical time scales in separate regions of large networks. Using perturbation theory, we systematically predict the stationary states of noisy networks. Our analytical predictions agree well with a Bayesian state estimation based on a hidden Markov model applied to simulated time series data on binary trees. While the number of stable states per tree scales exponentially with the number of edges, the mean number of activated modes in each state averages $\sim 1/4$ the number of edges. More broadly, these results suggest that the macroscopic response of active networks, from actin-myosin networks in cells to flow networks in *Physarum polycephalum*, can be dominated by a few select modes.

3:10PM D39.00002 Low Reynolds number flow near tiny leaves, stems, and trichomes¹, CHRISTOPHER STRICKLAND, University of North Carolina, Chapel Hill, VIRGINIA PASOUR, Army Research Office, LAURA MILLER, University of North Carolina, Chapel Hill — In terrestrial and aquatic environments such as forest canopies, grass fields, and seagrass beds, the density and shape of trunks, branches, stems, leaves and trichomes (the hairs or fine outgrowths on plants) can drastically alter both the average wind speed and profile through these environments and near each plant. While many studies of flow in these environments have focused on bulk properties of the flow at scales on the order of meters, the low Reynolds number flow close to vegetative structures is especially complex and relevant to nutrient exchange. Using three-dimensional immersed boundary simulations, we resolve the flow around trichomes and small leaves and quantify velocities, shear stresses, and mixing while varying the height and density of idealized structures.

¹National Science Foundation Grant DMS-1127914 to the Statistical and Applied Mathematical Sciences Institute, and the Army Research Office

3:23PM D39.00003 Effect of wind-induced drag on leaf shapes, JEAN-FRANCOIS LOUF, Virginia Tech, PIERRE NTOH SONG, Polytech'Marseille, TIM ZEHNBAUER, SUNGHWAN JUNG, Virginia Tech — Under windy conditions everyone can see leaves bending and twisting. From a geometrical point of view, a leaf is composed of two parts: a large flat plate called the lamina, and a small beam called the petiole, connecting the lamina to the branch/stem. While the wind is exerting forces (e.g. drag) on the lamina, the petiole undergoes twisting and bending stresses. To survive in harsh abiotic conditions, leaves might have evolved to form in many different shapes, resulting from a coupling between the lamina and the petiole. In this study we measure the twisting modulus (G) of the petiole using a twisting setup, and its Young modulus (E) by performing tensile tests. Micro-CT scan is used to precisely measure the cross section of the petiole allowing us to calculate the second moment of inertia (I) and the second moment of area (J). We then use the non-dimensional number EI/GJ and compare it to a geometrical non-dimensional number $(L_{\text{petiole}}+L_{\text{lamina}}/2)/W$, where L_{petiole} is the length of the petiole, L_{lamina} the length of the lamina, and W the width of the lamina. We found a linear relation between the ratio of the bending to twisting rigidity and the leaf geometry.

3:36PM D39.00004 Smart change in leaf morphology to tune the wettability, HOSUNG KANG, SARA FLEETWOOD, SUNGHWAN JUNG, Virginia Tech — Plants are sessile organisms, but some of them are able to change their features to survive. We found *Cercidiphyllum japonicum* (Katsura) leaves actively adapt to their fine structures on the leaf surface in response to external stimuli. It is fascinating how the structural changes can affect their physical properties. In this present study, we are investigating the effect of external environments (temperature, cell hydration, and acid rain) on microscale papillose epidermal cells and nanoscale waxes. Using environmental scanning electron microscopy (ESEM) and atomic force microscopy (AFM), we measured micro and nano structures of the Katsura leaves. We found a functional relation between the micro and nano structures and the contact angle of the leaf's surface. As the epidermal cells shrink and the waxes erode, the contact angle decreases. A simple Cassie-Baxter model based on the wettability of textured surfaces has been used to characterize changes of the contact angle.

3:49PM D39.00005 Stomatal design principles for gas exchange in synthetic and real leaves¹, KAARE H. JENSEN, KATRINE HAANING, Department of Physics, Technical University of Denmark, C. KEVIN BOYCE, Department of Geological Sciences, Stanford University, MACIEJ ZWIENIECKI, Department of Plant Sciences, University of California, Davis — Stomata are portals in plant leaves that control gas exchange for photosynthesis, a process fundamental to life on Earth. Gas fluxes and plant productivity depend on external factors such as light, water, and CO_2 availability and on geometric properties of the stomata pores. The link between stomata geometry and environmental factors have informed a wide range of scientific fields – from agriculture to climate science, where observed variations in stomata size and density is used to infer prehistoric atmospheric CO_2 content. However, the physical mechanisms and design principles responsible for major trends in stomatal patterning, are not well understood. Here we use a combination of biomimetic experiments and theory to rationalize the observed changes in stomatal geometry. We show that the observed correlations between stomatal size and density are consistent with the hypothesis that plants favor efficient use of space and maximum control of dynamic gas conductivity, and – surprisingly – that the capacity for gas exchange in plants has remained constant over at least the last 325 million years. Our analysis provides a new measure to gauge the relative performance of species based on their stomatal characteristics.

¹Supported by the Carlsberg Foundation (2013-01-0449), VILLUM FONDEN (13166) and the National Science Foundation (EAR-1024041).

4:02PM D39.00006 Dynamics of a freely-falling maple seed¹, INJAE LEE, HAECHON CHOI, Seoul National University — We conduct numerical simulations of a freely-falling maple seed using an immersed boundary method in a non-inertial reference frame (Kim and Choi, JCP, 2006). A three-dimensional seed model is obtained by scanning a maple seed. The seed reaches a steady autorotation after a transient period, and a stable leading-edge vortex is attached on the surface of the rotating seed, which increases the drag force during autorotation. In addition, two different approaches are considered to obtain scaling laws describing the relation among the seed weight and geometry, and descending and rotating velocities. The first uses the conservations of mass, linear and angular momentum, and energy. In this approach, a model constant to be determined, called axial induction factor, is obtained from the result of present simulation. The second approach employs a classical steady wing theory in which the vortical strength is scaled with the circulation around a wing and the lift force is modeled by the time derivative of vortical impulse (Lee et al., JFM, 2015). Available data on various seeds well fall on these scaling laws.

¹Supported by NRF-2014M3C1B1033848

4:15PM D39.00007 The flight of *Ruellia ciliatiflora* seeds, DWIGHT WHITAKER, ERIC COOPER, MOLLY MOSHER, YIJUN WANG, CHAELEE DALTON, Pomona College — The fruits of *Ruellia ciliatiflora* open explosively and launch mm-sized disks at speeds exceeding 10 m/s a distance of 5 m. Observations with high-speed video reveal that the seeds are launched in a streamline orientation that is maintained with a backspin of 1.5 kHz. Through a careful analysis of the high-speed videos of the seeds' flight we measure the aerodynamic forces on these spinning seeds. We find that the exceptional rotation rate both reduces drag on the seed by keeping its cross section as small as possible and generates a modest ($\sim 0.3g$) lift on the flying seeds. To understand the aerodynamic forces we create photometrically scanned, 3D printed models of the seeds for particle image velocimetry (PIV) in a flume or tow tank. We will discuss our method for producing accurately shaped model seeds as well as preliminary PIV data on the flow of fluid around the flying seed. This work marks the start of a longer-term project that will compare the dynamics of seed launch and flight within the Acanthaceae family, which has over 2000 species in habitats ranging from rainforest to savannah that all use a similar method for launching seeds.

4:28PM D39.00008 Fluid mechanics of osmotic pipe flows and limitations on the lengths of conifer needles., TOMAS BOHR, HANNA RADEMAKER, KAARE JENSEN, Physics Department, Technical University of Denmark, MACIEJ ZWIENIECKI, Department of Plant Sciences, University of California, Davis — Plant leaves produce sugars, which are exported osmotically through the sieve tubes of the leaf. Leaf sizes vary by more than 3 orders of magnitude, from a few millimeters to over one meter. Conifer leaves (needles), however, are relatively short and the majority of needles are no longer than 6 cm. The reason for this limitation is unknown, but we argue that it can be explained by the linear venation pattern and the narrow sieve tubes, combined with the osmotic flow mechanism. Thus sugars produced near the tip of long needles cannot be exported efficiently, because the pressure required to drive vascular flow would exceed the greatest available pressure (the osmotic pressure). This basic constraint leads to the formation of an inactive region of stagnant fluid near the needle tip, which does not contribute to sugar flow. The active region, emerging from the base of the needle, has the length $L_{eff} = r^{3/2} / (16\eta L_p)^{1/2}$, where r is the conduit radius, η is the sap viscosity, and L_p is the cell membrane permeability. It is independent of the needle length and corresponds well with maximal needle lengths observed in nature.

4:41PM D39.00009 The life of bubbles under negative pressure, JIN WOO CHOI, KEUNHWAN PARK, Seoul National University, SO NAGASHIMA, MYOUNG-WOON MOON, Korea Institute of Science and Technology, HO-YOUNG KIM, Seoul National University — Cavitation of sap in plant vessels, or embolism, may occur when the liquid pressure becomes negative either in a high elevation or a dry environment. Effective suppression of nucleation and growth of cavitation bubbles is important for continuous transport of water and thus survival of the plant. Here we investigate the life of cavitation bubbles under negative pressure from their nucleation through growth and maturation. As a model system for the plant vessel, we fabricate hydrogel microchannels whose inner pressure is reduced to a negative value. The roughness of the channel surface is modified by plasma treatment to form wrinkles emulating observed xylem wall surfaces. We find a finite effect of surface wrinkles on the critical nucleation pressure. Also, dense wrinkles tend to slow down bubble growth. In all the channel roughness conditions, the bubbles grow diffusively with time until their maturation. Then in the matured stage, the growth speed is substantially lowered and follows the value determined by Darcys law. Our results suggest that surface wrinkles or roughness can be used to control the nucleation pressure and bubble growth behavior. Also, the observations can give deeper insight into embolism control mechanisms of tall trees.

4:54PM D39.00010 Breakdown of water transport and resilient xylem structure in vascular plants, JEONGEUN RYU, Department of Mechanical Engineering, POSTECH, WONJUNG KIM, Department of Mechanical Engineering, Sogang University, SANG JOON LEE, Department of Mechanical Engineering, POSTECH — Plants can transport sap water without using a mechanical pump by exploiting a metastable state of water. However, sap water in a metastable state is vulnerable to cavitation and embolism, disrupting water transport in xylem vessels. We note that under this paradox, plants have been evolved to have resilient xylem network against breakdown of water transport as a survival strategy. In this study, we directly observe the onset of embolism and its spreading dynamics in live plants to establish a synthetic tree model. We also rationalize our experimental findings with a model describing embolism propagation under a metastable state of water and an interconnected xylem network structure which can minimize damages from cavitation and embolism. This study would shed light on the design of complex networks with resilience for effective transport as well as the physical understanding on the transport of metastable water.

Sunday, November 20, 2016 2:57PM - 5:07PM –
Session D40 Porous Media Flows: General Portland Ballroom 253-258-254-257 - Arezoo Ardekani, Purdue University

2:57PM D40.00001 Title: Spatial velocity fluctuations in flow through porous media, SOROUSH ARAMIDEH, TIANQI GUO, PAVLOS P. VLACHOS, AREZOO M. ARDEKANI, Purdue Univ — Understanding the flow in porous media is of great importance and has direct impact on many processes in chemical and oil industries, fuel cell design, and filtration. In this work, we use direct numerical simulations (DNS) to examine the flow through variety of sphere packings with different levels of complexity and heterogeneity. DNS results are validated with velocity fields obtained via volumetric particle tracking velocimetry at high resolution. We show that flow in random close packing of spheres has unique statistical properties while the medium is random itself. Furthermore, we quantify the relationship between pore geometry and velocity fluctuations.

3:10PM D40.00002 Volumetric microscale particle tracking velocimetry (PTV) in porous media., TIANQI GUO, SOROUSH ARAMIDEH, AREZOO M. ARDEKANI, PAVLOS P. VLACHOS, Purdue University — The steady-state flow through refractive-index-matched glass bead microchannels is measured using microscopic particle tracking velocimetry (μ PTV). A novel technique is developed to volumetrically reconstruct particles from oversampled two-dimensional microscopic images of fluorescent particles. Fast oversampling of the quasi-steady-state flow field in the lateral direction is realized by a nano-positioning piezo stage synchronized with a fast CMOS camera. Experiments at different Reynolds numbers are carried out for flows through a series of both monodispersed and bidispersed glass bead microchannels with various porosities. The obtained velocity fields at pore-scale (on the order of 10 μ m) are compared with direct numerical simulations (DNS) conducted in the exact same geometries reconstructed from micro-CT scans of the glass bead microchannels. The developed experimental method would serve as a new approach for exploring the flow physics at pore-scale in porous media, and also provide benchmark measurements for validation of numerical simulations.

3:23PM D40.00003 Morse-Smale spectra reveal topological phase transition in porous media flow, NORBERT STOOP, Massachusetts Inst of Tech-MIT, NICOLAS WAISBORD, Tufts University, VASILY KANTSLER, University of Warwick, JEFFREY S. GUASTO, Tufts University, JOERN DUNKEL, Massachusetts Inst of Tech-MIT — We introduce spectral Morse-Smale analysis to identify topological phase transitions in disordered continuous media. Combining microfluidic experiments with large-scale, pore-resolved simulations of porous media flow, we demonstrate that invariants of Morse-Smale graphs of flow speed provide a well-defined measure of the effects of spatial disorder on fluid transport. By systematically perturbing a microfluidic lattice, the fluid flow topology undergoes a phase transition from periodic to filamentous flow structure, which corresponds to a change in the spectral density of the Morse-Smale graphs and carries important implications for advective transport and front dispersion. Due to its generic formulation, the proposed spectral Morse-Smale analysis can be extended to characterize topological transformations in physical, chemical or biological continuum systems.

3:36PM D40.00004 Preferential paths in yield stress fluid flow through a porous medium¹, JEFFREY GUASTO, NICOLAS WAISBORD, Tufts University, NORBERT STOOP, JÖRN DUNKEL, MIT — A broad range of biological, geological, and industrial materials with complex rheological properties are subjected to flow through porous media in applications ranging from oil recovery to food manufacturing. In this experimental study, we examine the flow of a model yield stress fluid (Carbopol micro-gel) through a quasi-2D porous medium, fabricated in a microfluidic channel. The flow is driven by applying a precisely-controlled pressure gradient and measured by particle tracking velocimetry, and our observations are complemented by a pore-network model of the yield stress fluid flow. While remaining unyielded at small applied pressure, the micro-gel begins to yield at a critical pressure gradient, exhibiting a single preferential flow path that percolates through the porous medium. As the applied pressure gradient increases, we observe a subsequent coarsening and invasion of the yielded, fluidized network. An examination of both the yielded network topology and pore-scale flow reveal that two cooperative phenomena are involved in sculpting the preferential flow paths: (1) the geometry of the porous microstructure, and (2) the adhesive surface interactions between the micro-gel and substrate.

¹NSF CBET-1511340

3:49PM D40.00005 Simulation of incompressible two-phase flow in porous media with large timesteps, DANIEL COGSWELL, MICHAEL SZULCZEWSKI, Aramco Services Company — Simulations of flow in porous media suffer from severe timestep restrictions as the permeability and viscosity contrast become increasingly heterogeneous, even when solved with a fully implicit discretization. Previous efforts to alleviate these restrictions have focused on numerical methods, but the problem persists because it originates from the shape of the fractional flow function. Here we focus on regularizing the equations themselves with the addition of an energy constraint. The equations for the flow of two immiscible, incompressible fluid phases in porous media are recast as a gradient flow using the phase-field method, a macroscopic surface tension is introduced, and a convex energy splitting scheme is applied to enable unconditionally large timesteps. Using the phase-field formulation as a homotopy map, the unmodified flow equations can be solved with large timesteps, even with high degrees of heterogeneity in permeability and viscosity. For a 2D test problem, the homotopy method allows the timestep to be increased by more than four orders of magnitude relative to the unmodified equations.

4:02PM D40.00006 Dispersive effects on the multi-layer porous media flows with permeable and impermeable interfaces¹, PRABIR DARIPA, CRAIG GIN, Texas AM University — We investigate dispersive effects on the linear stability of multi-layer porous media flow models of enhanced oil recovery for two different types of interfaces: permeable and impermeable interfaces. Results presented are relevant for the design of smarter interfaces in the available parameter space of Capillary number, Peclet number, longitudinal and transverse dispersion and the viscous profile of the middle layer. The stabilization capacity of each of these two interfaces is explored numerically and conditions for complete dispersive stabilization are identified for each of these two types of interfaces. Several key results will be presented including our finding that for most values of the flow parameters, permeable interfaces suppress flow instability more than impermeable interfaces. Time permitting, full simulation results will also be presented.

¹NSF grant DMS-1522782 and QNRF NPRP grant 08-777-1-141

4:15PM D40.00007 Diffuse-Interface Modelling of Flow in Porous Media, DOUG ADDY, Department of Chemical Engineering, Imperial College, London, UK, MARC PRADAS, Department of Mathematics and Statistics, Open University, UK, MARCUS SCHMUCK, School of Mathematical and Computer Sciences and the Maxwell Institute for Mathematical Sciences, Heriot-Watt University, UK, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College, London, UK — Multiphase flows are ubiquitous in a wide spectrum of scientific and engineering applications, and their computational modelling often poses many challenges associated with the presence of free boundaries and interfaces. Interfacial flows in porous media encounter additional challenges and complexities due to their inherently multiscale behaviour. Here we investigate the dynamics of interfaces in porous media using an effective convective Cahn-Hilliard (CH) equation recently developed in [1] from a Stokes-CH equation for microscopic heterogeneous domains by means of a homogenization methodology, where the microscopic details are taken into account as effective tensor coefficients which are given by a Poisson equation. The equations are decoupled under appropriate assumptions and solved in series using a classic finite-element formulation with the open-source software FEniCS. We investigate the effects of different microscopic geometries, including periodic and non-periodic, at the bulk fluid flow, and find that our model is able to describe the effective macroscopic behaviour without the need to resolve the microscopic details. [1] M. Schmuck, M. Pradas, G.A. Pavliotis and S. Kalliadasis, 2013, Nonlinearity **26** 3259-3277.

4:28PM D40.00008 Modeling and Simulations of Particulate Flows through Functionalized Porous Media¹, CHUNHUI LI, PRASHANTA DUTTA, JIN LIU, Washington State University — Transport of particulate fluid through a functionalized porous material is of significant interest in many industrial applications, such as earth sciences, battery designs and water/air purifications. The entire process is complex, which involves the convection of fluid, diffusion of reactants as well as reversible chemical reactions at the fluid-solid interface. In this work we present a convection-diffusion-reaction model and simulate the transport of particulate fluid through a functionalized porous media. The porous structures are generated and manipulated through the quartet structure generation set method. The Navier-Stokes with convection-diffusion equations are solved using the lattice Boltzmann method. The chemical reactions at the interface are modeled by an absorption-desorption process and treated as the boundary conditions for above governing equations. Through our simulations we study the effects of porous structures, including porosity, pore orientation, and pore size as well as the kinetic rates of surface reactions on the overall performance of removal efficiency of the species from the solution. Our results show that whole process is highly affected by both the porous structures and absorption rate. The optimal parameters can be achieved by proper design.

¹This work is supported by NSF grants: CBET-1250107 and CBET -1604211

4:41PM D40.00009 Diffusion in random networks: Asymptotic properties, and numerical and engineering approximations, JUAN C. PADRINO, DUAN Z. ZHANG, Los Alamos National Laboratory — The ensemble phase averaging technique is applied to model mass transport by diffusion in random networks. The system consists of an ensemble of random networks, where each network is made of a set of pockets connected by tortuous channels. Inside a channel, we assume that fluid transport is governed by the one-dimensional diffusion equation. Mass balance leads to an integro-differential equation for the pores mass density. The so-called dual porosity model is found to be equivalent to the leading order approximation of the integration kernel when the diffusion time scale inside the channels is small compared to the macroscopic time scale. As a test problem, we consider the one-dimensional mass diffusion in a semi-infinite domain, whose solution is sought numerically. Because of the required time to establish the linear concentration profile inside a channel, for early times the similarity variable is $xt^{-1/4}$ rather than $xt^{-1/2}$ as in the traditional theory. This early time sub-diffusive similarity can be explained by random walk theory through the network. In addition, by applying concepts of fractional calculus, we show that, for small time, the governing equation reduces to a fractional diffusion equation with known solution. We recast this solution in terms of special functions easier to compute. Comparison of the numerical and exact solutions shows excellent agreement.

4:54PM D40.00010 Influence of the multiphase flow patterns on the transport properties

BOJAN MARKICEVIC, Pall Corp, Cortland NY — The capillary network model with the dynamic boundary condition at free interface for displacement flow in porous media is developed, in which the net flow into each pore at the free interface can be less, equal to or greater than zero. The spread of the liquid and form of the liquid flow patterns are resolved in the networks of different sizes and heterogeneity and for two types of the boundary conditions, the constant inlet pressure or constant flow rate. It has been shown that for the constant flow rate boundary condition, the pressure drop throughout the network remains constant due to the pressure increase at the inlet boundary. The constant pressure drop produces the similar flow patterns during the displacement flow, and the saturation remains constant in the flow direction. The liquid saturation in the network is varied gradually by increasing the liquid flow rate at network inlet. For each distinct flow rate, the sizes of the repeating flow pattern and corresponding pressure drop change accordingly. This implies that for sufficiently large networks in which the flow pattern is fully developed, the transport parameters do not depend on the network size. The flow pattern and transport parameters depend on the network heterogeneity, as the dynamic boundary condition changes at the free interface producing a different distribution of the liquid. A continuous development of the flow pattern is also observed for the constant inlet pressure boundary condition, where the pressure drop decreases as liquid advances into the network. Finally, a summary in changes of transport parameters, relative permeability and capillary pressure, is elaborated.

Sunday, November 20, 2016 5:37PM - 6:55PM –

Session E1 Mini-Symposium: Geophysical Turbulence Induced by Flow over Topography III A105 - Ruth Musgrave, Massachusetts Institute of Technology

5:37PM E1.00001 Internal waves trapped below a virtual seafloor

HARRY SWINNEY, University of Texas at Austin — Numerical simulations of tidal flow over random topography with the ocean seafloor spectrum reveal that the time-averaged tidal energy converted into far field internal wave radiation arises only from topography that rises above a local “virtual” seafloor (Zhang and Swinney, Phys. Rev. Lett. 112, 104502 (2014)). For topography below the virtual seafloor, destructive wave interference leads to no time-averaged far field internal wave power. The concept of a virtual floor extends the applicability of linear theory to global predictions of the conversion of tidal energy into internal wave energy in the oceans. The simulations show that for increasing topographic RMS height the emergent interference of internal waves from neighboring generation sites leads to a transition in the radiated power dependence on topographic height from quadratic to linear. The internal wave power radiated by random topography is found to increase as the horizontal spatial resolution scale is decreased, down to a length scale of typically 300 m, but smaller scales generate at most a few percent of the radiated power (Zhao, Zhang, and Swinney, Geophys. Res. Lett. 42, 8081–8087 (2015)). Tidal flow past 3D seamounts generates much less total internal wave power than that radiated by quasi-2D ridges (Zhang, Swinney, Comino, and Buijsman (2016)).

6:03PM E1.00002 On turbulent mixing in stably stratified geophysical flows¹

KARAN VENAYAGAMOORTHY, Colorado State University — The understanding and quantitative prediction of diapycnal (irreversible) mixing of density and momentum in geophysical flows remains an important ongoing challenge. This is not surprising given the complexity introduced into most geophysical flows by factors such as density stratification, complex topography and a host of physical phenomena associated with such flows. However, accurate prediction of the small-scale irreversible mixing induced by turbulent processes is critical for many applications such as the prediction of heat fluxes and global circulation in oceanic flows. From a practical standpoint, a major goal is the inference of turbulent heat and momentum fluxes using indirect measurements in field studies of geophysical flows. This usually involves the need to either measure directly or infer two key quantities namely: (1) the rate of dissipation of turbulent kinetic energy ϵ , and (2) the mixing efficiency R_f^* , which is a measure of the amount of turbulent kinetic energy that is irreversibly converted into background potential energy, respectively. Indirect estimates of ϵ in oceanic flows has been traditionally achieved by assuming a linear relationship between the Thorpe (vertical overturn) length scale L_T and the Ozmidov scale L_O . This approach is particularly attractive since the vertical scales of overturns can be readily obtained using a sorting algorithm from inversions in standard density profiles obtained from Conductivity-Temperature-Depth (CTD) measurements in the ocean. Hence, L_T is essentially a kinematic scale that provides a description of the turbulence at a given sampling location. On the other hand, L_O is a representative dynamic length scale of the largest eddy that is unaffected by buoyancy. A review of a number of recent studies that were conducted in our research group will be presented in this talk to highlight the lack of a linear relationship between L_T and L_O . These studies indicate that inferred estimates of ϵ may be biased high by up to an order of magnitude or more especially for large overturns in the ocean. An alternative unifying framework using a two-dimensional parameter space based on a buoyancy strength parameter (i.e. an inverse Froude number) and a shear strength parameter will be discussed to characterize the scaling correspondence of the overturning length scale with pertinent turbulent length scales. The second key quantity that is a necessary ingredient for the inference of diapycnal mixing from oceanic measurements is the flux Richardson number R_f^* . To date, however, no unifying parameterization of R_f^* exists due to both the variability inherent in geophysical flows as well as certain ambiguities that are introduced in descriptions based on ill-conditioned single parameters. A discussion on the mixing efficiency and implications for estimates of diapycnal mixing in geophysical flows will also be presented.

¹Funded by the Office of Naval Research and the National Science Foundation

6:29PM E1.00003 Internal Hydraulic Jumps in Shallow Flows over Topography

KELLY OGDEN, Massachusetts Institute of Technology/ Woods Hole Oceanographic Institution — A barotropically forced stratified flow over topography can generate an internal hydraulic jump with upstream shear. The structure and mixing of these jumps are investigated theoretically and numerically. The effect of upstream shear on simplified jumps in two-layer flows without topography results in jump types such as undular bores, smooth front turbulent jumps, and fully turbulent jumps (Ogden and Helffrich, 2016). Increased shear results in entrainment across the jump with jump structures that resemble expanding shear layers. The addition of topography increases the number of qualitative jump types. Idealized simulations are conducted to characterize the types of jumps that can occur under various parameter regimes. The effect of parameters such as the volume flow rate and topographic height are considered. Flow structures including first-mode jumps with wave overturning and higher-mode jumps with wedges of homogeneous stagnant fluid are found. The degree of mixing and the mass budget of the developing stagnant wedge illuminate the important physical characteristics of each jump type. Existing hydraulic jumps in the environment are compared to the parameter regimes the identified jump types. The applicability of two-layered theories for studying these jumps is also considered.

Sunday, November 20, 2016 5:37PM - 6:55PM –

Session E2 Wind Turbines: Field Applications A106 - Jiarong Hong, University of Minnesota

5:37PM E2.00001 Wind turbine wake meandering at the laboratory and field scales ,

MICHAEL HEISEL, MIRKO MUSA, JIARONG HONG, MICHELE GUALA, St Anthony Falls Laboratory, University of Minnesota - Twin Cities — Flow measurements were collected in the wake of the utility-scale (2.5MW) Eolos wind turbine using a ground-based light detection and ranging (LiDAR) wind profiler to identify the characteristics of wake meandering at the field scale. The investigation seeks to establish the influence of scale and atmospheric turbulence on wake meandering, which has been observed to leave a strong spectral signature on laboratory measurements in wind tunnel and channel flows. The experimental data include multiple test periods at various downstream distances within the turbine wake. Inflow conditions were assessed using a meteorological tower equipped with sonic anemometers. Additionally, an experiment was conducted in the Saint Anthony Falls Laboratory atmospheric boundary layer wind tunnel to provide a direct comparison for the utility-scale results and to reaffirm the findings of previous laboratory-scale investigations. Estimates of the wake and inflow one-dimensional velocity spectra were compared to determine whether wake meandering characteristics are present at both scales. An empirical correction to the velocity spectra of the LiDAR and a few options to extract a more local velocity signal are discussed to compensate for the inherent limitations of LiDAR in capturing turbulent fluctuations.

5:50PM E2.00002 Investigating coherent vortex structures in the near wake of a utility-scale wind turbine using flow visualization with natural snowfalls¹ , TEJA DASARI ,

JIARONG HONG, University of Minnesota — Flow visualization techniques using natural snowfall have been shown as an effective tool to probe coherent flow structures around utility-scale wind turbines (Hong et al. *Nature Comm.* 2014). Here we present a follow-up study using the data collected during multiple deployments from 2014 to 2016 around the 2.5 MW turbine at EOLOS wind energy research station. The data include flow visualization from different perspectives in the near wake of the turbine. Coherent wake structures, including blade tip vortex, trailing vortex sheet, nacelle-generated structures, and tower vortex characterized by the snow voids, are correlated with atmospheric conditions (e.g. turbulence intensity), turbine operational conditions (e.g. power and tip-speed ratio) as well as turbine response (e.g. tower and blade strain). Physical factors and processes that affect the features and the behaviors of tip vortices including their void size and shape, their stability (e.g. meandering and intermittent appearance) and vortex interaction (e.g. vortex merging and leapfrogging) are analyzed. In particular, a strong influence of the tower on tip-vortex structures is demonstrated through simultaneous comparison of vortex voids at elevations below and above the height of nacelle and the plan view visualization.

¹Sponsored by NSF Fluid Dynamics Program

6:03PM E2.00003 Drone Based Experimental Investigation of Wind Turbine Wake Evolution , DR. BALAJI SUBRAMANIAN, Fluid Energy Science Laboratory, University of California Santa Barbara, DR. NDAONA CHOKANI, PROF. DR. REZA ABHARI, Institut fr Energietechnik, LEC, ETH Zrich, Switzerland. —

The characteristics of the wake downstream of a wind turbine has an important bearing on the optimized micrositing of wind turbines in a given land area, as well as on the loads seen by downstream turbines. We use a novel measurement system to measure the flow field upstream and in the wake of a full-scale wind turbine. The system consists of a fast response aerodynamic probe, mounted on an autonomous drone that is equipped with a suite of sensors. These measurements detail, for the first time at full-scale Reynolds number conditions, the evolution and breakdown of tip vortices that are characteristic of the near wake, as well as the turbulent mixing and entrainment of more energised flow, which are distinctive in the far wake. A short-time Fourier transform (STFT) analysis method is used to derive time-localized TKE along the drone's trajectory. Detailed upstream and wake measurements are needed to understand the flow behavior, as it helps in developing and validating simplified wake models that can approximate the wake qualities. Comparisons of these measurements to recently developed wake prediction models highlights how these measurements can support further model development.

6:16PM E2.00004 Analysis of near-surface relative humidity in a wind turbine array boundary layer using an instrumented unmanned aerial system and large-eddy simulation , KEVIN ADKINS, OUMNIA ELFAJRI, ADRIAN SESCU, Mississippi State University —

Simulation and modeling have shown that wind farms have an impact on the near-surface atmospheric boundary layer (ABL) as turbulent wakes generated by the turbines enhance vertical mixing. These changes alter downstream atmospheric properties. With a large portion of wind farms hosted within an agricultural context, changes to the environment can potentially have secondary impacts such as to the productivity of crops. With the exception of a few observational data sets that focus on the impact to near-surface temperature, little to no observational evidence exists. These few studies also lack high spatial resolution due to their use of a limited number of meteorological towers or remote sensing techniques. This study utilizes an instrumented small unmanned aerial system (sUAS) to gather in-situ field measurements from two Midwest wind farms, focusing on the impact that large utility-scale wind turbines have on relative humidity. Results are also compared to numerical experiments conducted using large eddy simulation (LES). Wind turbines are found to differentially alter the relative humidity in the downstream, spanwise and vertical directions under a variety of atmospheric stability conditions.

6:29PM E2.00005 Proactive monitoring of an onshore wind farm through lidar measurements, SCADA data and a data-driven RANS solver¹ , GIACOMO VALERIO IUNGO, UT Dallas,

SIMONE CAMARRI, University of Pisa, UMBERTO CIRI, SAID EL-ASHA, STEFANO LEONARDI, MARIO A ROTEA, VIGNESH SANTHANAGOPALAN, UT Dallas, FRANCESCO VIOLA, EPFL, LU ZHAN, UT Dallas — Site conditions, such as topography and local climate, as well as wind farm layout strongly affect performance of a wind power plant. Therefore, predictions of wake interactions and their effects on power production still remain a great challenge in wind energy. For this study, an onshore wind turbine array was monitored through lidar measurements, SCADA and met-tower data. Power losses due to wake interactions were estimated to be approximately 4% and 2% of the total power production under stable and convective conditions, respectively. This dataset was then leveraged for the calibration of a data driven RANS (DDRANS) solver, which is a compelling tool for prediction of wind turbine wakes and power production. DDRANS is characterized by a computational cost as low as that for engineering wake models, and adequate accuracy achieved through data-driven tuning of the turbulence closure model. DDRANS is based on a parabolic formulation, axisymmetry and boundary layer approximations, which allow achieving low computational costs. The turbulence closure model consists in a mixing length model, which is optimally calibrated with the experimental dataset. Assessment of DDRANS is then performed through lidar and SCADA data for different atmospheric conditions.

¹This material is based upon work supported by the National Science Foundation under the I/UCRC WindSTAR, NSF Award IIP 1362033

6:42PM E2.00006 A Biomimetic Ultrasonic Whistle for Use as a Bat Deterrent on Wind Turbines¹, PAUL SIEVERT, University of Massachusetts, Amherst, BANAFSHEH SEYED-AGHAZADEH, Miami University, DANIEL CARLSON, ZARA DOWLING, YAHYA MODARRES-SADEGHI, University of Massachusetts, Amherst — As wind energy continues to gain worldwide prominence, more and more turbines are detrimentally influencing bat colonies. In 2012 alone, an estimated 600,000 bats were killed by wind turbines in the United States. Bats show a tendency to fly towards turbines. The objective of this work is to deter bats from the proximity of the swept area of operational wind turbine blades. Established field studies have shown that bats avoid broadband ultrasonic noise on the same frequency spectrum as their echolocation chirps. A biomimetic ultrasonic pulse generator for use as a bat deterrent on wind turbines is designed and studied experimentally. This device, which works based on the fundamentals of flow-induced oscillations of a flexible sheet is a whistle-like device inspired by a bat larynx, mechanically powered via air flow on a wind turbine blade. Current device prototypes have proven robust at producing ultrasound across the 20 - 70 kHz range for flow inlet velocities of 4 – 14 m/s. Ultimately, a deterrent as described here could provide a reliable, cost-effective means of alerting bats to the presence of moving turbine blades, reducing bat mortality at wind facilities, and reducing regulatory uncertainty for wind facility developers.

¹The financial support provided by the US Department of Energy, and the Massachusetts Clean Energy center is acknowledged.

Sunday, November 20, 2016 5:37PM - 6:55PM – Session E3 Vortex Dynamics: Theory B110-111 - Kamran Mohseni, University of Florida

5:37PM E3.00001 Sources of vorticity at interface curvature singularities and the triple contact point, PETER ZHANG, KAMRAN MOHSENI, University of Florida — In our recent two-phase experiments [DeVoria & Mohseni, Phys. Fluids, 27(1), 2015], high concentrations of positive and negative vorticity have been observed near the moving contact line. These distributions suggest that the moving contact line, characterized by singular interface curvature, may be a unique source of vorticity. Motivated by this possibility, we conduct an analytic investigation of vorticity generation near sharp corners. To model the problem, we assume that the fluid is governed by the Stokes flow equations whose solutions can be found analytically. The general solution is composed of an exterior and interior multipole expansion, indicating sources of vorticity at or far from the corner respectively. A vorticity monopole, characterized by constant vorticity generation from the corner singularity, is observed for corner flows with logarithmic interface normal velocity only. A vorticity dipole and quadrupole are identified as the vorticity distribution for a moving contact line and interface cusp respectively. Using the analytic solution, exact relations for the vorticity multipole strengths and orientations are derived. A comparison of the analytic model with experimental measurements and numerical simulations show agreement in the vicinity of the corner.

5:50PM E3.00002 The hydrodynamic vortex: an exactly solvable black hole analogue, NAIL USSEMBAYEV, KAUST — We consider the Cauchy problem for the Klein-Gordon equation on an effective Lorentzian manifold describing the sound propagation on a background flow undergoing a subsonic-supersonic transition. For the hydrodynamic vortex model, a particular case of a draining bathtub geometry with non-zero circulation and no draining, we derive an exact solution and discuss its properties.

6:03PM E3.00003 The boundary-constraint method for constructing vortex-surface fields¹, SHIYING XIONG, YUE YANG, Peking Univ — We develop a boundary-constraint method for constructing the vortex-surface field (VSF) in a three-dimensional fluid velocity field. The isosurface of VSF is a vortex surface consisting of vortex lines, which can be used to identify and track the evolution of vortical structures in a Lagrangian sense. The evolution equation with pseudo-time is solved under the boundary constraint of VSF to obtain an approximate solution of VSF. Using the boundary-constraint method, we construct the VSFs in Taylor-Green flow and transitional channel flow. The uniqueness of VSF are demonstrated with different initial conditions, and the consistency of this boundary-constraint method and the previous two-time approach for constructing VSF is discussed. In addition, the convergence error in the calculation of VSF is analyzed.

¹This work has been supported in part by the National Natural Science Foundation of China (Grant Nos. 11522215 and 11521091), and the Thousand Young Talents Program of China.

6:16PM E3.00004 Extreme Growth of Enstrophy on 2D Bounded Domains¹, BARTOSZ PROTAS, ADAM SLIWIAK, McMaster Univ — We study the vortex states responsible for the largest instantaneous growth of enstrophy possible in viscous incompressible flow on 2D bounded domain. The goal is to compare these results with estimates obtained using mathematical analysis. This problem is closely related to analogous questions recently considered in the periodic setting on 1D, 2D and 3D domains. In addition to systematically characterizing the most extreme behavior, these problems are also closely related to the open question of the finite-time singularity formation in the 3D Navier-Stokes system. We demonstrate how such extreme vortex states can be found as solutions of constrained variational optimization problems which in the limit of small enstrophy reduce to eigenvalue problems. Computational results will be presented for circular and square domains emphasizing the effect of geometric singularities (corners of the domain) on the structure of the extreme vortex states.

¹Supported by an NSERC (Canada) Discovery Grant

6:29PM E3.00005 Maximum Production of Enstrophy in Swirling Viscous Flows, DIEGO AYALA, CHARLES DOERING, Univ of Michigan - Ann Arbor — We study a family of axisymmetric vector fields that maximize the instantaneous production of enstrophy in 3-dimensional (3D) incompressible viscous flows. These vector fields are parametrized by their energy \mathcal{K} , enstrophy \mathcal{E} and helicity \mathcal{H} , and are obtained as the solution of suitable constrained optimization problems. The imposed symmetry is justified by the results reported in the seminal work of Doering & Lu (2008), recently confirmed independently by Ayala & Protas (2016), where highly-localized pairs of colliding vortex rings are found to be optimal for enstrophy production. The connection between these optimal axisymmetric fields and the “blow-up” problem in the 3D Navier-Stokes equation is discussed.

6:42PM E3.00006 The critical swirl of a subsonic compressible swirling flow of a perfect gas in a finite-length straight circular pipe, HARRY LEE, U. Michigan, ZVI RUSAK, Rensselaer Polytechnic Institute, SHIXIAO WANG, U. Auckland, New Zealand — Functional analysis techniques are used to rigorously determine the range of flow Mach number Ma_0 for the existence of the critical swirl ratio ω_1 for exchange of stability of a base columnar compressible swirling flow of a perfect gas in a finite-length straight circular pipe. For swirling flows with a monotonic circulation profile, it is first established that ω_1 definitely exists in the range $0 < Ma_0 < 2\sqrt{\gamma - 1}/\gamma$, where $\gamma > 1$ is the ratio of specific heats of the gas. Then, the existence of a limit Mach number Ma_{0l} between $2\sqrt{\gamma - 1}/\gamma$ and 1 is proven for a subclass of swirling flows; i.e. ω_1 does not exist and the flow is stable for all swirl level when $Ma_{0l} < Ma_0 < 1$. For example, $0.903 < Ma_{0l} < 1$ when $\gamma = 1.4$. In particular, an analytical solution of ω_1 as a function of Ma_0 , γ and pipe length L for a solid-body rotation flow with a uniform axial velocity and temperature is also derived. The asymptotic behavior of this solution as Ma_0 tends to zero matches the results of Renac *et al.* (2007). In addition, Ma_{0l} of this flow is between 0.903 and 0.928 for $\gamma = 1.4$; it increases from 0.903 to 0.928 as L tends to infinity. This matches Rusak & Lee (2002) results.

Sunday, November 20, 2016 5:37PM - 6:55PM –

Session E4 Multiphase Flows: Cavitation B112 - Casey Harwood, University of Michigan

5:37PM E4.00001 The Hydroelastic Response of a Flexible Surface-Piercing Strut in Wetted, Ventilated, and Cavitating Flows¹, CASEY HARWOOD, JACOB WARD, YIN LU YOUNG, University of Michigan, MARIO FELLI, MASSIMO FALCHI, CNR INSEAN, STEVEN CECCIO, University of Michigan — High-speed and highly loaded lifting surfaces are prone to ventilation and cavitation. Increasing use of compliant materials (e.g. composites) in such systems necessitates a better understanding of the fluid-structure interactions of lifting surfaces in multiphase flow. Experiments on a flexible surface-piercing hydrofoil have been performed in a towing tank and a free-surface cavitation tunnel. The objectives are (i) to demonstrate the effects of material compliance upon hydrodynamic performance and stability of multiphase flow regimes, and (ii) to quantify the effects of multiphase flow upon the structural response and hydroelastic stability of flexible lifting bodies. A non-optical shape-sensing method is developed, which permits 3D bending and twisting deformations of the hydrofoil to be accurately inferred. The effects of the foil's compliance on hydrodynamic loads, structural motions and flow regimes are discussed. Partial immersion of the hydrofoil causes a mode-dependent change in added-mass that can encourage coalescence of higher modes. At the same time, increasing flow speed and ventilated flow decrease the damping associated with certain modes. Unsteady cavity shedding modulates the system parameters, causing a broadening of the frequency response.

¹The authors would like to acknowledge the support of Dr. Ki-Han Kim under ONR grant number N00014-13-1-0383 and N00014-16-1-2433.

5:50PM E4.00002 Study of Cavitation Shedding Dynamics on a NACA0015 Hydrofoil Using X-Ray Densitometry¹, HARISH GANESH, JULIANA WU, STEVEN CECCIO, University of Michigan — Cavitation dynamics on the NACA0015 hydrofoil at several attack angles are found to be spectrally rich, being multi-modal with abrupt changes in Strouhal number with change in cavitation number. Present study focusses on identifying the physical mechanisms responsible for the change in cavitation dynamics on a NACA0015 hydrofoil in a re-circulating water tunnel using time resolved X-ray densitometry. Time-resolved void fraction flow fields obtained using X-ray densitometry, synchronized with acoustic noise measurements using a hydrophone, are used to identify different flow features and mechanisms that are responsible for the change in the observed spectral behavior. It is shown that under higher cavitation numbers, the shedding mechanism is predominantly re-entrant liquid flow based, but as the cavitation number drops many different processes are at play. At lower cavitation numbers, the shed cavity cloud collapse arrests cavity growth and this results in altered cycle dynamics and hence the Strouhal number. In addition, propagation bubbly shock waves are also found to be a dominant mechanism of shedding for certain conditions. The multi-modal nature of the acoustic pressure signature is explained by presence of different flow features, which could be concurrent or alternating.

¹Office of Naval Research

6:03PM E4.00003 Cavitation Dynamics on a Backward Facing Wedge With Streamwise Grooves¹, JULIANA WU, HARISH GANESH, STEVEN CECCIO, University of Michigan — Cavitation dynamics in the separated region of backward facing wedges has been studied extensively in the past. From the previous studies it has been found that cavity shedding mechanisms could be due to the presence of a re-entrant liquid flow and/or a propagation bubbly shock wave. Recently, the effect of streamwise grooves on the aft portion of the wedge on the cavity shedding dynamics was explored. In the present study, X-ray densitometry is used to study the effect of streamwise grooves of different sizes and starting locations on the cavitation behavior on the wedge. Using time resolved void fraction flow fields from X-ray densitometry, the effect of the grooves in altering the flow structures and cavity shedding mechanisms are characterized.

¹Office of Naval Research

6:16PM E4.00004 Study of Cavitation in Wakes of Circular Cylinders and Symmetric Wedges Using X-ray Densitometry¹, JOACHIM KOOT, University of Twente, JULIANA WU, HARISH GANESH, STEVEN CECCIO, University of Michigan — Cavitation in wakes behind canonical objects can exhibit variation in Strouhal number with a reduction in cavitation number. Circular cylinders of two diameters and symmetric wedges with a wedge angle of 15, 30, and 60 degrees are used to study cavitation in their wakes using X-ray densitometry. Using high speed video and X-ray densitometry, the nature of cavitation is studied in near-wake and a part of the far-wake region. In addition, acoustic measurements are also carried out to understand the spectral content of such wake cavities. Based on void fraction flow field and high-speed video measurements, the effect of cavitation on the Kármán vortex street spacing in the far wake region is studied. The results are interpreted to explain the physical mechanisms responsible for the observed change in Strouhal number.

¹Office of Naval Research

6:29PM E4.00005 The effect of microbubble cavity formation on the breakup of glowing sprays, DENNIS VAN DER VOORT, NICO DAM, WILLEM VAN DE WATER, RUDIE KUNNEN, HERMAN CLERCX, GERTJAN VAN HEIJST, Eindhoven University of Technology — Spray atomization is affected by an interplay of several physical phenomena. To understand the breakup as a whole, it is essential to understand the contribution of individual phenomena (turbulence, cavitation, etc). This work investigates the contribution of cavity formation inside the nozzle by seeding a system with microbubbles, generating transient cavities. Using transparent nozzles, pressure sensors, and high-speed imaging, we find that the pressure pulses generated by cavity collapse can be used to correlate the time and magnitude of the spray angle increase, valuable for application in non-transparent setups. Laser-induced phosphorescence is used to quantitatively measure liquid dispersion, which excites ('tags') 10-20 nL of fluid at the nozzle exit and tracks the spread of glowing fluid. The dispersion correlated to cavity formation events is compared to the average. We show that the dispersion (which is independent of the starting tagged spray width) does not increase with the formation of a cavity. This indicates that, while the spray angle may increase, the turbulent processes (both liquid and gas phase) that governs the dispersion remains the same, and the cavitation events do not influence this process.

6:42PM E4.00006 Large eddy simulation of sheet to cloud cavitation¹, MRUGANK BHATT, KRISHNAN MAHESH, University of Minnesota - Twin Cities — Large eddy simulation is used to study sheet to cloud cavitation. A homogeneous mixture model is employed to represent the multiphase mixture of water and water vapor. A novel predictor-corrector method (Gnanaskandan and Mahesh, Int. Journal of Multiphase Flow, 2015, 70:2234) is used to numerically solve the compressible NavierStokes equations for the liquid/vapor mixture along with a transport equation for the vapor mass fraction. The algorithm is implemented on an unstructured grid and a parallel platform, with a fully coupled implicit time advancement of both viscous and advection terms. Simulation of sheet to cloud cavitation over a wedge at a Reynolds number, $Re = 200,000$ and cavitation number, $\sigma = 2.1$ is performed. A propagating condensation shock similar to the one observed in the experiments of Harish et.al. (Harish Ganesh, PhD thesis- University of Michigan, 2015) is observed in the computed flow field. Results will be presented and the flow physics will be discussed.

¹This work is supported by the Office of Naval Research

Sunday, November 20, 2016 5:37PM - 6:55PM –
Session E5 DFD GPC: Geophysical Fluid Dynamics: Earths Core & Dynamos B113
- Daniel Lathrop, University of Maryland

5:37PM E5.00001 A liquid sodium model of the Earth's core¹, DANIEL LATHROP, MATTHEW ADAMS, DOUGLAS STONE, MINH DOAN, University of Maryland — We present results from the three meter liquid sodium spherical Couette experiment at full speed (4 Hz outer sphere rotation rate and a range of inner sphere rates). The experiment is geometrically similar with the earth's core. We study hydrodynamic and hydromagnetic phenomena in rapidly rotating turbulence, as well as magnetic field induction by those flows. Two external electromagnets apply dipole or quadrupole magnetic fields, while an array of 31 external Hall sensors measure the resulting induced magnetic field. This allows us to study dynamo gain (as we yet have no self-generating magnetic dynamo) and broader range of rotating turbulence phenomena. We report substantial magnetic field gain for a variety of flow states. One of these states exhibits bistability in the hydrodynamic flow with magnetic field gain only in one of the two states. Zonal flow shear drives large azimuthal magnetic fields, prompting a need to measure the zonal flows. This has prompted us to develop acoustic mode velocimetry measurements adapted from helioseismology. Prior to measurements in the larger experiment, we develop this technique in our 60 cm diameter spherical Couette experiment in nitrogen gas. There, we compare acoustic mode frequency splittings with theoretical predictions for solid body flow and turbulent flow, and obtain excellent agreement. We also use this technique to estimate the zonal shear in those experiments.

¹NSF EAR 1417148

5:50PM E5.00002 Experimental Studies of Acoustics in a Spherical Couette Flow, SAVANNAH GOWEN, University of Maryland College Park and Mount Holyoke College, MATTHEW ADAMS, DOUGLAS STONE, DANIEL LATHROP, University of Maryland College Park — The Earth, like many other astrophysical bodies, contains turbulent flows of conducting fluid which are able to sustain magnetic field. To investigate the hydromagnetic flow in the Earth's outer core, we have created an experiment which generates flows in liquid sodium. However, measuring these flows remains a challenge because liquid sodium is opaque. One possible solution is the use of acoustic waves. Our group has previously used acoustic wave measurements in air to infer azimuthal velocity profiles, but measurements attempted in liquid sodium remain challenging. In the current experiments we measure acoustic modes and their mode splittings in both air and water in a spherical Couette device. The device is comprised of a hollow 30-cm outer sphere which contains a smaller 10-cm rotating inner sphere to drive flow in the fluid in between. We use water because it has material properties that are similar to those of sodium, but is more convenient and less hazardous. Modes are excited and measured using a speaker and microphones. Measured acoustic modes and their mode splittings correspond well with the predicted frequencies in air. However, water modes are more challenging. Further investigation is needed to understand acoustic measurements in the higher density media.

6:03PM E5.00003 Magnetostrophic Rotating Magnetoconvection¹, ERIC KING, Nike Sustainable Innovation , JONATHAN AURNOU, UCLA Earth and Space Sciences — Planetary magnetic fields are generated by turbulent convection within their vast interior liquid metal cores. Although direct observation is not possible, this liquid metal circulation is thought to be dominated by the controlling influences of Coriolis and Lorentz forces. Theory famously predicts that local-scale convection naturally settles into the so-called magnetostrophic state, where the Coriolis and Lorentz forces partially cancel, and convection is optimally efficient. To date, no laboratory experiments have reached the magnetostrophic regime in turbulent liquid metal convection. Furthermore, computational dynamo simulations have as yet failed to produce a globally magnetostrophic dynamo, which has led some to question the existence of the magnetostrophic state. Here, we present results from the first turbulent magnetostrophic rotating magnetoconvection experiments using the liquid metal gallium. We find that turbulent convection in the magnetostrophic regime is, in fact, maximally efficient. The experimental results clarify these previously disparate results, suggesting that the fluid dynamics saturate in magnetostrophic balance within turbulent liquid metal, planetary cores.

¹The authors thank the NSF Geophysics Program for financial support.

6:16PM E5.00004 Equatorially trapped convection in a rapidly rotating spherical shell, BENJAMIN MIQUEL, KEITH JULIEN, University of Colorado, Boulder, EDGAR KNOBLOCH, University of California, Berkeley — Convection plays a preponderant role in driving geophysical flows. Unfortunately, these flows are often characterized by rapid rotation (i.e. small Ekman number E) which renders the equations stiff and introduces a scale separation in the system: for example the wavelength of the marginal mode at the onset of convection in a rapidly rotating sphere scales like $E^{1/3}$ and is modulated by a $E^{1/6}$ envelope. These scalings keep the fully nonlinear dynamics of the internal convection in Earth's core ($E \sim 10^{15}$) out of reach from direct numerical simulations, analytical work and experiments on one hand, but advocate for the development of reduced models on the other hand. We present a reduced model derived in a shallow gap spherical shell geometry. As the Rayleigh number is increased, the flow is first destabilized in the equatorial region where the dynamics remains trapped. The linear stability is analyzed and the fully nonlinear dynamics is presented.

6:29PM E5.00005 Identification of dominant flow structures in rapidly rotating convection of liquid metals using Dynamic Mode Decomposition, SUSANNE HORN, Department of Mathematics, Imperial College London, JONATHAN M. AURNOU, Department of Earth, Planetary, and Space Sciences, University of California, Los Angeles, PETER J. SCHMID, Department of Mathematics, Imperial College London — We will present results from direct numerical simulations of rapidly rotating convection in a fluid with $Pr \approx 0.025$ in cylindrical containers and Ekman numbers as low as 5×10^{-6} . In this system, the Coriolis force is the source of two types of inertial modes, the so-called wall modes, that also exist at moderate Prandtl numbers, and cylinder-filling oscillatory modes, that are a unique feature of small Prandtl number convection. The obtained flow fields were analyzed using the Dynamic Mode Decomposition (DMD). This technique allows to extract and identify the structures that govern the dynamics of the system as well as their corresponding frequencies. We have investigated both the regime where the flow is purely oscillatory and the regime where wall modes and oscillatory modes co-exist. In the purely oscillatory regime, high and low frequency oscillatory modes characterize the flow. When both types of modes are present, the DMD reveals that the wall-attached modes dominate the flow dynamics. They precess with a relatively low frequency in retrograde direction. Nonetheless, also in this case, high frequency oscillations have a significant contribution.

6:42PM E5.00006 The influence of the magnetic field on the heat transfer rate in rotating spherical shells.¹, ARES CABELLO, RUBEN AVILA, Facultad de Ingenieria. UNAM. Mexico — Studies of the relationship between natural convection and magnetic field generation in spherical annular geometries with rotation are essential to understand the internal dynamics of the terrestrial planets. In such studies it is important to calculate and analyze the heat transfer rate at the inner and the outer spheres that confine the spherical gap. Previous investigations indicate that the magnetic field has a stabilizing effect on the onset of the natural convection, reduces the intensity of convection and modifies the flow patterns. However so far it is still unclear how the magnetic field change the heat transfer rate behaviour. We investigate the heat transfer rate (Nu) in a rotating spherical gap with a self gravity field varying linearly with radius, and its relation with the intensity of the magnetic field induced by the geodynamo effect. 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 mesh. Several cases are simulated in which the Rayleigh number, the magnetic Reynolds number and the Taylor number are the variable parameters. The flow patterns, the temperature distribution and the Nusselt numbers at both spheres are calculated.

¹Special thanks to DGAPA-UNAM Project PAPIIT IN11731, sponsor of this investigation.

Sunday, November 20, 2016 5:37PM - 6:55PM – Session E6 Aerodynamics: Mathematical Models, Simulation and Shear Flow B114 - Ricardo Vinuesa, Royal Institute of Technology (KTH)

5:37PM E6.00001 High-order numerical simulations of the flow around wings at moderately high Reynolds number¹, RICARDO VINUESA, PRABAL NEGI, SEYED M. HOSSEINI, ARDESHIR HANIFI, DAN S. HENNINSON, PHILIPP SCHLATTER, Royal Institute of Technology (KTH) — The results of a DNS of the flow around a wing section represented by a NACA4412 profile, with $Re_c = 400,000$ and 5° angle of attack, are presented in this study. The high-order spectral-element code Nek5000 was used for the computations. The Clauser pressure-gradient parameter β ranges from $\simeq 0$ to 85 on the suction side, and the maximum Re_θ and Re_τ values are around 2,800 and 373, respectively. The adverse pressure gradient (APG) on the suction side of the wing leads to a progressively increasing value of the inner peak in the tangential velocity fluctuations, as well as the development of an outer peak, which is also observed in the other components of the Reynolds-stress tensor. Close to the trailing edge, i.e., at $x/c \simeq 0.9$, the outer peak in the inner-scaled tangential velocity profile is larger than the inner peak. These effects are connected to the fact that the large-scale motions of the flow become energized due to the APG, as apparent from spanwise-premultiplied power spectral density plots. Preliminary comparisons between DNS and well-resolved LES data, based on a relaxation-term filtering approach, are also presented with the aim of further extending the Reynolds number to $Re_c \simeq 1,000,000$.

¹Funded by the Swedish Research Council (VR) and the Knut and Alice Wallenberg Foundation

5:50PM E6.00002 An EnKF-based Flow State Estimator for Airfoils at High Angles of Attack, ANDRE FERNANDO DE CASTRO DA SILVA¹, TIM COLONIUS², California Institute of Technology — Robust flow estimation from available measurements remains a major obstacle to successful flow control applications. Although several estimation methodologies have been developed in the past decades, the high dimensionality of fluid systems renders many of them computationally intractable. In this work, we employ the Ensemble Kalman Filter (EnKF) and the two-dimensional incompressible Navier-Stokes equations to estimate the state of the flow past a NACA 0009 airfoil at high angles of attack and moderate Reynolds number. The pressure distribution on the airfoil and the velocity field in the wake, both randomized by synthetic noise, are sampled as measurement data. In order to evaluate the relative importance of each sensor location to the estimate correction, their influence fields (also known as representers) are analyzed. The performance of the estimator is then assessed for different choices of ensemble size, noise levels, and number/location of sensors.

¹Graduate Student

²Professor of Mechanical Engineering

6:03PM E6.00003 Mathematical Model for a Simplified Calculation of the Input Momentum Coefficient for AFC Purposes¹, DAMIAN HIRSCH, MORTEZA GHARIB, California Institute of Technology — Active Flow Control (AFC) is an emerging technology which aims at enhancing the aerodynamic performance of flight vehicles (i.e., to save fuel). A viable AFC system must consider the limited resources available on a plane for attaining performance goals. A higher performance goal (i.e., airplane incremental lift) demands a higher input fluidic requirement (i.e., mass flow rate). Therefore, the key requirement for a successful and practical design is to minimize power input while maximizing performance to achieve design targets. One of the most used design parameters is the input momentum coefficient C_μ . The difficulty associated with C_μ lies in obtaining the parameters for its calculation. In the literature two main approaches can be found, which both have their own disadvantages (assumptions, difficult measurements). A new, much simpler calculation approach will be presented that is based on a mathematical model that can be applied to most jet designs (i.e., steady or sweeping jets). The model-incorporated assumptions will be justified theoretically as well as experimentally. Furthermore, the models capabilities are exploited to give new insight to the AFC technology and its physical limitations.

¹Supported by Boeing

6:16PM E6.00004 Experimental Study of Thin NACA Symmetric and Cambered Airfoils at Low Reynolds Numbers, VIBHAV DURGESH, ELIFALET GARCIA, HAMID JOHARI, Cal State Univ - Northridge — The low-Reynolds number performance of airfoils is intriguing due to the complex fluid dynamics phenomena associated with flow at these Reynolds numbers, like laminar separated flow, increased transition susceptibility, and the separated shear layer that undergoes a rapid transition to a turbulent flow. Therefore, the objective of this investigation was to experimentally study the aerodynamic performance of a thin symmetric airfoil (NACA-0012) and a cambered (NACA-6412) airfoil at low Reynolds numbers, and to identify the flow structures responsible for altering the aerodynamic performance. Lift and drag force measurements were performed for both airfoils along with flow visualization measurements for Reynolds numbers of 20,000, 30,000, 40,000, and 50,000 and angles of attack between -8° to 15° with an increment of 1° . All the measurements for this study were performed in the water tunnel facility at California State University Northridge. A significant difference in the aerodynamic performance and flow behavior of the thin cambered airfoil is observed as compared to that of the thin symmetric airfoil. The presentation will discuss the correlation between observed flow structures and aerodynamic performance of both airfoils at low-Reynolds numbers.

6:29PM E6.00005 Lift on a Steady Airfoil in Low Reynolds Number Shear Flow¹, PATRICK HAMMER, Michigan State University, MIGUEL VISBAL, Air Force Research Laboratory, AHMED NAGUIB, MANOOCHEHR KOOCHESFAHANI, Michigan State University — Current understanding of airfoil aerodynamics is primarily based on a uniform freestream velocity approaching the airfoil, without consideration for possible presence of shear in the approach flow. Inviscid theory by Tsien (1943) shows that a symmetric airfoil at zero angle of attack experiences positive lift, i.e. a shift in the zero-lift angle of attack, in the presence of positive mean shear in the approach flow. In the current work, 2D computations are conducted on a steady NACA 0012 airfoil at a chord Reynolds number of $Re = 12,000$, at zero angle of attack. A uniform shear profile (i.e. a linear velocity variation) is used for the approach flow by modifying the FDL3DI Navier-Stokes solver (Visbal and Gaitonde, 1999). Interestingly, opposite to the inviscid prediction of Tsien (1943), the results for the airfoil at zero angle of attack show that the average lift is negative in the shear flow. The magnitude of this lift grows as the shear rate increases. Additional results are presented regarding the physics underlying the shear effect on lift. A companion experimental study is also given in a separate presentation.

¹This work was supported by AFOSR award number FA9550-15-1-0224.

6:42PM E6.00006 Experiments on a Steady Low Reynolds Number Airfoil in a Shear Flow¹, DAVID OLSON, AHMED NAGUIB, MANOOCHEHR KOOCHESFAHANI, Michigan State University — The aerodynamics of steady airfoils in uniform flow have received considerably more attention than that of an airfoil operating in a non-uniform flow. Inviscid theory by Tsien (1943) shows that an airfoil experiences a decrease in the zero lift angle of attack for a shear flow with uniform clockwise vorticity. The current work utilizes a shaped honeycomb technique to create a velocity profile with a large region of uniform shear in a water tunnel. Direct force measurements are implemented and validated using experiments on a circular cylinder and NACA 0012 in a uniform cross-flow. Results for a NACA 0012 airfoil with a chord Reynolds number of 1.2×10^4 in a non-uniform approach flow are compared to concurrent CFD calculations (presented in a companion talk) showing an increase in the zero lift angle of attack; in contradiction with inviscid theory. The effect of shear on the mean lift coefficient over a wide range of angles of attack is also explored.

¹This work was supported by AFOSR award number FA9550-15-1-0224.

Sunday, November 20, 2016 5:37PM - 6:55PM –
Session E7 Separation Flows in Aero and HydroDynamics B115 - Anya Jones, University of Maryland

5:37PM E7.00001 Adaptation of the Leishman-Beddoes Dynamic Stall Model for Reverse Flow, ANDREW LIND, ANYA JONES, University of Maryland — The Leishman-Beddoes dynamic stall model has long been used for the prediction of unsteady airloads acting on rotorcraft and wind turbines. However, little work has been completed that attempts to model the unsteady airloads experienced by a blade in the reverse flow region of a high advance ratio rotor. The present work describes modifications to the Leishman-Beddoes model and evaluates its suitability for the prediction of unsteady airloads for a sinusoidally oscillating NACA 0012 in reverse flow. Specifically, the ability of the model to capture early dynamic stall vortex formation (due to the sharp aerodynamic leading edge) and delayed reattachment is assessed. Results from the modified Leishman-Beddoes model are compared to measured unsteady pressure distributions for reduced frequencies up to 0.511 and a maximum pitch angle of 25 degrees. The model is also evaluated against numerical simulations of reverse flow dynamic stall where complete pressures distributions (and thus unsteady airloads) are available. This work is foundational for the development of more complex low-order models of the reverse flow region of a high advance ratio rotor where the time-varying local freestream and spanwise flow are also expected to play an important role.

5:50PM E7.00002 Experimental Investigation of a Yawed Airfoil in Reverse Flow

Dynamic Stall, LUKE SMITH, DR. ANDREW LIND, DR. ANYA JONES, University of Maryland, College Park — When a rotating blade enters high advance ratio flight, a significant portion of the blade is subject to reverse flow, where flow travels from the blade's geometric trailing edge to the geometric leading edge. The purpose of this work is to determine the influence of spanwise flow on a blade undergoing dynamic stall in reverse flow. Without spanwise flow, an oscillating sharp trailing edge airfoil in reverse flow experiences separation about its sharp aerodynamic leading edge, leading to the formation of a dynamic stall vortex at low angles of attack. With spanwise flow, an airfoil experiences a delay in lift stall, possibly due to the convection of a vortex along the freestream. This work characterizes the three-dimensional flow field of an oscillating airfoil at static yaw angles in reverse flow. Time-resolved velocity fields and chordwise pressure distributions are presented for several span locations, reduced frequencies, and Reynolds numbers. The unsteady velocity fields allow for the identification of dynamic stall vortex locations, and the unsteady pressure distributions allow for the analysis of spanwise variation in aerodynamic forces. By comparing the yawed and un-yawed cases, this work illustrates the relative importance of spanwise flow in reverse flow dynamic stall.

6:03PM E7.00003 Formation mechanisms of rapid pressure recovery around a laminar separation bubble¹

DONGHWI LEE, University of Tokyo, TAKU NOMONURA, AKIRA OYAMA, KOZO FUJII, Institute of Space and Astronautical Science, JAXA — Large-eddy simulations around 5% thickness flat plate are conducted at $Re_c = 5,000, 8,000$ and 20,000 and formation mechanisms of rapid pressure recovery in the surface pressure distribution around laminar separation bubbles are analyzed. Three analyses are applied to investigate the mechanisms of rapid pressure recovery. First, by using the Reynolds averaged streamwise pressure gradient equation, it is confirmed that the "overall Reynolds stress diffusion (ORSD)" is an important factor for inducing rapid pressure recovery. Second, we decompose the ORSD into the "normal Reynolds stress diffusion in the streamwise direction" and the "tangential Reynolds stress diffusion (TRSD) in the wall-normal direction". We show that the TRSD in the wall-normal direction, which corresponds to the momentum transfer in the same direction, is the main contributor to the ORSD. Third, the TRSD in the wall-normal direction is decomposed into two- and three-dimensional components. The results indicate that the rapid pressure recovery is strongly affected by the presence of Reynolds stress rather than by the type of physical phenomena that creates the Reynolds stress. In other words, the three-dimensional turbulent structures are not a necessary condition for the rapid pressure recovery.

¹Grain-in-Aid for JSPS Fellows

6:16PM E7.00004 Experimental Study of Unsteady Separation in a Laminar Boundary Layer¹

ANDREW BONACCI, AMY LANG, REDHA WAHIDI, LEO SANTOS, The University of Alabama — Separation, caused by an adverse pressure gradient, can be a major problem to aircraft. Reversing flow occurs in separated regions and an investigation of how this backflow forms is of interest due to the fact that this could be used as a means of initiating flow control. Specifically, backflow can bristle shark scales which may be linked to a passive, flow actuated separation control mechanism. An experiment was conducted in a water tunnel to replicate separation, with a focus on the reversing flow development near the wall within a laminar boundary layer. Using a rotating cylinder, an adverse pressure gradient was induced creating a separated region over a flat plate. In this experiment the boundary layer grows to sizes great enough that the scale of the flow is increased, making it more measurable to DPIV. In the future, this research can be utilized to better understand flow control mechanisms such as those enabled by shark skin.

¹Funding from Army Research Office and NSF REU site grant EEC 1358991 is greatly appreciated

6:29PM E7.00005 The actuation of microflaps inspired by shark scales deeply embedded in a boundary layer¹

JACKSON MORRIS, AMY LANG, PAUL HUBNER, Univ of Alabama - Tuscaloosa — Thanks to millions of years of natural selection, sharks have evolved to become quick apex predators. Shark skin is made up of microscopic scales on the order of 0.2 mm in size. This array of scales is hypothesized to be a flow control mechanism where individual scales are capable of being passively actuated by reversed flow in water due to their preferential orientation to attached flow. Previous research has proven shark skin to reduce flow separation in water, which would result in lower pressure drag. We believe shark scales are strategically sized to interact with the lower 5 percent of the boundary layer, where reversed flow occurs close to the wall. To test the capability of micro-flaps to be actuated in air various sets of flaps, inspired by shark scale geometry, were rapidly prototyped. These microflaps were tested in a low-speed wind tunnel at various flow speeds and boundary layer thicknesses. Boundary layer flow conditions were measured using a hot-wire probe and microflap actuation was observed. Microflap actuation in airflow would mean that this bio-inspired separation control mechanism found on shark skin has potential application for aircraft.

¹Boeing

6:42PM E7.00006 Reversing flow development in a separating turbulent boundary

layer¹, LEONARDO SANTOS, AMY LANG, REDHA WAHIDI, ANDREW BONACCI, The University of Alabama — Fast swimming sharks have micro-structures on their skin consisting of bristling scales. These scales are hypothesized to bristle in response to backflow generated from the separated turbulent boundary layer (TBL) in regions of adverse pressure gradient (APG) on the shark body. Vortices are trapped in the cavities between the scales, which induce momentum exchange between the higher momentum fluid in the outer flow and that in the separated region. This momentum exchange causes reattachment of the separated TBL, causing the scales to return to the unbristled location, and the cycle continues. The rows of scales have widths that are comparable to the spanwise length scale of the intermittent backflow patches that appear in the region of incipient detachment of TBLs. In this experimental investigation, correlations between the shark scale's width and the spanwise size of the low backflow streaks are examined, as well as details of the incipient detachment region. The experiments are conducted in a water tunnel facility and the flow field is measured using PIV. Turbulent boundary layers are subjected to an APG via a rotating cylinder. Separated TBLs are investigated on a flat plate.

¹The authors would like to greatly acknowledge the Army Research Office for funding this project.

Sunday, November 20, 2016 5:37PM - 6:55PM –
Session E8 Nonlinear Dynamics: Immersed Boundary, Wakes and Bounded Flows
B116 - Philipp Schlatter, KTH Royal Institute of Technology

5:37PM E8.00001 Nonlinear Dynamics of a Spring-Supported Piston in a Vibrated Liquid-Filled Housing: I. Analysis

J.R. TORCZYNSKI, T.J. O'HERN, J.R. CLAUSEN, Sandia National Laboratories — The nonlinear dynamics of a piston supported by a spring in a vibrated liquid-filled housing is analyzed. The liquid is viscous, and the flow passages are narrow and depend on piston position. Ordinarily, the piston motion is highly damped. However, if bellows are added to both ends of the housing, then the piston, liquid, and bellows can execute a collective motion that forces relatively little liquid through the flow passages and thus has low damping and a strong resonance. At this frequency, the motion is large, and the nonlinearity from the flow passages produces a net force on the piston that can cause it to compress its spring. This nonlinear dynamical system is analyzed using a perturbation expansion of the Navier-Stokes equations, and the perturbation results are compared to corresponding ALE Navier-Stokes simulations. 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.

5:50PM E8.00002 Nonlinear Dynamics of a Spring-Supported Piston in a Vibrated Liquid-Filled Housing: II. Experiments

T.J. O'HERN, J.R. TORCZYNSKI, J.R. CLAUSEN, Sandia National Laboratories — The nonlinear dynamics of a piston supported by a spring in a vibrated liquid-filled housing is investigated experimentally. The housing containing the piston and the liquid is subjected to vibrations along its axis. A post fixed to the housing penetrates a hole through the piston and produces a flow resistance that depends on piston position. Flexible bellows attached to the housing ends enable the piston, liquid, and bellows to execute a collective motion that forces little liquid through the flow resistance. The low damping of this motion leads to a resonance, at which the flow-resistance nonlinearity produces a net force on the piston that can cause it to compress its spring. Experiments are performed to investigate the nonlinear dynamics of this system, and these results are compared to theoretical and numerical results. 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.

6:03PM E8.00003 Computation of the deformation spectrum for flows on a sphere

SHAWN SHADDEN, UC Berkeley — The most common example of flow on a manifold is flow on a sphere with numerous geophysical examples. In this talk we consider the direct and accurate computation of the nonlinear deformation tensor describing fluid kinematics on manifold surfaces, and in particular efficient computation on spherical domains. We demonstrate that standard spherical coordinate computations are undesirable and instead integration of the singular deformation tensor in Cartesian coordinates restricted to the sphere can be advantageous. This approach yields a set of differential algebraic equations that can be reduced by symmetry to yield differential equations for a 2D flow. We have applied our method to the steady and unsteady flows generated by vortex sets on the sphere as well as geophysical flow models. For the former, we demonstrate that the evolution equations near the vortices can become singular and numerically unstable. To resolve this, we derived an exact solution for the spectral components of the deformation tensor near the vortices, which also enables us to match and validate our numerical solution.

6:16PM E8.00004 Vortical and modal network analysis of unsteady cylinder wake¹

MURALIKRISHNAN GOPALAKRISHNAN MEENA, KUNIHIKO TAIRA, Florida State Univ, STEVEN BRUNTON, University of Washington — Characterization of vortical and modal interactions among coherent structures in unsteady fluid flows is essential in understanding its complex behavior. Through a canonical example of incompressible flow over a circular cylinder at low Reynolds number, we quantify the interaction properties for both the vortical-wake and modal-interaction networks. For the vortical interactions, we represent the vortex elements as nodes and induced velocity between them as edge weights. With this formulation, we are able to capitalize upon network-theoretical toolsets to identify key vortical nodes and edges. Analogously, the modal-interaction network can be formulated using the modal decomposition bases and the coupling functions over the network. Based on this viewpoint, perturbations can then be tracked in terms of their amplitude and phase dynamics. We compare and contrast these network-based approaches to analyze unsteady fluid flows and discuss their implications in uncovering complex nonlinear dynamics and potential strategies towards flow field manipulation.

¹National Science Foundation (CBET 1632003)

6:29PM E8.00005 Nonlinear wavetrains in viscous conduits¹

MICHELLE MAIDEN, MARK HOEFER, University of Colorado Boulder — Viscous fluid conduits provide an ideal system for the study of dissipationless, dispersive hydrodynamics. A dense, viscous fluid serves as the background medium through which a lighter, less viscous fluid buoyantly rises. If the interior fluid is continuously injected, a deformable pipe forms. The long wave interfacial dynamics are well-described by a dispersive nonlinear partial differential equation. In this talk, experiments, numerics, and asymptotics of the viscous fluid conduit system will be presented. Structures at multiple length scales are discussed, including solitons, dispersive shock waves, and periodic waves. Modulations of periodic waves will be explored in the weakly nonlinear regime with the Nonlinear Schrödinger (NLS) equation. Modulational instability (stability) is identified for sufficiently short (long) periodic waves due to a change in dispersion curvature. These asymptotic results are confirmed by numerical simulations of perturbed nonlinear periodic wave solutions. Also, numerically observed are envelope bright and dark solitons well approximated by NLS.

¹This work was partially supported by NSF CAREER DMS-1255422 (M.A.H.) and NSF GRFP (M.D.M.).

6:42PM E8.00006 Observation of dispersive shock waves, solitons, and their interactions in viscous fluid conduits¹

DALTON ANDERSON, MICHELLE MAIDEN, University of Colorado Boulder, NICHOLAS LOWMAN, None, MARIKA SCHUBERT, MARK HOEFER, University of Colorado Boulder — Dispersive shock waves (DSWs) and solitons are fundamental structures in dispersive hydrodynamics, but studies have been severely constrained. Here we report on a novel testbed called the conduit system where one fluid is moved through another via a fluid pipe with virtually no mass diffusion. The interfacial dynamics of this pipe are conservative and are modeled by a scalar, nonlinear, dispersive wave equation, similar to those describing a superfluid. Resultantly, the interfacial waves are effectively dissipationless, which enables high fidelity observations of coherent phenomena such as large amplitude DSWs [1]. Experiments involving solitons, wavebreaking leading to DSWs, and their interactions will be presented. The results include the refraction and absorption of a soliton by a DSW and the refraction of a DSW by a second DSW, resulting in two-phase behavior. Excellent agreement between nonlinear wave averaging, numerics, and laboratory experiments will be presented. The nonlinear wave dynamics observed in this model system have implications for a broad range of other conservative dispersive hydrodynamic systems. Reference: [1] Maiden et al., PRL 116, 174501 (2016).

¹NSF

Sunday, November 20, 2016 5:37PM - 6:55PM –
Session E9 Fluid Dynamics - Education, Outreach, and Diversity II B117 - Frank Jacobitz,
University of San Diego

5:37PM E9.00001 Water Channel Facility for Fluid Dynamics Experiments , AZAR

ESLAM-PANAH, Penn State University, DANIEL SABATINO, Lafayette College — This study presents the design, assembly, and verification process of the circulating water channel constructed by undergraduate students at the Penn State University at Berks. This work was significantly inspired from the closed-loop free-surface water channel at Lafayette College (Sabatino and Maharjan, 2015) and employed for experiments in fluid dynamics. The channel has a 11 ft length, 2.5 ft width, and 2 ft height glass test section with a maximum velocity of 3.3 ft/s. First, the investigation justifies the needs of a water channel in an undergraduate institute and its potential applications in the whole field of engineering. Then, the design procedures applied to find the geometry and material of some elements of the channel, especially the contraction, the test section, the inlet and end tanks, and the pump system are described. The optimization of the contraction design, including the maintenance of uniform exit flow and avoidance of flow separation, is also included. Finally, the discussion concludes by identifying the problems with the undergraduate education through this capstone project and suggesting some new investigations to improve flow quality.

5:50PM E9.00002 A Geophysical Fluid Dynamics Lab Founded by Undergraduate Students¹ , SHIWEI SUN, Nanjing University, HAO FU, YUNJIAO PU, Nanjing University & Chinese Academy of Sciences, MINGRUI LIU, ZHIMING FENG, YILUN HAN, ANG ZHOU, JINGYI ZHUO, YUE HU, RUOYU WANG, NANA WU, ZIXUAN XIANG, JING XI, SALTANAT JAPPAR, JINGNAN YIN, CONGYUAN LI, JINJIE SONG, BOWEN ZHOU, YUAN WANG, Nanjing University — An atmospheric and oceanic fluid dynamics lab has been established by a group of undergraduate students in the School of Atmospheric Sciences at Nanjing University. A series of classical experiments have been conducted including Taylor column, topographic Rossby waves, and propagating density currents. With very limited funding, all instruments were designed and assembled by students. Their hands-on experimental abilities and understanding of the fundamental theories of geophysical fluid dynamics are greatly enhanced. The students work in groups on a dedicated experiment. A student project on rotating convection was even presented in APS DFD fall meeting last year. This year, we present some new laboratory demonstrative experiments of geophysical flow and introduce how they are incorporated in the undergraduate coursework.

¹Funding: National Science Talent Training Project (J1103410) and "LMSWE Lab Funding No.14380001"

6:03PM E9.00003 SFO-Project: The New Generation of Sharable, Editable and Open-Access CFD Tutorials , TEYMOUR JAVAHERCHI, ARDESHIR JAVAHERCHI, ALBERTO ALISEDA, University of Washington — One of the most common approaches to develop a Computational Fluid Dynamic (CFD) simulation for a new case study of interest is to search for the most similar, previously developed and validated CFD simulation among other works. A simple search would result into a pool of written/visual tutorials. However, users should spend significant amount of time and effort to find the most correct, compatible and valid tutorial in this pool and further modify it toward their simulation of interest. SFO is an open-source project with the core idea of saving the above-mentioned time and effort. This is done via documenting/sharing scientific and methodological approaches to develop CFD simulations for a wide spectrum of fundamental and industrial case studies in three different CFD solvers; STAR-CCM+, FLUENT and Open FOAM (SFO). All of the steps and required files of these tutorials are accessible and editable under the common roof of Github (a web-based Git repository hosting service). In this presentation we will present the current library of 20+ developed CFD tutorials, discuss the idea and benefit of using them, their educational values and explain how the next generation of open-access and live resource of CFD tutorials can be built further hand-in-hand within our community.

6:16PM E9.00004 Peer Learning in a MATLAB Programming Course , SHANON RECKINGER, Montana State University — Three forms of research-based peer learning were implemented in the design of a MATLAB programming course for mechanical engineering undergraduate students. First, a peer learning program was initiated. These undergraduate peer learning leaders played two roles in the course, (I) they were in the classroom helping students' with their work, and, (II) they led optional two hour helps sessions outside of the class time. The second form of peer learning was implemented through the inclusion of a peer discussion period following in class clicker quizzes. The third form of peer learning had the students creating video project assignments and posting them on YouTube to explain course topics to their peers. Several other more informal techniques were used to encourage peer learning. Student feedback in the form of both instructor-designed survey responses and formal course evaluations (quantitative and narrative) will be presented. Finally, effectiveness will be measured by formal assessment, direct and indirect to these peer learning methods. This will include both academic data/grades and pre/post test scores. Overall, the course design and its inclusion of these peer learning techniques demonstrate effectiveness.

6:29PM E9.00005 Motivating students to read the textbook before class , RACHEL E. PEPPER, University of Puget Sound — Many faculty in STEM courses assign textbook reading in advance of lecture, yet evidence shows few students actually read the textbook. Those students that do read often do so only after the material has been presented in class. Preparing for class by reading the textbook beforehand improves student learning and is particularly critical for classes that employ active engagement strategies. Here I present strategies I have used to successfully motivate my students to read the textbook before class in physics classes ranging from introductory algebra-based physics to advanced courses for physics majors. In the introductory course, I used pre-class reading quizzes, a common strategy that has been shown effective in previous studies, but one that is somewhat time-consuming to implement. In my more advanced courses I used reading reflections, which required considerably less time. While it was typical for less than 25% of students to read the textbook before I implemented reading quizzes or reflections, after implementing these strategies 70-90% of students reported reading the textbook before class most of the time. Students also report finding both the readings themselves and the quizzes and reflections valuable for their learning.

6:42PM E9.00006 An oral exam model for teaching advanced "Batchelor-level" fluid mechanics in the US. , JONATHAN FREUND, University of Illinois at Urbana-Champaign — A teaching model is developed to meet the challenge of teaching fluid mechanics at what might be considered a high level, at least by the current norms in the US. The initial goal was to avoid loss of concepts amidst the challenge of particular mathematical manipulations on particular assignments. However, it evolved toward fostering facile working knowledge of challenging material, such as in the books by Batchelor (e.g. streaming flow), Whitham (e.g. ship waves), and van Dyke (e.g. second-order boundary layer). To this end, the course model forgoes traditional assigned problems to focus on completion, augmentation, and in-depth understanding of the lecture material. The lectures are relatively traditional in structure, albeit with somewhat more interactive examples. The main unusual feature—again, by modern US standards—was assessment via multiple half-hour oral exams. This model has now been successful over 8 semesters for 3 different graduate courses in 2 departments. For all, students were assumed to have already completed a full course at a "Navier-Stokes level". The presentation will include specifics of the course and exam structure, impressions of positive outcomes from the instructor, and a summary of the overwhelmingly positive student feedback.

**Sunday, November 20, 2016 5:37PM - 6:55PM –
Session E10 Jets, Waves, and Markovianity B118-119 - Steven Brunton Eurika Kaiser, University of Washington**

5:37PM E10.00001 Thin film deposition using rarefied gas jet , DR. SAHADEV PRADHAN,

Department of Chemical Engineering, Indian Institute of Science, Bangalore- 560 012, India — The rarefied gas jet of aluminium is studied at Mach number $Ma = (U_j / \sqrt{kb T_j / m})$ in the range $.01 < Ma < 2$, and Knudsen number $Kn = (1 / (\sqrt{2} \pi d^2 n_d H))$ in the range $.01 < Kn < 15$, using two-dimensional (2D) direct simulation Monte Carlo (DSMC) simulations, to understand the flow phenomena and deposition mechanisms in a physical vapor deposition (PVD) process for the development of the highly oriented pure metallic aluminum thin film with uniform thickness and strong adhesion on the surface of the substrate in the form of ionic plasma, so that the substrate can be protected from corrosion and oxidation and thereby enhance the lifetime and safety, and to introduce the desired surface properties for a given application. Here, H is the characteristic dimension, U_j and T_j are the jet velocity and temperature, n_d is the number density of the jet, m and d are the molecular mass and diameter, and kb is the Boltzmann constant. An important finding is that the capture width (cross-section of the gas jet deposited on the substrate) is symmetric around the centerline of the substrate, and decreases with increased Mach number due to an increase in the momentum of the gas molecules. DSMC simulation results reveals that at low Knudsen number ($(Kn = 0.01)$;shorter mean free paths), the atoms experience more collisions, which direct them toward the substrate. However, the atoms also move with lower momentum at low Mach number,which allows scattering collisions to rapidly direct the atoms to the substrate.

5:50PM E10.00002 Turbulence statistics in a negatively buoyant particle plume – laboratory measurement , ANKUR BORDOLOI, Los Alamos National Laboratory, LAURA CLARK, GERARDO VELIZ, MICHAEL HEATH, EVAN VARIANO, University of California Berkeley —

Negatively buoyant plumes of nylon particles are investigated in quiescent salt-water solution using flow visualization and stereoscopic PIV. Particles of the size 2 mm are continuously released through a nozzle from the top inside a water tank using a screw-conveyor based release mechanism. The plume propagates downward due to gravity, and by virtue of interacting particle wakes, becomes turbulent. The two phases are refractive index matched, so that the velocity field in the interstitial fluid can be quantified using PIV. We examine the velocity fields in the fluid phase to characterize turbulence statistics, such as turbulent kinetic energy, Reynolds stresses in the fully developed region of the plume. Further, we develop an image processing method to obtain particle distribution and particle slip inside the plume. In the presentation, we will discuss these results in the light of existing literature for rising plumes of bubbles under similar experimental conditions.

6:03PM E10.00003 The interaction between atmospheric gravity waves and large-scale flows: an efficient description beyond the non-acceleration paradigm , BRUNO RIBSTEIN,

CMLA, ENS, F-94230, Cachan, France and LMD, ENS, F-75005, Paris, France, Gergely Blni, JEWGENIJA MURASCHKO, CHRISTINE SGOFF, JUNHONG WEI, ULRICH ACHATZ, Goethe University Frankfurt, Frankfurt am Main, Germany — With the aim of contributing to the improvement of subgrid-scale gravity wave (GW) parameterizations in numerical-weather-prediction and climate models, the comparative relevance in GW drag of direct GW-mean-flow interactions and turbulent wave breakdown are investigated. Of equal interest is how well Wentzel-Kramer-Brillouin (WKB) theory can capture direct wave-mean-flow interactions, that are excluded by applying the steady-state approximation. WKB is implemented in a very efficient Lagrangian ray-tracing approach that considers wave action density in phasespace, thereby avoiding numerical instabilities due to caustics. It is supplemented by a simple wave-breaking scheme based on a static-instability saturation criterion. Idealized test cases of horizontally homogeneous GW packets are considered where wave-resolving Large-Eddy Simulations (LES) provide the reference. In all of theses cases the WKB simulations including direct GW-mean-flow interactions reproduce the LES data, to a good accuracy, already without wave-breaking scheme. The latter provides a next-order correction that is useful for fully capturing the total-energy balance between wave and mean flow. This is not the case when a steady-state WKB implementation is used, as used in present GW parameterizations.

6:16PM E10.00004 Non-Markov effects in intersecting sprays , MAHESH PANCHAGNULA,

DHIVYARAJA KUMARAN, SRI VALLABHA DEEVI, ARUN TANGIRALA, Indian Inst of Tech-Madras — Sprays have been assumed to follow a Markov process. In this study, we revisit that assumption relying on experimental data from intersecting and non-intersecting sprays. A phase Doppler Particle Analyzer (PDPA) is used to measure particle diameter and velocity at various axial locations in the intersection region of two sprays. Measurements of single sprays, with one nozzle turned off alternatively are also obtained at the same locations. This data, treated as an unstructured time series is classified into three bins each for diameter (small, medium, large) and velocity (slow, medium, fast). Conditional probability analysis on this binned data showed a higher static correlation between droplet velocities, while diameter correlation is significantly alleviated (reduced) in intersecting sprays, compared to single sprays. Further analysis using serial correlation measures: auto-correlation function (ACF) and partial auto-correlation function (PACF) shows that the lagged correlations in droplet velocity are enhanced while those in the droplet diameter are significantly debilitated in intersecting sprays. We show that sprays are not necessarily Markov processes and that memory persists, even though curtailed to fewer lags in case of size, and enhanced in case of droplet velocity.

6:29PM E10.00005 Towards a thorough use of the Mori-Zwanzig formalism for statistical coarse-graining of turbulent flows¹ , AYOUB GOUASMI, ERIC PARISH, KARTHIK DURAISAMY, Univ of Michigan - Ann Arbor, COMPUTATIONAL AEROSCIENCES LABORATORY TEAM —

The Mori-Zwanzig formalism provides a mathematically-consistent framework to represent the unresolved physics in coarse-grained simulations. The closure terms that arise can be formally represented as memory or non-Markovian effects. However, the general procedure to compute these memory effects is not tractable in fluid flow problems. Accordingly, existing Mori-Zwanzig closure models only scratch the surface of the framework: they simplify the memory effect by making assumptions that cannot be numerically assessed. We propose a more tractable procedure to approximate memory effects with a good level of accuracy. This approach is demonstrated on the Viscous Burgers Equation and the Kuramoto-Sivashinsky Equation. Building on these results, we provide perspectives in the development of Mori-Zwanzig-based coarse-grained models for turbulent flows.

¹This work was supported in part by AFOSR under the project "LES Modeling of Non-local effects using Statistical Coarse-graining" with Dr. Jean-Luc Cambier as the technical monitor

6:42PM E10.00006 A probabilistic approach to modeling and controlling fluid flows , EURIKA KAISER, University of Washington, BERND R. NOACK, LIMSI-CNRS, France, ANDREAS SPOHN, ENSMA, France, LOUIS N. CATTAFESTA, Florida State University, MAREK MORZYNSKI, Poznan University of Technology, Poland, GUILLAUME DAVILLER, CERFACS, France, BINGNI W. BRUNTON, STEVEN L. BRUNTON, University of Washington —

We extend cluster-based reduced-order modeling (CROM) (Kaiser et al, 2014) to include control inputs in order to determine optimal control laws with respect to a cost function for unsteady flows. The proposed methodology frames high-dimensional, nonlinear dynamics into low- dimensional, probabilistic, linear dynamics which considerably simplifies the optimal control problem while preserving nonlinear actuation mechanisms. The data-driven approach builds upon the unsupervised partitioning of the data into few kinematically similar flow states using a clustering algorithm. The coarse-grained dynamics are then described by a Markov model which is closely related to the approximation of Perron-Frobenius operators. The Markov model can be used as predictor for the ergodic probability distribution for a particular control law approximating the long-term behavior of the system on which basis the optimal control law is determined. Moreover, we combine CROM with a recently developed approach for optimal sparse sensor placement for classification (Brunton et al., 2013) as a critical enabler for in-time control and for the systematic identification of dynamical regimes from few measurements. The approach is applied to a separating flow and a mixing layer exhibiting vortex pairing.

Sunday, November 20, 2016 5:37PM - 6:55PM –

Session E11 Electrokinetic Flows: Computations C120-121-122 - Carlos Hidrovo, Northeastern University

5:37PM E11.00001 A numerical study on liquid charging inside electrostatic atomizers¹, BABAK KASHIR, ANTHONY PERRI, ABHILASH SANKARAN, CHRISTOPHER STASZEL, ALEXANDER YARIN, FARZAD MASHAYEK, Univ of Illinois - Chicago — The charging of the dielectric liquid inside an electrostatic atomizer is studied numerically by developing codes based on the OpenFOAM platform. Electrostatic atomization is an appealing technology in painting, fuel injection and oil coating systems due to improved particle-size distribution, enhanced controllability of droplets' trajectories and lower power consumption. The numerical study is conducted concurrently to an experimental investigation to facilitate the validation and deliver feedback for further development. The atomizer includes a pin electrode that is placed at the center of a converging chamber. The chamber orifice is located at a known distance from the electrode tip. The pin electrode is connected to a high voltage that leads to the charging of the liquid. In the present work, the theoretical foundations of separated treatment of the polarized layer and the electroneutral bulk flow are set by describing the governing equations, relevant boundary conditions and the matching condition between these two domains. The resulting split domains are solved numerically to find the distribution of velocity and electrostatic fields over the specified regions.

¹National Science Foundation Award number: 1505276

5:50PM E11.00002 Electrically Controllable Microparticle Synthesis and Digital Microfluidic Manipulation by Electric-Field-Induced Droplet Dispensing into Immiscible Fluids¹, TAEWOONG UM, JIWOO HONG, IN SEOK KANG, POSTECH — The dispensing of tiny droplets is a basic and crucial process in a myriad of applications, such as DNA/protein microarray, cell cultures, chemical synthesis of microparticles, and digital microfluidics. This work demonstrates the droplet dispensing into immiscible fluids through electric charge concentration (ECC) method. Three main modes (i.e., attaching, uniform and bursting modes) are exhibited as a function of flow rates, applied voltage and gap distance between the nozzle and the oil surface. Through a conventional nozzle with diameter of a few millimeters, charged droplets with volumes ranging from a few μL to a few tens of nL can be uniformly dispensed into the oil chamber without reduction in nozzle size. Based on the features of the proposed method (e.g., formation of droplets with controllable polarity and amount of electric charge in water and oil system), a simple and straightforward method is developed for microparticle synthesis, including preparation for colloidosomes and fabrication of Janus microparticles with anisotropic internal structures. Finally, a combined system consisting of ECC-induced droplet dispensing and electrophoresis of charged droplet (ECD)-driven manipulation systems is constructed.

¹This work was supported by the BK21Plus Program for advanced education of creative chemical engineers of the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP).

6:03PM E11.00003 Capacitive Deionization: a coupled 2D electro-adsorption/convective-diffusive simulation for various system configurations, CARLOS HIDROVO, YASAMIN SALAMAT, Northeastern University — Capacitive Deionization (CDI) is a relatively new electrically based desalination method that uses porous media to adsorb ions in solution from water, with the potential to recover part of the energy used during the desalination process. Previous studies have investigated the physics underlying the electro-adsorption process in the electrical double layers in the CDI porous electrodes. In order to improve CDI performance in terms of minimum average concentration, total amount of water treated, and duration of the desalination process, herein we propose and evaluate different CDI architectures. Two previously validated 2D and 1D models are used alongside each other to study different CDI system configurations based on various convective-diffusive layer regimes. Moreover, the effects of micro pore and macro pore capacities on the total number of ions adsorbed in the porous media is investigated. This will open new opportunities for further researches toward engineered CDI units for better desalination.

6:16PM E11.00004 Accounting for Finite Size of Ions in Nanofluidic Channels Using Density Functional Theory¹, CHRISTOPHER MCCALLUM, Univ of California - Santa Barbara , DIRK GILLESPIE, Rush Medical College, Rush University Medical Center , SUMITA PENNATHUR, University of California - Santa Barbara — The physics of nanofluidic devices are dominated by ion-wall interactions within the electric double layer (EDL). A full understanding of the EDL allows for better exploitation of micro and nanofluidic devices for applications such as biologic separations, desalination, and energy conversion. Although continuum theory is generally used to study the fluidics within these channels, in very confined geometries, high surface charge channels, or significant solute concentration systems, continuum theories such as Poisson-Boltzmann cease to be valid because the finite size of ions is not considered. Density functional theory (DFT) provides an accurate and efficient method for predicting the concentration of ions and the electrostatic potential near a charged wall because it accounts for more complex electrostatic and hard-sphere correlations. This subsequently allows for a better model for ion flux, fluid flow, and current in electrokinetic systems at high surface charge, confined geometries, and concentrated systems. In this work, we present a theoretical approach utilizing DFT to predict unique flow phenomena in nanofluidic, electrokinetic systems.

¹ CBET-1402736 from the National Science Foundation

6:29PM E11.00005 Impact of pore size variability and network coupling on electrokinetic transport in porous media, SHIMA ALIZADEH, Stanford University, MARTIN Z. BAZANT, Massachusetts Institute of Technology, ALI MANI, Stanford University — We have developed and validated an efficient and robust computational model to study the coupled fluid and ion transport through electrokinetic porous media, which are exposed to external gradients of pressure, electric potential, and concentration. In our approach a porous media is modeled as a network of many pores through which the transport is described by the coupled Poisson-Nernst-Planck-Stokes equations. When the pore sizes are random, the interactions between various modes of transport may provoke complexities such as concentration polarization shocks and internal flow circulations. These phenomena impact mixing and transport in various systems including deionization and filtration systems, supercapacitors, and lab-on-a-chip devices. In this work, we present simulations of massive networks of pores and we demonstrate the impact of pore size variation, and pore-pore coupling on the overall electrokinetic transport in porous media.

6:42PM E11.00006 Simulation of 3D Chaotic Electroconvection in Shear Flow¹, SCOTT DAVIDSON, ALI MANI, Stanford University — Electroconvection, a microscale electrohydrodynamic phenomenon with chaotic features reminiscent of turbulence, provides the dominant transport mechanism in many electrochemical processes where ions are driven through ion-selective surfaces under large applied voltages. Electrodialysis, for example, desalinates water by flowing it between layers of ion-selective membranes with alternating selectivity while an electric field is applied normal to the membranes. This process leads to alternating channels becoming enriched and depleted of ions. Despite its key importance, much about how electroconvection enhances ion transport, particularly in the presence of crossflow, remains a mystery. We present results of 3D direct numerical simulations of electroconvection in a canonical geometry of an electrolyte between an ion-selective membrane and a reservoir with periodic sides subject to applied shear flow. We analyze the effects of crossflow on both flow statistics and qualitative structures in the fully chaotic regime.

¹Stanford Graduate Fellowship, NSF GRFP

Sunday, November 20, 2016 5:37PM - 6:55PM – Session E12 Multiphase Flows: Mixing C123 - Cheng Li, Johns Hopkins University

5:37PM E12.00001 Subsurface Droplet Size Distribution generated as breaking waves entrain an oil slick¹, CHENG LI, JESSE MILLER, JOSEPH KATZ, Johns Hopkins University — Breaking waves are a primary mechanism for entraining and dispersing oil spills. Knowledge of the resulting droplet size distribution is crucial for predicting the transport and fate of this oil. In this on-going experimental study, a controlled oil slick of varying viscosity (μ_d), density (ρ_d), interfacial tension (σ), and thickness $\delta = 0.5\text{mm}$ are entrained by waves of varying energy (E_w). The changes to droplet size over time, from seconds to hours, are measured at several locations using multi-resolution holography, which covers sizes ranging from μm to mm. Using dispersants to reduce σ , the Webber number, $We = E_w \delta / \sigma$, and Ohnesorge number, $Oh = \mu_d / (\rho_d \delta \sigma)^{0.5}$, are varied from 6 to 813 and from 0.09 to 0.95, respectively. Droplets smaller than the turbulence scale (2-30 μm – diameter), are generated by “micro-threading”. Their size distribution becomes steeper and their total number increase substantially with decreasing interfacial tension. For slopes smaller than -3, measured for σ around 10^{-1} mN/m , the volumetric size distribution decreases with diameter, i.e. most of the oil breaks into micron-scale droplets. For high interfacial tension oil, the concentration of small droplets increases with wave energy, but this effect diminishes as σ decreases. Droplets larger than 100 μm are generated by turbulent shear. Hence, their number is impacted by μ_d and E_w . Increasing We from 6 to 15 (Oh from 0.09 to 2.95) increases the initial number of droplets by up to 5 times, but the distribution slopes remain largely similar.

¹Supported by Gulf of Mexico Research Initiative (GoMRI)

5:50PM E12.00002 Laser imaging in liquid-liquid flows¹, M.I.I. ZAINAL ABIDIN, KYEONG H. PARK, VICTOR VOULGAROPOULOS, MAXIME CHINAUD, PANAGIOTA ANGELI, Department of Chemical Engineering, University College London, Torrington Place, London WC1E 7JE, UK — In this work, the flow patterns formed during the horizontal flow of two immiscible liquids are studied. The pipe is made from acrylic, has an ID of 26 mm and a length of 4 m. A silicone oil (5cSt) and a water/glycerol mixture are used as test fluids. This set of liquids is chosen to match the refractive indices of the phases and enable laser based flow pattern identification. A double pulsed Nd:Yag laser was employed (532nm) with the appropriate optics to generate a laser sheet at the middle of the pipe. The aqueous phase was dyed with Rhodamine 6G, to distinguish between the two phases. Experiments were carried out for mixture velocities ranging from 0.15 to 2 m/s. Different inlet designs were used to actuate flow patterns in a controlled way and observe their development downstream the test section. A static mixer produced dispersed flow at the inlet which separated downstream due to enhanced coalescence. On the other hand, the use of a cylindrical bluff body at the inlet created non-linear interfacial waves in initially stratified flows from which drops detached leading to the transition to dispersed patterns. From the detailed images important flow parameters were measured such as wave characteristics and drop size.

¹Project funded under the UK Engineering and Physical Sciences Research Council (EPSRC) Programme Grant MEMPHIS

6:03PM E12.00003 Flow separation characteristics of unstable dispersions¹, VICTOR VOULGAROPOULOS, Department of Chemical Engineering, University College London, Torrington Place, London WC1E 7JE, UK, LUSHENG ZHAI, School of Electrical Engineering and Automation, Tianjin University, Tianjin 300072, Peoples Republic of China, PANAGIOTA ANGELI, Department of Chemical Engineering, University College London, Torrington Place, London WC1E 7JE, UK — Drops of a low viscosity oil are introduced through a multi-capillary inlet during the flow of water in a horizontal pipe. The flow rates of the continuous water phase are kept in the turbulent region while the droplets are injected at similar flow rates (with oil fractions ranging from 0.15 to 0.60). The acrylic pipe (ID of 37mm) is approximately 7m long. Measurements are conducted at three different axial locations to illustrate how the flow structures are formed and develop along the pipe. Initial observations are made on the flow patterns through high-speed imaging. Stratification is observed for the flow rates studied, indicating that the turbulent dispersive forces are lower than the gravity ones. These results are complemented with a tomography system acquiring measurements at the same locations and giving the cross-sectional hold-up. The coalescence dynamics are strong in the dense-packed drop layer and thus measurements with a dual-conductance probe are conducted to capture any drop size changes. It is found that the drop size variations depend on the spatial configuration of the drops, the initial drop size along with the continuous and dispersed phase velocities.

¹Project funded under Chevron Energy Technology

6:16PM E12.00004 Jet-mixing of initially-stratified liquid-liquid pipe flows: experiments and numerical simulations¹, STUART WRIGHT, ROBERTO IBARRA-HERNANDES, ZHIHUA XIE, CHRISTOS MARKIDES, OMAR MATAR, Imperial College London — Low pipeline velocities lead to stratification and so-called phase slip in horizontal liquid-liquid flows due to differences in liquid densities and viscosities. Stratified flows have no suitable single point for sampling, from which average phase properties (e.g. fractions) can be established. Inline mixing, achieved by static mixers or jets in cross-flow (JICF), is often used to overcome liquid-liquid stratification by establishing unstable two-phase dispersions for sampling. Achieving dispersions in liquid-liquid pipeline flows using JICF is the subject of this experimental and modelling work. The experimental facility involves a matched refractive index liquid-liquid-solid system, featuring an ETFE test section, and experimental liquids which are silicone oil and a 51-wt% glycerol solution. The matching then allows the dispersed fluid phase fractions and velocity fields to be established through advanced optical techniques, namely PLIF (for phase) and PTV or PIV (for velocity fields). CFD codes using the volume of a fluid (VOF) method are then used to demonstrate JICF breakup and dispersion in stratified pipeline flows. A number of simple jet configurations are described and their dispersion effectiveness is compared with the experimental results.

¹Funding from Cameron for Ph.D. studentship (SW) gratefully acknowledged

6:29PM E12.00005 Experiments and simulations of oil-water flows in horizontal pipes¹, ROBERTO IBARRA-HERNANDES, STUART WRIGHT, ZHIHUA XIE, CHRISTOS MARKIDES, OMAR MATAR, Imperial College London — The extraction of detailed information (e.g. velocity and turbulent data) in the flow of two immiscible liquid phases in horizontal pipes is of great importance for the fundamental understanding of the in situ hydrodynamics (and transport properties) of these flows, and the validation and improvement of advanced multiphase flow models. This detailed flow information can be obtained by the application of advanced laser-based diagnostic techniques, such as Planar Laser-Induced Fluorescence (PLIF) and Particle Image Velocimetry (PIV), however, the difference in the refractive index between the most relevant test fluids (oil, water) prevents the extraction of accurate information simultaneously in both phases, especially when the phases begin to develop interfacial instabilities, droplets and dispersions. In this work, a simultaneous, combined two-line technique is employed to obtain spatiotemporally resolved information in a 32 mm ID quartz pipe in terms of liquid phase distributions, velocity profiles and turbulence measurements. The experimental results are compared with numerical simulations carried out using the Fluidity code based on control-volume, finite-elements, and adaptive, unstructured meshing.

¹Funding from BP (for R-IH), Cameron (for SW), and the EPSRC UK through the MEMPHIS programme (Grant number EP/K0039761/1) is gratefully acknowledged.

6:42PM E12.00006 Proper Orthogonal Decomposition on Experimental Multi-phase Flow in a Pipe, BIANCA VIGLIANO, Portland State University, MURAT TUTKUN, Institute for Energy Technology, Kjeller Norway, RAL BAYON CAL, Portland State University — Multi-phase flow in a 10 cm diameter pipe is analyzed using proper orthogonal decomposition. The data were obtained using X-ray computed tomography in the Well Flow Loop at the Institute for Energy Technology in Kjeller, Norway. The system consists of two sources and two detectors; one camera records the vertical beams and one camera records the horizontal beams. The X-ray system allows measurement of phase holdup, cross-sectional phase distributions and gas-liquid interface characteristics within the pipe. The mathematical framework in the context of multi-phase flows is developed. Phase fractions of a two-phase (gas-liquid) flow are analyzed and a reduced order description of the flow is generated. Experimental data deepens the complexity of the analysis with limited known quantities for reconstruction. Comparison between the reconstructed fields and the full data set allows observation of the important features. The mathematical description obtained from the decomposition will deepen the understanding of multi-phase flow characteristics and is applicable to fluidized beds, hydroelectric power and nuclear processes to name a few.

Sunday, November 20, 2016 5:37PM - 6:55PM – Session E13 DFD GPC: Geophysical Fluid Dynamics: Bluff Bodies in Stratified Flows
C124 - Jean-Marc Chomaz, CNRS-Ecole Polytechnique

5:37PM E13.00001 Fluttering in Stratified Flows, TRY LAM, LIONEL VINCENT, EVA KANSO, Univ of Southern California — The descent motion of heavy objects under the influence of gravitational and aerodynamic forces is relevant to many branches of engineering and science. Examples range from estimating the behavior of re-entry space vehicles to studying the settlement of marine larvae and its influence on underwater ecology. The behavior of regularly shaped objects freely falling in homogeneous fluids is relatively well understood. For example, the complex interaction of a rigid coin with the surrounding fluid will cause it to either fall steadily, flutter, tumble, or be chaotic. Less is known about the effect of density stratification on the descent behavior. Here, we experimentally investigate the descent of discs in both pure water and in a linearly salt-stratified fluids where the density is varied from 1.0 to 1.14 of that of water where the Brunt-Vaisala frequency is 1.7 rad/sec and the Froude number $Fr < 1$. We found that stratification enhances the radial dispersion of the disc at landing, and simultaneously, decrease the descent speed and the inclination (or nutation) angle while falling. We conclude by commenting on the relevance of these results to the use of unpowered vehicles and robots for space exploration and underwater missions.

5:50PM E13.00002 Internal Gravity Wave Fluxes Radiated by a Stably Stratified Turbulent Wake¹, KRISTOPHER ROWE, PETER DIAMESSIS, Cornell University — The study of the turbulent wake generated by a bluff body moving through a stably stratified fluid has important applications for naval hydrodynamics as well as geophysical flows around topography. Significant progress has been made in terms of investigating the structure and dynamics of the turbulent wake core and the associated near and far-field spectral properties of the wake-radiated internal gravity wave (IGW) fields, namely in the context of high Reynolds stratified turbulence within the wake itself. Nevertheless, little has been done to quantify the amount of energy and momentum radiated away by the IGWs generated by the wake. Through analysis of a broad Large Eddy Simulation dataset, spanning values of body-based Reynolds and Froude numbers, $Re = 5 \times 10^3, 10^5$ and 4×10^5 and $Fr = 4, 16$ and 64 , we compute the energy and momentum fluxes of IGWs radiated by the stratified turbulent wake of a towed sphere and explore the relevant parametric dependence. The analysis further aims to determine the potential of the IGWs as a sink for energy and momentum relative to the dissipation of turbulent kinetic energy in the wake itself. Finally, we discuss the implications that for our findings for wake mean-flow self-similarity and turbulence subgrid scale models.

¹Office of Naval Research grants N00014-13-1-0665 and N00014-15-1-2513

6:03PM E13.00003 Near wake characteristics in a stably-stratified fluid¹, TRYSTAN MADISON, XINJIANG XIANG, PRABU SELLAPPAN, GEOFFREY SPEDDING, University of Southern California — Decaying stratified turbulence that is free to evolve in the presence of a stable density gradient eventually reaches a state dominated by low Froude number dynamics where persistent patterns emerge. Whether or not information from the initial turbulence creator persists in the formation of these patterns is still an open question. For example the late time evolution of bluff body wakes have been shown to have universal characteristics that are independent of the details of the original generator while experiments on the near wake of a towed grid suggest that the earliest stages of flow development do depend on the initial conditions. Here we present near wake characteristics of two grids with varied mesh spacing and similar solidity, a disk, and a sphere, all of equivalent drag, for $Re = \{1000, 3000\}$ and $Fr = \{1, 4\}$. Quantitative measures in the wake signature deriving from whole-field PIV measurements will be used to specify when and how near wakes are similar, or different from each other, and from expectations or suppositions in the literature.

¹Support from ONR grant N00014-15-1-2506 is gratefully acknowledged

6:16PM E13.00004 Experiments and simulations of low Re sphere wakes with and without stratification¹, XINJIANG XIANG, KEVIN CHEN, TRYSTAN MADISON, GEOFFREY SPEDDING, University of Southern California — Bluff body wakes in both stratified and unstratified background have been studied extensively due to their geophysical and naval applications. A global map showing the dependence of near-wake structures behind a towed sphere on initial Reynolds number and Froude number, has been provided by Lin et al. (*J. Fluid Mech.*, 240, 315-354, 1992) and Chomaz et al. (*J. Fluid Mech.*, 254, 1-21, 1993). Here full-field measurements of the sphere wakes in both homogeneous and linearly stratified ambient are provided by simulations and experiments, at Reynolds number $Re \leq 1000$ and Froude number $Fr \geq 0.5$. In a homogeneous fluid, the structural transitions with increasing Re are consistent with previous studies. The stratified results from simulations and experiments are in good agreement. Stratified wakes undergo similar transitions with decreasing Fr for Re in current range, except that similar transitions occur at larger Re as Fr decreases, thus making the wake structure at high Re , low Fr similar to that at low Re , high Fr .

¹Support from ONR N00014-15-1-2506 under the management of Dr. R. Joslin is most gratefully acknowledged. The simulations were performed at the High Performance Computing Center of University of Southern California.

6:29PM E13.00005 The sensitivity of stratified flow stability to base flow modifications¹, KEVIN CHEN, GEOFFREY SPEDDING, University of Southern California — We present a novel theory that determines the sensitivity of linear stability to changes in the density or velocity of a base flow. The sensitivity is based on global direct and adjoint eigenmodes of the linearized Boussinesq equation, and is inspired by constant-density sensitivity analysis. The theory can be applied broadly to incompressible flows with small density variations, but it specifically provides new insight into the stability of density-stratified flows. Examples are given for the flows around a transverse thin plate at a Reynolds number of 30, a Prandtl number of 7.19, and Froude numbers of ∞ and 1. In the unstratified flow, the sensitivity is largest in the recirculation bubble; the stratified flow, however, exhibits high sensitivity in regions immediately upstream and downstream of the bluff body.

¹Supported by the Viterbi Postdoctoral Fellowship, provided by the Viterbi School of Engineering at the University of Southern California.

6:42PM E13.00006 Stabilization of triadic resonance of a finite amplitude gravity wave in the ocean : when a daughter wave is engaged with two fiancés¹, JEAN-MARC CHOMAZ, LadHyX, CNRS-Ecole Polytechnique, SABINE ORTIZ, LadHyX-UME, ENSTA-Paristech, GAÉTAN LERISSON, LadHyX, CNRS-Ecole Polytechnique — Triadic instability is a very generic mechanism by which a primary wave of finite amplitude is destabilized by two secondary (daughter) waves forming a resonant triad. For gravity wave in the ocean, as shown by Phillips, O.M. (CUP, 1967) the resonant triads form several continuous family that may be represented in twodimension (2D) as resonant lines in the 2D wave vector space of the secondary wave. We show here that the crossing of two od these branches may results in a double triadic instability where the instability is reduced. Building on McEwan, A.D. & Plum, R.A. (Dyn. Atm. & Oceans, 1977) we show that this double triadic instability stabilization domain expends from a singular point to a finite significant region when the amplitude of the primary wave is increased. Comparison with direct computation of the instability branches shows that, from very small to order unity primary wave amplitude, the theoretical prediction stay valid and is able to explain the strong departure from the classical triadic instability theory.

¹Support by DGA is acknowledged

Sunday, November 20, 2016 5:37PM - 6:55PM – Session E14 Waves: Faraday Waves C125-126 - Pablo Guterrez, Universidad de Chile

5:37PM E14.00001 Faraday Waves revisited, DIDIER CLAMOND, Laboratoire J.A. Dieudonné CNRS UMR 7351 Université de Nice, JEAN RAJCHENBACH, Physique de la Matière Condensée CNRS UMR 7336 Université de Nice — We revisit the theoretical description of Faraday waves. We show that, consistently with experiments, the relation of dispersion is not that of free unforced waves; the forcing amplitude and the viscosity play a significant role in the dispersion relation. We then determine the instability thresholds and the wavenumber selection in cases of both short and long waves. We also show that, depending on the depth, the instability leading to the formation can be either supercritical or subcritical, as shown by experimental observations.

5:50PM E14.00002 Streaming patterns in Faraday waves, PABLO GUTIERREZ, NICOLAS PERINET, Universidad de Chile, HECTOR URRA, Pontificia Universidad Católica de Valparaíso, NICOLAS MUJICA, Universidad de Chile, LEONARDO GORDILLO, University of Minnesota — Wave patterns in the Faraday instability have been studied for decades. Besides the rich wave dynamics observed at the interface, Faraday waves hide elusive flow patterns in the bulk –the streaming patterns– that have not been studied in detail until now. We analyse these streaming flows by conducting experiments in a Faraday-wave setup. To visualize the flows, we perform stroboscopic measurements: tracers are used to generate both trajectory maps and to probe the streaming velocity field via PIV. We identify three types of patterns that can coexist under identical Faraday waves. Next we propose a three-dimensional model that explains streaming flows in quasi-inviscid fluids. We show that the streaming inside the fluid arises from a complex coupling between the bulk and the boundary layers. This coupling can be taken into account by applying modified boundary conditions in a three-dimensional Navier-Stokes formulation for the streaming in the bulk. Numerical simulations based on this theoretical framework show good agreement with experimental results. Simulations reveal that the variety of experimental patterns is linked to the boundary condition at the top interface, which may be strongly affected by the presence of contaminants along the surface.

6:03PM E14.00003 Hysteretic Faraday waves¹, NICOLAS PÉRINET, CLAUDIO FALCÓN, DFI-FCFM, universidad de Chile, JALEL CHERGUI, LIMSI-CNRS, SEUNGWON SHIN, Department of Mechanical and System Design Engineering, Hongik University, DAMIR JURIC, LIMSI-CNRS — We study with numerical simulations the two-dimensional Faraday waves in two immiscible incompressible fluids when the lower fluid layer is shallow. After the appearance of the well known subharmonic stationary waves, a further instability is observed while the control parameter passes a secondary threshold. A new state then arises, composed of stationary waves with about twice the original pattern amplitude [1],[2]. The bifurcation presents hysteresis: there exists a finite range of the control parameter in which both states are stable. By means of a simple stress balance, we show that a change of the shear stress can explain this hysteresis [1]. Our predictions based on this model are in agreement with our numerical results.

[1] X. Li, Z. Yu and S. Liao, Phys. Rev. E 92, 033014 (2015)

[2] N. Péritet, C. Falcón, J. Chergui, D. Juric and S. Shin, Phys. Rev. E 93, 063114 (2016)

¹Project funded by FONDECYT grants 1130354, 3140522 and the National Research Foundation of Korea (NRF-2014R1A2A1A11051346). Computations supported by the supercomputing infrastructures of the NLHPC (ECM-02) and GENCI (IDRIS).

6:16PM E14.00004 Observation of Phillips's spectrum in Faraday waves¹, GUSTAVO CASTILLO, CLAUDIO FALCON, Universidad de Chile — We consider the problem of wave turbulence generated by singularities from an experimental point of view. We study a system of Faraday waves interacting with waves generated by a wave-maker driven with a random forcing. We measure the temporal fluctuations of the surface wave amplitude at a given location and we show that for a wide range of forcing parameters the surface height displays a power-law spectra that greatly differs from the one predicted by the WT theory. In the capillary region the power spectrum turns out to be proportional to f^{-5} , which we believe is due to singularities moving across the system.

¹Proyecto Postdoctorado Fondecyt Nro 3160032

6:29PM E14.00005 Galaxy-like organization of floaters at the air-water interface of Faraday waves¹, HÉCTOR ALARCÓN, NICOLAS PÉRINET, PABLO GUTIÉRREZ, Univ de Chile, LEONARDO GORDILLO, Univ of Minnesota - Twin Cities, NICOLÁS MUJICA, Univ de Chile — The fluid properties mismatch across an air-liquid interface allows to trap particles at it. These particles are called floaters and appear in nature at different scales: plankton, organic residues, and garbage, all relevant for the oceanic ecosystem. In static systems they tend to attract or repel each other, depending on their wetting properties and buoyancy [1]. When they are subjected to a flow, such as surface waves, they may drift and form structures at the interface [2]. In a recent work using PIV on Faraday waves, we have measured a streaming flow that emerges inside the bulk, leading to a slow circulation of fluid particles across the liquid [3]. The flow is mainly generated by the viscous shearing at the walls of the container. Our new experiments show that this flow has a remarkable effect on the drift of small hydrophilic particles (floaters), which leads to a rare arrangement of the floaters that resemble rotating galaxies. The forcing amplitude determines the galaxy shape, controlling the number and the length of its arms as well as its rotation velocity.
[1] Vella D. and Mahadevan L., Am. J. Phys. 73, 814 (2005)
[2] Sanli C. et al., Phys. Rev. E 89 (5), 053011 (2014)
[3] Péritet N. et al., <http://arxiv.org/abs/1603.07353> (2016)

¹Thanks to FONDECYT POSTDOCTORADO N3160341, N3140522, N3140550

6:42PM E14.00006 Wave - fluid particle interaction in the Faraday waves¹, NICOLAS FRANCOIS, HUA XIA, HORST PUNZMANN, MICHAEL SHATS, Australian Natl Univ — Faraday waves are parametrically excited perturbations that appear on a liquid surface when the latter is vertically vibrated. Recently it has been discovered that: 1) such wave field can be described as a disordered lattice made of localised oscillating excitations, termed oscillons, 2) the horizontal motion of fluid particles on the water surface reproduces in detail the motion of fluid in two-dimensional turbulence. Here we report experimental measurements of the motion of both entities using Particle Image Velocimetry and Particle Tracking Velocimetry techniques. Those techniques allow to measure Lagrangian and Eulerian features of the oscillon motion and compare them with those of the fluid motion. A strong coupling is uncovered between the erratic motion of the waves and the turbulent agitation of the fluid particles. Both motions show Brownian-type dispersion and the r.m.s velocity of oscillons is directly related to the r.m.s. velocity of the fluid particles in a broad range of vertical accelerations. These results offer new perspectives for predicting surface fluid transport from the knowledge of the wave fields and vice versa. In particular, the broadening of the wave spectra at high wave amplitude can be predicted if the 2D turbulence energy is known.

¹This work was supported by the Australian Research Council's Discovery Projects funding scheme (DP150103468 and DP160100863). NF acknowledges support by the Australian Research Council's DECRA Award (DE160100742).

Sunday, November 20, 2016 5:37PM - 6:55PM – Session E15 Bio: Vesicles and Micelles E143/144 - R. Sureshkumar, Syracuse University

5:37PM E15.00001 Branching, Superdiffusion and Stress Relaxation in Surfactant Micelles¹, R. SURESHKUMAR, S. DHAKAL, Syracuse University, SYRACUSE UNIVERSITY TEAM — We investigate the mechanism of branch formation and its effects on the dynamics and rheology of a model cationic micellar fluid using molecular dynamics (MD) simulations. Branched structures are formed upon increasing counter ion density. A sharp decrease in the solution viscosity with increasing salinity has long been attributed to the sliding motion of micellar branches along the main chain. Simulations not only provide firm evidence of branch sliding in real time, but also show enhanced diffusion of surfactants by virtue of such motion. Insights into the mechanism of stress relaxation associated with branch sliding will be discussed. Specifically, an externally imposed stress damps out more quickly in a branched system compared to that in an unbranched one. References: Dhakal and Sureshkumar, J. Chem. Phys. 143, 024905 (2015); ACS Macro Letters, 5, 108-11 (2016).

¹NSF Grants 1049489, 1049454.

5:50PM E15.00002 Brownian dynamics simulations of lipid bilayer membrane with hydrodynamic interactions in LAMMPS¹, SZU-PEI FU, YUAN-NAN YOUNG, New Jersey Institute of Technology, ZHANGLI PENG, University of Notre Dame, HONGYAN YUAN, University of Rhode Island — Lipid bilayer membranes have been extensively studied by coarse-grained molecular dynamics simulations. Numerical efficiencies have been reported in the cases of aggressive coarse-graining, where several lipids are coarse-grained into a particle of size $4 \sim 6$ nm so that there is only one particle in the thickness direction. Yuan *et al.* proposed a pair-potential between these one-particle-thick coarse-grained lipid particles to capture the mechanical properties of a lipid bilayer membrane (such as gel-fluid-gas phase transitions of lipids, diffusion, and bending rigidity). In this work we implement such interaction potential in LAMMPS to simulate large-scale lipid systems such as vesicles and red blood cells (RBCs). We also consider the effect of cytoskeleton on the lipid membrane dynamics as a model for red blood cell (RBC) dynamics, and incorporate coarse-grained water molecules to account for hydrodynamic interactions. The interaction between the coarse-grained water molecules (explicit solvent molecules) is modeled as a Lennard-Jones (L-J) potential. We focus on two sets of LAMMPS simulations: 1. Vesicle shape transitions with varying enclosed volume; 2. RBC shape transitions with different enclosed volume.

¹This work is funded by NSF under grant DMS-1222550

6:03PM E15.00003 Lamellar ordering, droplet formation and phase inversion in exotic active emulsions, GIUSEPPE GONNELLA, Universit degli Studi di Bari, FRANCESCO BONELLI, Politecnico di Bari, DAVIDE MARENDOZZO, University of Edinburgh, ENZO ORLANDINI, ADRIANO TIRIBOCCHI, Universit degli Studi di Padova — We present the results of numerical simulations of the behaviour of a mixture of a passive isotropic fluid and an active polar nematic gel, in presence of surfactant favouring emulsification. The active stress appearing in the Navier-Stokes equation depends on the polarization field and on an activity parameter whose sign determines the contractile or extensile character of the gel. Focussing on cases for which the underlying free energy favours the lamellar phase in the passive limit, we show that the interplay between nonequilibrium and thermodynamic forces creates a range of multifarious exotic emulsions. When the active component is contractile (e.g., an actomyosin solution), moderate activity greatly enhances the efficiency of lamellar ordering, whereas strong activity favours the creation of passive droplets within an active matrix. For extensile activity (e.g., materials based on bacterial suspensions), instead, we observe an emulsion of spontaneously rotating droplets. By tuning the overall composition, we can also create high internal phase emulsions, which undergo catastrophic phase inversion when switching off the activity. Therefore, we find that activity may provide a single control parameter to design composite materials with a rich range of morphologies.

6:16PM E15.00004 Stable low-resolution simulations of two-dimensional vesicle suspensions, GOKBERK KABACAOGLU, University of Texas at Austin, BRYAN QUAIFE, Florida State University, GEORGE BIROS, University of Texas at Austin — Vesicles, which resist bending and are locally inextensible, serve as experimental and numerical proxies for red blood cells. Vesicle flows, which are governed by hydrodynamic and elastic forces, refer to flow of vesicles that are filled with and suspended in a Stokesian fluid. In this work we present algorithms for stable and accurate low-resolution simulations of the vesicle flows in two-dimensions. We use an integral equation formulation of the Stokes equation coupled to the interface mass continuity and force balance. The problem poses numerical difficulties such as long-range hydrodynamic interactions, strong nonlinearities and stiff governing equations. These difficulties make simulations with long time horizons challenging, especially at low resolutions. We develop algorithms to control aliasing errors, correct errors in vesicle's area and arc-length, and avoid collision of vesicles. Additionally, we discuss several error measures to study the accuracy of the simulations. Then we closely look at how accurate the low-resolution simulations can capture true physics of the vesicle flows.

6:29PM E15.00005 Inducing morphological changes in lipid bilayer membranes with microfabricated substrates, FANGJIE LIU, LIAM F. COLLINS, RANA ASHKAR, FREDERICK A. HEBERLE, BERNADETA R. SRIJANTO, C. PATRICK COLLIER, Oak Ridge National Laboratory — Lateral organization of lipids and proteins into distinct domains and anchoring to a cytoskeleton are two important strategies employed by biological membranes to carry out many cellular functions. However, these interactions are difficult to emulate with model systems. Here we use the physical architecture of substrates consisting of arrays of micropillars to systematically control the behavior of supported lipid bilayers – an important step in engineering model lipid membrane systems with well-defined functionalities. Competition between attractive interactions of supported lipid bilayers with the underlying substrate versus the energy cost associated with membrane bending at pillar edges can be systematically investigated as functions of pillar height and pitch, chemical functionalization of the microstructured substrate, and the type of unilamellar vesicles used for assembling the supported bilayer. Confocal fluorescent imaging and AFM measurements highlight correlations that exist between topological and mechanical properties of lipid bilayers and lateral lipid mobility in these confined environments. This study provides a baseline for future investigations into lipid domain reorganization on structured solid surfaces and scaffolds for cell growth.

6:42PM E15.00006 Equilibrium Shapes of Compound Vesicles, CANGJIE XU, MICHAEL MIKSIS, STEPHEN DAVIS, Northwestern University — Many biological structures have a fine internal structure in which a membrane is geometrically confined by another membrane. Here we investigate how the equilibrium shape of a double membrane system changes as the length of the internal membrane is increased. A repulsive pressure is introduced between the membranes to prevent the membranes from intersecting. Large repulsive pressures yield complex response diagrams with bifurcation points where modal identities may change. Regions in parameter space where such behavior occurs are then mapped.

Sunday, November 20, 2016 5:37PM - 6:55PM – Session E16 Drops: Solidification D133/134 - Detlef Lohse, University of Twente

5:37PM E16.00001 Dynamics of ice drop explosions in supercooled clouds, DETLEF LOHSE, SANDER WILDEMAN, SEBASTIAN STERL, CHAO SUN, University of Twente — The rate at which ice particles are produced in the cold top of natural clouds is crucial in predicting whether a cloud will finally develop precipitation. It has been speculated that ice particles could multiply through freezing and subsequent bursting of supercooled cloud droplets. Here we present high-speed footage of cracking and explosive bursting of spherical water droplets that freeze radially inwards under carefully controlled conditions. We model the processes of freezing, the stress build up in the ice shell, and the fast dynamics following the crack formation. This allows us to predict the time it takes for a freezing droplet to explode and the energy released in this event as a function of the size of the droplet and the temperature of the surroundings. Both predictions are in good agreement with our experiments. The models also predict a minimum droplet radius of approximately $50\mu\text{m}$ below which ice explosions are unlikely to occur. This finding has direct consequences in the modeling of cloud microphysics, as the droplet sizes in clouds generally fall in this critical range. Furthermore, we identify several mechanisms, besides the final explosion, by which a freezing drop can shed ice particles. This is important for the formation of ice nucleation avalanches.

5:50PM E16.00002 Dueling Mechanisms for Dry Zones around Frozen Droplets, CAITLIN BISBANO, SAURABH NATH, JONATHAN BOREYKO, Virginia Tech — Ice acts as a local humidity sink, due to its depressed saturation pressure relative to that of supercooled water. Hygroscopic chemicals typically exhibit annular dry zones of inhibited condensation; however, dry zones do not tend to form around ice because of inter-droplet frost growth to nearby liquid droplets that have already condensed on the chilled surface. Here, we use a humidity chamber with an embedded Peltier stage to initially suppress the growth of condensation on a chilled surface containing a single frozen droplet, in order to characterize the dry zone around ice for the first time. The length of the dry zone was observed to vary by at least two orders of magnitude as a function of surface temperature, ambient humidity, and the size of the frozen droplet. The surface temperature and ambient humidity govern the magnitudes of the in-plane and out-of-plane gradients in vapor pressure, while the size of the frozen droplet effects the local thickness of the concentration boundary layer. We develop an analytical model that reveals two different types of dry zones are possible: one in which nucleation is inhibited and one where the net growth of condensate is inhibited. Finally, a phase map was developed to predict the parameter space in which nucleation dry zones versus flux dry zones are dominant.

6:03PM E16.00003 Frozen Impacted Drop: from Fragmentation to Hierarchical Crack Patterns, THOMAS SON, ELISABETH GHABACHE, CHRISTOPHE JOSSERAND, CNRS, Institut d'Alembert, Paris — We investigate experimentally the quenching of a liquid pancake, obtained through the impact of a water drop on a cold solid substrate (0°C to -60°C). We show that, below a certain substrate temperature, fractures appear on the frozen pancake and the crack patterns change from a 2D fragmentation regime to a hierarchical fracture regime as the thermal shock increases. The different regimes are discussed and the transition temperatures are estimated through classical fracture scaling arguments. Finally, a phase diagram presents how these regimes can be controlled by the drop impact parameters.

6:16PM E16.00004 Scaling Laws for Inter-droplet Ice Bridging, SAURABH NATH, FARZAD AHMADI, JONATHAN BOREYKO, Virginia Tech — In this work, we study the dynamics of an ice bridge growing from a frozen droplet towards its neighboring supercooled liquid droplet. Experiments were done on a Peltier stage inside a humidity chamber with deposited or condensed droplets where the substrate temperature and ambient humidity could be controlled. Following a quasi-steady diffusion-driven model, we develop scaling laws to show how the growth rate depends on the substrate temperature, droplet sizes and inter-droplet distances over and above other environmental parameters such as air temperature and humidity. The growth rate as well as the success or failure of an ice bridge to connect to its neighboring liquid droplet depend on a nondimensional number called the separation parameter S^* , defined as the ratio of the initial inter-droplet spacing to the diameter of the evaporating liquid droplet. It is shown that the maximum value of S^* for connection scales as 1 as long as frozen drop is larger than the liquid droplet. For the converse case of a larger water drop, there are at least three separate regimes of critical S^* , depending on whether the water drop is a puddle, a spherical cap or if the frozen drop is a puddle.

6:29PM E16.00005 Contact line arrest in solidifying spreading drops, RIELLE DE RUITER, University of Twente, PIERRE COLINET, Universit Libre de Bruxelles, JACCO SNOEIJER, HANNEKE GELDERBLOM, University of Twente — When does a drop, deposited on a cold substrate, stop spreading? Despite the practical relevance of this question, for example in airplane icing and 3D metal printing, the exact mechanism of arrest in solidifying spreading drops has not yet been unraveled. Here, we consider the spreading and arrest of hexadecane drops of constant volume on two smooth wettable substrates; copper with a high thermal conductivity and glass with a low thermal conductivity. We record the spreading radius and contact angle in time for a range of substrate temperatures. We show that our measurements on both copper and glass are well explained by a contact line arrest condition based on crystallization kinetics, which takes into account the effect of kinetic undercooling and the thermal conductivity of the substrate.

6:42PM E16.00006 Interface solidification of impinging metal drops, JOLET DE RUITER, DAN SOTO, KRIPA VARANASI, Massachusetts Institute of Technology — Molten metal droplet deposition is important in manufacturing techniques such as spray deposition and metal inkjet printing. Key parameters are the final splat morphology and its adhesion to the base substrate. How to control these parameters is still poorly understood, since droplet deformation, cooling and solidification happen simultaneously. Here, we studied the contact patch formed between the metal drop and the base substrate, varying the thermal properties and the initial temperature of the substrate, and the initial temperature of the drop. We identify various scenarios for interface solidification, including smooth liquid-spread contact patches, entrapment of air pockets, and transient re-melting of the interface. The transitions between various scenarios can be rationalized from the interfacial temperature estimated by heat conduction, and taking into account the flow of liquid metal.

Sunday, November 20, 2016 5:37PM - 6:55PM –
Session E17 Reacting Flows: Numerical Methods D131 - D.C. Haworth, Pennsylvania State University

5:37PM E17.00001 Improved Stiff ODE Solvers for Combustion CFD, A. IMREN, D.C. HAWORTH, Penn State — Increasingly large chemical mechanisms are needed to predict autoignition, heat release and pollutant emissions in computational fluid dynamics (CFD) simulations of in-cylinder processes in compression-ignition engines and other applications. Calculation of chemical source terms usually dominates the computational effort, and several strategies have been proposed to reduce the high computational cost associated with realistic chemistry in CFD. Central to most strategies is a stiff ordinary differential equation (ODE) solver to compute the change in composition due to chemical reactions over a computational time step. Most work to date on stiff ODE solvers for computational combustion has focused on backward differential formula (BDF) methods, and has not explicitly considered the implications of how the stiff ODE solver couples with the CFD algorithm. In this work, a fresh look at stiff ODE solvers is taken that includes how the solver is integrated into a turbulent combustion CFD code, and the advantages of extrapolation-based solvers in this regard are demonstrated. Benefits in CPU time and accuracy are demonstrated for homogeneous systems and compression-ignition engines, for chemical mechanisms that range in size from fewer than 50 to more than 7,000 species.

5:50PM E17.00002 Efficient partially implicit integration method for stiff chemistry in high-fidelity simulations of turbulent reacting flows¹, HAO WU, MATTHIAS IHME, Stanford University — High-fidelity turbulent reactive flow simulations are typically associated with small time step sizes ($h \leq 10^{-8}$ sec) due to the CFL condition imposed by the fine grid. Although the step size is not sufficiently small to allow fully explicit time integration in the presence of stiff chemistry, it makes the use of classical implicit multi-step ODE solvers (e.g. VODE) an inefficient approach in combustion simulations due to the reduced number of internal iterations and excessive implicitness. In this study, an improved 4th-order Rosenbrock-Krylov (ROK4L) scheme is developed for the system of chemical reactions. This class of schemes replaces the Jacobian matrix by its low-rank Krylov approximation, thus introducing partial implicitness. The scheme is improved in both accuracy and efficiency by fulfilling additional order conditions and reducing the number of function evaluations. The ROK4L scheme is demonstrated to possess superior efficiency in comparison to CVODE due to the minimal degree of implicitness for small time-step sizes and the avoidance of other overhead associated with the start-up process of multi-step methods.

¹Financial support from NASA Transformational Tools and Technologies Project with Award No. NNX15AV04A.

6:03PM E17.00003 A high-order immersed boundary method for high-fidelity turbulent combustion simulations¹, YUKI MINAMOTO, KOZO AOKI, KOSUKE OSAWA, Tokyo Institute of Technology, TUO SHI, The Hong Kong University of Science and Technology, ALEXANDRU PRODAN, Delft University of Technology, MAMORU TANAHASHI, Tokyo Institute of Technology — Direct numerical simulations (DNS) have played important roles in the research of turbulent combustion. With the recent advancement in high-performance computing, DNS of slightly complicated configurations such as V-, various jet and swirl flames have been performed, and such DNS will further our understanding on the physics of turbulent combustion. Since these configurations include walls that do not necessarily conform with the preferred mesh coordinates for combustion DNS, most of these simulations use presumed profiles for inflow/near-wall flows as boundary conditions. A high-order immersed boundary method suited for parallel computation is one way to improve these simulations. The present research implements such a boundary technique in a combustion DNS code, and simulations are performed to confirm its accuracy and performance.

¹This work was partly supported by Council for Science, Technology and Innovation, Cross-ministerial Strategic Innovation Promotion Program (SIP), Innovative Combustion Technology (Funding agency: JST).

6:16PM E17.00004 ABSTRACT WITHDRAWN —

6:29PM E17.00005 Adjoint based sensitivity analysis of a reacting jet in crossflow, PALASH SASHITAL, TARANEH SAYADI, University of Illinois, Urbana-Champaign, PETER SCHMID, Imperial College London — With current advances in computational resources, high fidelity simulations of reactive flows are increasingly being used as predictive tools in various industrial applications. In order to capture the combustion process accurately, detailed/reduced chemical mechanisms are employed, which in turn rely on various model parameters. Therefore, it would be of great interest to quantify the sensitivities of the predictions with respect to the introduced models. Due to the high dimensionality of the parameter space, methods such as finite differences which rely on multiple forward simulations prove to be very costly and adjoint based techniques are a suitable alternative. The complex nature of the governing equations, however, renders an efficient strategy in finding the adjoint equations a challenging task. In this study, we employ the modular approach of Fosas de Pando *et al.* (2012), to build a discrete adjoint framework applied to a reacting jet in crossflow. The developed framework is then used to extract the sensitivity of the integrated heat release with respect to the existing combustion parameters. Analyzing the sensitivities in the three-dimensional domain provides insight towards the specific regions of the flow that are more susceptible to the choice of the model.

6:42PM E17.00006 Computational flow field in energy efficient engine (EEE)¹, KENJI MIKI, JEFF MODER, MENG-SING LIOU, NASA Glenn Research Center — In this paper, preliminary results for the recently-updated Open National Combustor Code (Open NCC) as applied to the EEE are presented. The comparison between two different numerical schemes, the standard Jameson-Schmidt-Turkel (JST) scheme and the advection upstream splitting method (AUSM), is performed for the cold flow and the reacting flow calculations using the RANS. In the cold flow calculation, the AUSM scheme predicts a much stronger reverse flow in the central recirculation zone. In the reacting flow calculation, we test two cases: gaseous fuel injection and liquid spray injection. In the gaseous fuel injection case, the overall flame structures of the two schemes are similar to one another, in the sense that the flame is attached to the main nozzle, but is detached from the pilot nozzle. However, in the exit temperature profile, the AUSM scheme shows a more uniform profile than that of the JST scheme, which is close to the experimental data. In the liquid spray injection case, we expect different flame structures in this scenario. We will give a brief discussion on how two numerical schemes predict the flame structures inside the Eusing different ways to introduce the fuel injection.

¹Supported by NASA's Transformational Tools and Technologies project.

Sunday, November 20, 2016 5:37PM - 6:42PM — Session E18 Flow Instability: Global Modes D135 - Clarence Rowley, Princeton University

5:37PM E18.00001 Efficient randomized methods for stability analysis of fluids systems¹, SCOTT DAWSON, CLARENCE ROWLEY, Princeton University — We show that probabilistic algorithms that have recently been developed for the approximation of large matrices can be utilized to numerically evaluate the properties of linear operators in fluids systems. In particular, we present an algorithm that is well suited for optimal transient growth (i.e., nonmodal stability) analysis. For non-normal systems, such analysis can be important for analyzing local regions of convective instability, and in identifying high-amplitude transients that can trigger nonlinear instabilities. Our proposed algorithms are easy to wrap around pre-existing timesteppers for linearized forward and adjoint equations, are highly parallelizable, and come with known error bounds. Furthermore, they allow for efficient computation of optimal growth modes for numerous time horizons simultaneously. We compare the proposed algorithm to both direct matrix-forming and Krylov subspace approaches on a number of test problems. We will additionally discuss the potential for randomized methods to assist more broadly in the speed-up of algorithms for analyzing both fluids data and operators.

¹Supported by AFOSR grant FA9550-14-1-0289

5:50PM E18.00002 Global-mode based linear feedback control of a supersonic jet for noise reduction¹, 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 with a control strategy that uses global modes to model their dynamics and structural sensitivity of the linearized compressible Navier-Stokes operator to identify an effective linear feedback control. For a case with co-located actuators and sensors adjacent the nozzle, we demonstrate the method on an axisymmetric Mach 1.5 jet. Direct numerical simulations using this control show significant noise reduction. Eigenanalysis of the controlled mean flows reveal fundamental changes in the spectrum at frequencies lower than that used by the control, with the quieter flows having unstable eigenvalues that correspond to eigenfunctions without significant support in the acoustic field. A specific trend is observed in the mean flow quantities as the flow becomes quieter, with changes in the mean flow becoming significant only further downstream of the nozzle exit. The quieter flows also have a stable shock-cell structure that extends further downstream. A phase plot of the POD coefficients for the flows show that the quieter flows are more regular in time.

¹Funded by the Office of Naval Research

²PhD student, Department of Aerospace Engineering

³Professor, Department of Mechanical Science and Engineering and Department of Aerospace Engineering

⁴Associate Professor, Department of Aerospace Engineering

6:03PM E18.00003 Linear stability analysis of axisymmetric flow over a sudden expansion in an annular pipe, BEHNAZ BELADI, HENDRIK CHRISTOPH KUHLMANN, TU Wien — A global temporal linear stability analysis is performed of the fully-developed axisymmetric incompressible Newtonian flow in an annular pipe with a sudden radially-inward expansion. The geometry is characterized by the radial expansion ratio (radial step height to the outlet gap width) and the outlet radius ratio (inner-to-outer radius). Stability boundaries have been calculated with finite volumes for an outlet radius ratio of 0.1 and expansion ratios from 0.25 to 0.75. For expansion ratios less than 0.55 the most dangerous mode has an azimuthal wave number $m = 3$, whereas $m = 2$ for larger expansion ratios. An *a posteriori* analysis of the kinetic energy transferred between the basic state and the critical mode allows to check the energy conservation and to identify the physical instability mechanism. For all expansion ratios considered the basic flow arises as an annular jet between two separation zones which are located immediately after the step. The jet gradually widens downstream before reattaching to the cylinders. The deceleration of the flow associated with the widening of the jet is found to be the primary source of energy for the critical modes.

6:16PM E18.00004 Stabilization of flow past a rounded cylinder¹, RAVI SAMTANEY, WEI ZHANG, King Abdullah University of Science and Technology — We perform global linear stability analysis on low- Re flow past a rounded cylinder. The cylinder corners are rounded with a radius R , normalized as $R^+ = R/D$ where D is the cylinder diameter, and its effect on the flow stability characteristics is investigated. We compute the critical Reynolds number (Re_{cr}) for the onset of first instability, and quantify the perturbation growth rate for the super-critical flows. It is found that *the flow can be stabilized by partially rounding the cylinder*. Compared with the square and circular cylinders, the partially rounded cylinder has a higher Re_{cr} , attaining a maximum at around $R^+ = 0.30$, and the perturbation growth rate of the super-critical flows is reduced for $Re \leq 100$. We perform sensitivity analysis to explore the source of the stabilization. The growth rate sensitivity to base flow modification has two different spatial structures: the growth rate is sensitive to the wake backflow in a large region for square-like cylinders ($R^+ \rightarrow 0.00$), while only the near-wake backflow is crucial for circular-like cylinders ($R^+ \rightarrow 0.50$). The stability analysis results are also verified with those of the direct simulations and very good agreement is achieved.

¹Supported by the KAUST Office of Competitive Research Funds under Award No. URF/1/1394-01. The supercomputer Shaheen at KAUST was utilized for the simulations.

6:29PM E18.00005 Stability of optimal streaks in the buffer layer of a turbulent channel flow with variable viscosity, ASHISH PATEL, Delft University of Technology, ENRICO RINALDI, Linne FLOW Centre, KTH Mechanics, RENE PECNIK, Delft University of Technology, PHILIPP SCHLATTER, SHERVIN BAGHERI, Linne FLOW Centre, KTH Mechanics — Direct Numerical Simulations (DNS) of turbulent channel flows with variable viscosity (Patel et al., 2015, PoF) show that low speed streaks in the buffer layer strengthen and are stabilized for increasing viscosity away from the wall, as they do not lift and tilt as intensely as in a constant property flow. The opposite holds for cases where viscosity decreases away from the wall. In this work, we investigate the above observation by studying the linear stability of the mean turbulent velocity profile obtained from DNS of variable viscosity flows. Examples of such studies for constant property turbulent flows include work of del Alamo & Jimnez, 2006, JFM and Pujals et al., 2009, PoF. The calculated optimal buffer layer streaks show larger transient energy growth for a case where the viscosity increases away from the wall. We further study the stability of the saturated optimal streaks by imposing a secondary sinuous perturbation and by following the nonlinear evolution of the structures in time. The present investigation will improve the understanding of the near-wall turbulence cycle for wall-bounded turbulent flows with viscosity gradients.

Sunday, November 20, 2016 5:37PM - 6:55PM –
Session E19 Bio: Motion in Non-Newtonian Fluids D136 - Derek Tretheway, Portland State University

5:37PM E19.00001 Role of passive body dynamics in micro-organism swimming in complex fluids, BECCA THOMASES, ROBERT GUY, University of California, Davis — We investigate the role of passive body dynamics in the kinematics of swimming micro-organisms in complex fluids. Asymptotic analysis and linear theory are used to predict shape changes that result as body elasticity and fluid elasticity are varied. The analysis is compared with a computational model of a finite length swimmer in a Stokes-Oldroyd-B fluid. Simulations and theory agree quantitatively for small amplitude motions with low fluid elasticity (Deborah number). This may not be surprising as the theory is expected hold in these two regimes. What is more remarkable is that the predicted shape changes match the computational shape changes quantitatively for large amplitudes, even for large Deborah numbers. Shape changes only tell part of the story. Swimming speed depends on other effects as well. We see that shape changes can predict swimming speed well when either the amplitude is small (including large Deborah number) or when the Deborah number is small (including large amplitudes). It is only in the large De AND large amplitude regime where the theory breaks down and swimming speed can no longer be inferred from shape changes alone.

5:50PM E19.00002 An active particle in a complex fluid, CHARU DATT, GIOVANNIANTONIO NATALE, SAVVAS G. HATZIKIRIAKOS, GWYNNE J. ÉLFRING, University of British Columbia, Vancouver — Active particles are self-driven units capable of converting stored or ambient free-energy into systematic movement. We discuss here the case when such particles move through non-Newtonian fluids. Neglecting inertial forces, we employ the reciprocal theorem to calculate the propulsion velocity of a single swimmer in a weakly non-Newtonian fluid with background flow. We also derive a general expression for the velocity of an active particle modelled as a squirmer in a second-order fluid. We then discuss how active colloids are affected by the medium rheology, namely viscoelasticity and shear-thinning.

6:03PM E19.00003 Swimming in mud, NEIL BALMFORTH, University of British Columbia, DUNCAN HEWITT, University of Cambridge — We extend G.I. Taylor's classic problem of the swimming of a flexible sheet in a viscous fluid driven by waves propagating down its length. In particular, we add a yield stress to the problem and calculate how the swimming speed is modified for waves of both low and high amplitude. We examine the flow patterns created around the swimmer as it locomotes and comment on designing strategies for optimal progress.

6:16PM E19.00004 Undulatory Swimming in Shear-Thinning Fluids: Flow Fields & Power Consumption, DAVID GAGNON, University of Pennsylvania, THOMAS MONTENEGRO-JOHNSON, ERIC LAUGA, University of Cambridge, PAULO ARRATIA, University of Pennsylvania — In this talk, we investigate the flow and dynamics of the undulatory swimmer *Caenorhabditis elegans* in shear-thinning fluids. Recent theoretical and numerical studies have shown that the cost of swimming, or mechanical power, for a 2D waving sheet is reduced in shear-thinning fluids. Here, we use velocimetry and tracking techniques to experimentally investigate this hypothesis using two methods: (i) an estimate of the mechanical power of the swimmer and (ii) the viscous dissipation rate of the flow field. We find the cost of swimming for *C. elegans* in shear-thinning fluids is reduced when compared to the cost of swimming in Newtonian fluids, scales with a fluids effective viscosity, and can be predicted using fluid rheology and simple swimming kinematics. These results, however, have a caveat: only a planar (2D) slice of the 3D flow field around swimmer is accessible for analysis. In order to better interpret our flow measurements, we compare our planar velocimetry to a full 3D boundary element method simulation. We find that nearly all deviations between experiments and simulations can be accounted for by a simple correction factor involving the out-of-plane velocity gradient, which can be computed directly from planar experimental data using incompressibility.

6:29PM E19.00005 Micro-scale undulatory locomotion in heterogeneous viscoelastic environments, ARSHAD KAMAL, ERIC KEAVENY, Imperial College London — While many microorganisms swim in viscoelastic fluids, there are notable examples where the suspended microstructure that makes the fluid viscoelastic is at the same length scale as the swimmers. Here, the swimming cells experience the surrounding medium as a set of obstacles suspended in a viscous fluid, rather than a viscoelastic continuum. Using simulations based on the force-coupling method, we explore this situation for a simple undulatory swimmer as it moves through an environment of obstacles that are tethered to random points in space via linear springs. We examine how swimming behaviour is altered by mechanical interactions with the obstacles by varying obstacle density and tether stiffness. We find that the mechanical interactions can either enhance or hinder locomotion, and often for fixed tether stiffness, there is an obstacle density for which the average speed is maximized. We also find cases where the swimmer is completely trapped by the environment. In addition, we find that the velocity fluctuations, and consequently the effective swimmer diffusion, are also highly dependent on environment composition and a non-monotonic dependence on the relevant parameters can be found here as well.

6:42PM E19.00006 Helicobacter pylori Couples Motility and Diffusion to Actively Create a Heterogeneous Complex Medium in Gastric Mucus¹, HENRY FU, SEYED AMIR MIRBAGHERI, University of Utah — *Helicobacter pylori* swims through mucus gel by generating ammonia that locally neutralizes the acidic gastric environment, turning nearby gel into a fluid pocket. The size of the fluid zone is important for determining the physics of the motility: in a large zone swimming occurs as in a fluid through hydrodynamic principles, while in a very small zone the motility could be strongly influenced by nonhydrodynamic cell-mucus interactions including chemistry and adhesion. We calculate the size of the fluid pocket. We model how swimming depends on the de-gelation range using a Taylor sheet swimming through a layer of Newtonian fluid bounded by a Brinkman fluid. Then, we model how the de-gelation range depends on the swimming speed by considering the advection-diffusion of ammonia exuded from a translating sphere. Self-consistency between both models determines the values of the swimming speed and the de-gelation range. We find that *H. pylori* swims through mucus as if unconfined, in a large pocket of Newtonian fluid.

¹Funded by National Science Foundation award CBET-1252182

Sunday, November 20, 2016 5:37PM - 6:55PM – Session E20 Bio: Red Blood Cells D137-138 - Zhangli Peng, University of Notre Dame

5:37PM E20.00001 Particle-based modeling effect of shape transform of single sickle red blood cells, JUN YANG, MIT, GEORGE KARNIADAKIS, Brown University, MING DAO, MIT — Sickle red blood cells often exhibit various sickled shapes as well as higher shear and bending stiffness. To study the membrane biomechanical properties related to cell morphology, we employed multiscale coarse grain models based on dissipative particle dynamics (DPD). Through the proper orthogonal decomposition (POD) we analyse the membrane fluctuation of a single cell which probe the membrane mechanical properties. In this work, the membrane mechanics alteration caused by cell volume and surface area variation are tested. We verified that with same ratio of surface area and volume, volume differences will not affect the membrane fluctuation. We also found that by expanding the whole cell the membrane fluctuation performance does not change. To further quantify the pure shape effects, we generate cells with different aspect ratio of major axis and minor axis at which membrane exhibit different fluctuation indicating the mechanical properties divergence. Through the spatial-temporal autocorrelation of membrane fluctuations characteristics, the membrane bending stiffness and shear modulus are carefully calibrated against QPI experimental data.

5:50PM E20.00002 Human spleen and red blood cells, IGOR PIVKIN, Università della Svizzera italiana (Lugano), Switzerland, ZHANGLI PENG, University of Notre Dame, USA, GEORGE KARNIADAKIS, Brown University, USA, PIERRE BUFFET, Institut National de la Transfusion Sanguine, France, MING DAO, Massachusetts Institute of Technology, USA — Spleen plays multiple roles in the human body. Among them is removal of old and altered red blood cells (RBCs), which is done by filtering cells through the endothelial slits, small micron-sized openings. There is currently no experimental technique available that allows us to observe RBC passage through the slits. It was previously noticed that people without a spleen have less deformable red blood cells, indicating that the spleen may play a role in defining the size and shape of red blood cells. We used detailed RBC model implemented within the Dissipative Particle Dynamics (DPD) simulation framework to study the filter function of the spleen. Our results demonstrate that spleen indeed plays major role in defining the size and shape of the healthy human red blood cells.

6:03PM E20.00003 Dynamics of Red Cells in Spleen: How Does Vesiculation Happen?, QIANG ZHU, SARA SALEHYAR, PEDRO CABRALES, ROBERT ASARO, UC San Diego — Vesiculation of red blood cells as a result of local separation between lipid bilayer and cytoskeleton is known to happen in vivo, most likely inside spleen where they sustain large mechanical loads during the passage through venus slits. There is, however, little knowledge about the detailed scenario and condition. We address this question via a fluid-cell interaction model by coupling a multiscale model of the cell membrane (including molecular details) with a fluid dynamics model based on boundary-integral equations. A numerical flow channel is created where the cell is driven through a narrow slit by pressure (imitating the transit through venus slits in spleen). The concentration is the occurrence of large dissociation (negative) pressure between the skeleton/membrane connection that promotes separation, a precursor of vesicle formation. Critical levels for the negative pressure are estimated using published data. By following the maximum range of pressure, we conclude that for vesiculation to happen there must be biochemical influences (e.g. binding of degraded haemoglobin) that significantly reduce effective attachment density. This is consistent with reported trends in vesiculation that are believed to occur in cases of various hereditary anemias and during blood storage. Our findings also suggest the criticality of understanding the biochemical phenomena involved with cytoskeleton/membrane attachment.

6:16PM E20.00004 Dynamics of Red Blood Cells through submicronic splenic slits¹

, EMMANUELLE HELFER, PRIYA GAMBHIKE, SCOTT ATWELL, FREDERIC BEDU, IGOR OZEROV, ANNIE VIALLAT, ANNE CHARRIER, Aix Marseille Univ, CNRS, CINaM, Marseille, France, CATHERINE BADENS, Aix Marseille Univ, APHM, La Timone, Department of genetics, Marseille, France, CENTRE DE REFERENCE THALASSEMIE, BADENS TEAM, PHYSICS AND ENGINEERING OF LIVING SYSTEMS TEAM — Red Blood Cells (RBCs) are periodically monitored for changes in their deformability by the spleen, and are entrapped and destroyed if unable to pass through the splenic interendothelial slits (IESs). In particular, in sickle cell disease (SCD), where hemoglobin form fibers inside the RBCs, and in hereditary spherocytosis (HS), where RBCs are more spherical and membrane-cytoskeleton bonds are weakened, the loss of RBC deformability leads to spleen dysfunction. By combining photolithography and anisotropic wet etching techniques, we developed a new on-chip PDMS device with channels replicating the submicronic physiological dimensions of IESs to study the mechanisms of deformation of the RBCs during their passage through these biomimetic slits. For the first time, with HS RBCs, we show the disruption of the links between the RBC membrane and the underlying spectrin network. In the case of SCD RBCs we show the appearance of a tip at the front of the RBC with a longer time relaxation due to the increased cytoplasmic viscosity.

¹This work has been carried out thanks to the support of the A*MIDEX project (n ANR-11-IDEX-0001-02) funded by the Investissements d'Avenir French Government program, managed by ANR

6:29PM E20.00005 Multiscale Modeling of Red Blood Cells Squeezing through Sub-micron Slits , ZHANGLI PENG, HUIJIE LU, University of Notre Dame — A multiscale model is applied to study the dynamics of healthy red blood cells (RBCs), RBCs in hereditary spherocytosis, and sickle cell disease squeezing through submicron slits. This study is motivated by the mechanical filtration of RBCs by inter-endothelial slits in the spleen. First, the model is validated by comparing the simulation results with experiments. Secondly, the deformation of the cytoskeleton in healthy RBCs is investigated. Thirdly, the mechanisms of damage in hereditary spherocytosis are investigated. Finally, the effects of cytoplasm and membrane viscosities, especially in sickle cell disease, are examined. The simulations results provided guidance for future experiments to explore the dynamics of RBCs under extreme deformation.

6:42PM E20.00006 Stretching Behavior of Red Blood Cells at High Strain Rates , JORDAN MANCUSO, WILLIAM RISTENPART, Dept. Chemical Engineering, University of California Davis — Most work on the mechanical behavior of red blood cells (RBCs) has focused on simple shear flows. Relatively little work has examined RBC deformations in the physiologically important extensional flow that occurs at the entrance to a constriction. In particular, previous work suggests that RBCs rapidly stretch out and then retract upon entering the constriction, but to date no model predicts this behavior for the extremely high strain rates typically experienced there. In this work, we use high speed video to perform systematic measurements of the dynamic stretching behavior of RBCs as they enter a microfluidic constriction. We demonstrate that a simple viscoelastic model captures the observed stretching dynamics, up to strain rates as high as 1000 s^{-1} . The results indicate that the effective elastic modulus of the RBC membrane at these strain rates is an order of magnitude larger than moduli measured by micropipette aspiration or other low strain rate techniques.

Sunday, November 20, 2016 5:37PM - 6:55PM –

Session E21 Bubbles: Cavitation, Collapse D139-140 - Eric Johnsen, University of Michigan

5:37PM E21.00001 Simulations of Shock-induced Bubble Collapse near Hard and Soft Objects¹ , MAURO RODRIGUEZ, ERIC JOHNSEN, University of Michigan, Ann Arbor — Understanding the dynamics of cavitation bubbles and shock waves in and near hard and soft objects is important particularly in various naval and medical applications. Two examples are therapeutic ultrasound procedures, which utilize this phenomenon for breaking kidney stones (lithotripsy) and ablation of pathogenic tissue (histotripsy), 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 of the pulses. To understand the damage mechanism, it is of key interest to quantifying the influence of the shock waves on the material and the response of the material to the shock waves. A novel Eulerian numerical approach for simulating shock and acoustic wave propagation in viscoelastic media is leveraged to understand this influence. High-fidelity simulations of the bubble collapse dynamics for various experimental configurations (i.e. the viscous or viscoelastic material surrounding the bubble and neighboring object's rigidity are varied) will be conducted. In particular, we will discuss the shock emission from collapse and its propagation in the neighboring object, including stresses thereby produced.

¹This research was supported in part by ONR grant N00014-12-1-0751 under Dr. Ki-Han Kim and by NSF grant number CBET 1253157.

5:50PM E21.00002 The effects of bubble-bubble interactions on pressures and temperatures produced by bubbles collapsing near a rigid surface , SHAHABODDIN ALAHYARI BEIG, ERIC JOHNSEN, Univ of Michigan - Ann Arbor — Cavitation occurs in a wide range of hydraulic applications, and one of its most important consequences is structural damage to neighboring surfaces following repeated bubble collapse. A number of studies have been conducted to predict the pressures produced by the collapse of a single bubble. However, the collapse of multiple bubbles is known to lead to enhanced collapse pressures. In this study, we quantify the effects of bubble-bubble interactions on the bubble dynamics and pressures/temperatures produced by the collapse of a pair of bubbles near a rigid surface. For this purpose, we use an in-house, high-order accurate shock- and interface-capturing method to solve the 3D compressible Navier-Stokes equations for gas/liquid flows. The non-spherical bubble dynamics are investigated and the subsequent pressure and temperature fields are characterized based on the relevant parameters entering the problem: stand-off distance, geometrical configuration, collapse strength. We demonstrate that bubble-bubble interactions amplify/reduce pressures and temperatures produced at the collapse, and increase the non-sphericity of the bubbles and the collapse time, depending on the flow parameters.

6:03PM E21.00003 Uniting the family of jets of single cavitation bubbles¹, OUTI SUPPONEN, Ecole polytechnique fédérale de Lausanne, DANAIL OBRESCHKOW, University of Western Australia, MARC TINGUELY, PHILIPPE KOBEL, NICOLAS DORSAZ, MOHAMED FARHAT, Ecole polytechnique fédérale de Lausanne — Micro-jets are high-speed liquid jets that are produced when a cavitation bubble experiences a non-spherical collapse. Such jets may be driven by any anisotropy in the liquid, such as those induced by near surfaces, gravity, pressure gradients in flows or shock waves. Here we unify this diverse family of micro-jets by describing their dynamics with a single anisotropy parameter $\zeta \geq 0$ that represents a dimensionless version of the liquid momentum at the collapse point. We observe, experimentally and numerically, that the dimensionless jet parameters describing the jet speed, jet impact time, bubble displacement, bubble volume at jet impact and vapor-jet volume, all reduce to functions of ζ . Consequently, a measurement of a single parameter, such as the bubble displacement, may be used to estimate any other parameter, such as the jet speed. The jets are phenomenologically categorized into three visually distinct regimes: weak jets that hardly pierce the bubble, intermediate jets that pierce the bubble late during the collapse, and strong jets that pierce the bubble at an early stage of the collapse. In the weak and intermediate jet regimes, that is, when $\zeta < 0.1$, the dimensionless jet parameters scale as simple power laws of ζ independently of the jet driver.

¹Swiss National Science Foundation, University of Western Australia Research Collaboration Award, European Space Agency

6:16PM E21.00004 Dynamics of a vapor nanobubble collapsing near a solid boundary¹, CARLO MASSIMO CACIOLA, FRANCESCO MAGALETTI, MIRKO GALLO, GIORGIA SINIBALDI, LUCA MARINO, Dipartimento di Ingegneria Meccanica e Aerospaziale, Università di Roma La Sapienza — The collapse of a nano-bubble near a solid wall is addressed exploiting a phase field model (Magaletti et al., Phys. Rev. Lett. 2015). The dynamics, triggered by a shock wave in the liquid, is explored for different conditions. It is characterized by a sequence of collapses and rebounds of the pure vapor bubble accompanied by the emission of shock waves in the liquid. The shocks are reflected by the wall to impinge back on the re-expanding bubble. The presence of the wall and the impinging shock wave break the symmetry of the system, leading, for sufficiently strong intensity of the incoming shock wave, to the poration of the bubble and the formation of an annular structure and a liquid jet. Intense peaks of pressure and temperatures are found also at the wall, confirming that the strong localized loading combined with the jet impinging the wall is a potential source of substrate damage induced by the cavitation (Magaletti et al., J. Multiphase Flows 2016). Comparison of the numerical results with recent experiments on the collapse of a Laser induced cavitation bubble will also be discussed.

¹The research leading to these results has received funding from the European Research Council under the European Unions Seventh Framework Programme (FP7/2007-2013)/ERC project No. [339446], BIC - Following Bubbles from Inception to Cavitation

6:29PM E21.00005 Ultrafast cavitation induced by an X-ray laser in water drops, CLAUDIO STAN, PHILIP WILLMOTT, SLAC National Accelerator Laboratory, HOWARD STONE, Princeton University, Department of Mechanical and Aerospace Engineering, JASON KOGLIN, MENGNING LIANG, ANDREW AQUILA, JOSEPH ROBINSON, KARL GUMERLOCK, GABRIEL BLAJ, RAYMOND SIERRA, SEBASTIEN BOUTET, SERGE GUILLET, ROBIN CURTIS, SHARON VETTER, HENRIK LOOS, JAMES TURNER, FRANZ-JOSEF DECKER, SLAC National Accelerator Laboratory — Cavitation in pure water is determined by an intrinsic heterogeneous cavitation mechanism, which prevents in general the experimental generation of large tensions (negative pressures) in bulk liquid water. We developed an ultrafast decompression technique, based on the reflection of shock waves generated by an X-ray laser inside liquid drops, to stretch liquids to large negative pressures in a few nanoseconds. Using this method, we observed cavitation in liquid water at pressures below -100 MPa. These large tensions exceed significantly those achieved previously, mainly due to the ultrafast decompression. The decompression induced by shock waves generated by an X-ray laser is rapid enough to continue to stretch the liquid phase after the heterogeneous cavitation occurs in water, despite the rapid growth of cavitation nanobubbles. We developed a nucleation-and-growth hydrodynamic cavitation model that explains our results and estimates the concentration of heterogeneous cavitation nuclei in water.

6:42PM E21.00006 Bursting of a bubble confined in between two plates¹, MAYUKO MURANO, NATSUKI KIMOTO, KO OKUMURA, Ochanomizu University — Rupture of liquid thin films, driven by surface tension, has been studied for more than a century [1,2]. As for a three-dimensional film, it is reported theoretically and numerically that the film edge, regardless of its viscosity, eventually attains the Taylor-Culick velocity predicted on the basis of inviscid theory [3]. Here, we studied the bursting of films in confined geometries. The confined film bursts at a speed three to five orders of magnitude slower, which means that the bursting dynamics is completely different from that of three dimensional films. We quantify the shape of rims and velocity field inside the film via strongly magnified high-speed images of bursting tips, and provide physical insights on the bursting dynamics by using a simple model. Under a certain condition, the confined film bursts like a three-dimensional film. We will also discuss the transition of the bursting dynamics from three-dimensional to confined one. [1] L. Rayleigh, Nature 44 (1891) [2] F. E. Culick, J. Appl. Phys. 31 (1960) [3] N. Savva and J. W. M. Bush, J. Fluid Mech. 626 (2009)

¹This research was partly supported by ImPACT Program of Council for Science, Technology and Innovation (Cabinet Office, Government of Japan).

Sunday, November 20, 2016 5:37PM - 6:55PM – Session E22 Nano Flows: Basic Flow Physics E141/142 - Nicolas Hadjiconstantinou, MIT

5:37PM E22.00001 A Reynolds lubrication equation for dense fluids valid beyond Navier-Stokes¹, NISHA CHANDRAMOORTHY, NICOLAS HADJICONSTANTINOU, MIT — Based on an approach for describing wave propagation in narrow channels, originally attributed to Lamb, we develop a method for extending the Reynolds Lubrication approximation to small scales for which the Navier-Stokes constitutive closures fail. The basic idea behind this approach is that the Reynolds equation is an averaged description of mass conservation and thus does not involve spatially resolved flow profiles in the transverse (gap) direction. In other words, the constitutive information required is significantly simpler and is limited to the local flowrate as a function of the gap height. Such a constitutive relation is significantly easier to obtain by experiments and/or off-line molecular simulations of pressure driven flow in constant height channels in which other control parameters of the flow rate are held constant. Using this constitutive equation results in a Reynolds-type equation that enables continuum modelling of lubrication problems at any lengthscale. The proposed methodology is demonstrated and validated for a nanoscale lubrication problem by comparison to Molecular Dynamics simulations.

¹This work was supported by the MIT Lubrication Symposium

5:50PM E22.00002 Harvesting liquid from unsaturated vapor nanoflows induced by capillary condensation, OLIVIER VINCENT, BASTIEN MARGUET, ABRAHAM STROOCK, Cornell University — A vapor, even subsaturated, can spontaneously form liquid in nanoscale spaces. This process, known as capillary condensation, plays a fundamental role in various contexts, such as the formation of clouds or the dynamics of hydrocarbons in the geological subsurface. However, large uncertainties remain on the thermodynamics and fluid mechanics of the phenomenon, due to experimental challenges as well as outstanding questions about the validity of macroscale physics at the nanometer scale. We studied experimentally the spatio-temporal dynamics of water condensation in a model nanoporous medium (pore radius ~ 2 nm), taking advantage of the color change of the material upon hydration. We found that at low relative humidities ($< 60\%$ RH), capillary condensation progressed in a diffusive fashion, while it occurred through a well-defined capillary-viscous imbibition front at $> 60\%$ RH, driven by a balance between the pore capillary pressure and the condensation stress given by Kelvin equation. Further analyzing the imbibition dynamics as a function of saturation allowed us to extract detailed information about the physics of nano-confined fluids. Our results suggest excellent extension of macroscale fluid dynamics and thermodynamics even in pores ~ 10 molecules in diameter.

6:03PM E22.00003 Thermophoresis around dimer of gold spheres for enhancement of trapping range of plasmonic tweezers, TETSUYA OGINO, KYOSUKE YASUDA, KEN YAMAMOTO, MASAHIRO MOTOSUKE, Tokyo University of Science — Trapping of nanomaterials by an optical radiation pressure can be effectively performed by combining an enhanced localized electric field on plasmonic structures due to surface plasmon resonance. Since an effective trapping area is limited in nanoscale, target transportation to the area from far would gain the trapping performance. This study investigates a potential of the nanomaterials transportation dispersed in the bulk liquid into the trapping area by thermophoresis. We performed numerical simulation of the electromagnetic field around a gold nanosphere dimer whose diameters are 20 - 300 nm and gap width is 1 - 50 nm as a plasmonic structure under irradiation of plane electromagnetic wave with the finite element method. Then the corresponding temperature field generated by photothermal heating was obtained. 1 to 100 nm polystyrene spheres (PS) in water was considered. The trapping force, which includes optical gradient force, thermophoretic force, and drag force exerting on the PS, was calculated, and the range for the trapping was investigated. The results indicates that the overall trapping range strongly depends on the thermophoretic property, Soret coefficient. The possibility of wide-ranged nanomaterial trap by controlling the temperature field was confirmed.

6:16PM E22.00004 Capillary freezing of ionic liquids confined between metallic interfaces, JEAN COMTET, ANTOINE NIGUS, VOJTECH KAISER, LYDRIC BOCQUET, ALESSANDRO SIRIA, Ecole Normale Supérieure — Using a quartz tuning fork based AFM, we investigate the behavior of ionic liquids under confinement. Using nanorheological measurements, we show that nanometric confinements can lead to solidification and capillary freezing of the ionic liquid. We find that the critical confinement at which the liquid-solid transition occurs depends strongly on the bulk electronic properties of the confining substrate, with stronger effects observed for more metallic surfaces. This behavior is rationalized on the basis of a Gibbs-Thompson framework for the shift of the freezing transition, taking into account surface energies with the imperfect metal at the level of a Thomas-Fermi model. Finally, we show that capillary freezing can also be tuned by electrifying the confining interfaces.

6:29PM E22.00005 From the Nano- to the Macroscale – Bridging Scales for the Moving Contact Line Problem, ANDREAS NOLD, Imperial College London, DAVID SIBLEY, Loughborough University, BENJAMIN GODDARD, University of Edinburgh, SERAFIM KALLIADASIS, Imperial College London, COMPLEX MULTISCALE SYSTEMS TEAM — The moving contact line problem remains an unsolved fundamental problem in fluid mechanics. At the heart of the problem is its multiscale nature: a nanoscale region close to the solid boundary where the continuum hypothesis breaks down, must be resolved before effective macroscale parameters such as contact line friction and slip can be obtained. To capture nanoscale properties very close to the contact line and to establish a link to the macroscale behaviour, we employ classical density-functional theory (DFT) [1,2], in combination with extended Navier-Stokes-like equations. Using simple models for viscosity and slip at the wall, we compare our computations with the Molecular Kinetic Theory, by extracting the contact line friction, depending on the imposed temperature of the fluid [3]. A key fluid property captured by DFT is the fluid layering at the wall-fluid interface, which has a large effect on the shearing properties of a fluid. To capture this crucial property, we propose an anisotropic model for the viscosity, which also allows us to scrutinize the effect of fluid layering on contact line friction. [1] Math. Model. Nat. Phenom. 10 111 (2015) [2] Phys. Fluids 26 072001 (2014) [3] A. Nold, PhD Thesis, Imperial College London (2016)

6:42PM E22.00006 Thermophoresis of water droplets inside carbon nanotubes¹, HARVEY ZAMBRANO, University of Concepcion, JH WALTHER², Technical University of Denmark, ELTON OYARZUA, ANDRES ROJANO, University of Concepcion — Carbon Nanotubes (CNTs) offer unique possibilities as fluid conduits with applications ranging from lab on a chip devices to encapsulation media for drug delivery. CNTs feature high mechanical strength, chemical and thermal stability and biocompatibility therefore they are promising candidates for nanodevice fabrication. Thermal gradients have been proposed as mechanism to drive particles, fullerenes and droplets inside CNTs. Here, by conducting Molecular Dynamics (MD) simulations, we study thermophoresis of water droplets inside CNTs. We systematically change the size of the droplets, the axial thermal gradient and CNT chirality. We find that the droplet motion in the armchair CNTs exhibits two clearly delimited stages, a regime wherein the droplet is accelerated and subsequently, a regime wherein the droplet moves with constant velocity. Inside the zig-zag CNTs, the droplet accelerates during a very short time and then it moves with constant velocity. We compute the net force during the droplet acceleration and find a correlation between the droplet acceleration and the magnitude of the thermal gradient without any dependence on the droplet size. Moreover, we conduct velocity constrained MD simulations to determine the friction and thermophoretic forces acting on the droplet.

¹We acknowledge partial funding from FONDECYT through the project No. 11130559 and from VRID Universidad de Concepcion.

²also at Chair of Computational Science, ETH Zurich

Sunday, November 20, 2016 5:37PM - 6:55PM –
Session E25 Microscale Flows: Locomotion E145 - Shreyas Mandre, Brown University

5:37PM E25.00001 Around a camphoric-acid boat, is the surfactant adsorbed on to the interface or dissolved in the bulk?, SHREYAS MANDRE, Brown University, SATHISH AKELLA, DHIRAJ SINGH, Okinawa Institute of Science and Technology Graduate University, RAVI SINGH, Brown University, MAHESH BANDI, Okinawa Institute of Science and Technology Graduate University — A camphoric-acid boat (c-boat for short), a cylindrical gel tablet infused with camphoric acid, moves spontaneously when placed on an air-water interface. This system is a classic example of propulsion driven by Marangoni forces. Despite rich history on particles propelled by Marangoni forces, including contributions by figures such as Benjamin Franklin, Alessandro Volta, and Giovanni Venturi, the underlying fluid dynamics remains poorly understood. A key missing piece is the nature of the surfactant; in our case, the question is whether the camphoric acid is dissolved in the bulk or adsorbed on to the interface. We gain insight into this piece by holding the c-boat stationary and measuring the surrounding axisymmetric flow velocity to a precision needed to distinguish between the two possibilities. For soluble surfactants, it is known that the velocity field decays as $r^{-2/3}$, where r is the distance from the center of the c-boat. Whereas, for surfactant adsorbed on to the air-water interface, we derive that the surrounding velocity fields decays as $r^{-3/5}$. Based on our measurements we deduce that, even though soluble in water, the Marangoni flow results from a layer of camphoric acid adsorbed to the air-water interface.

5:50PM E25.00002 Steering artificial nanoscale swimmers using teardrop shaped posts¹, MEGAN DAVIES WYKES, XIAO ZHONG, TAKIJI ADACHI, New York University, YANPENG LIU, Beijing University of Aeronautics and Astronautics, JIAJUN TONG, LEIF RISTROPH, MICHAEL WARD, New York University, JUN ZHANG, New York University/NYU-Shanghai, MICHAEL SHELLEY, New York University — Microorganisms use various strategies to bias their swimming to achieve long-time directed motion against a flow, against gravity, or up a chemical gradient. To make use of artificial swimmers for transporting cargo, to separate swimming particles from diffusing ones, or to concentrate a solution of motile particles, ways of steering such swimmers are required. We make use of the attraction of artificial bi-metallic swimmers to vertical walls to direct their long-time motion. We will describe how these swimmers are attracted to the surface of teardrop-shaped posts and leave preferentially at regions of higher curvature. We use this understanding to interpret their behavior when interacting with arrays of teardrop-shaped posts.

¹This work was supported partially by the Materials Research Science and Engineering Center (MRSEC) program of the National Science Foundation under Award Number DMR-1420073.

6:03PM E25.00003 Convective self-propulsion of chemically active particles, OLEG SHKLYAEV, HENRY SHUM, ANNA BALAZS, University of Pittsburgh — A mechanism of particle self-propulsion activated by transduction of chemical energy into convective motion of fluid that drags microscale particles is proposed. The convection is generated by an active spherical particle located on the bottom of a microchannel and coated with a catalyst that decomposes reagent dissolved in the solution into less dense products and gives rise to a buoyancy force. The symmetry of the flow generated around the active particle can be broken if a passive spherical particle, which does not produce the flow, is present in the vicinity of the first one. The generated flow drags the passive particle toward the active one along the bottom wall until they form a dimer. The resulting asymmetric fluid flow, which is generated by only one of the particles, imposes a different drag on the different sides on the dimer. The net force causes the dimer to translate along the bottom wall. By varying numbers of active and passive particles, as well as their positions within a group, one can control the structure of the generated convective flow and, therefore, design clusters with different mobile properties. The proposed mechanism can be harnessed to transport cargo in microchannels.

6:16PM E25.00004 Wall-induced self-diffusiophoresis of active isotropic colloids, EHUD YARIV, Technion — While chemically-active homogeneous spherical particles do not undergo self-diffusiophoresis in free solution, they may do so when suspended in the vicinity of a solid boundary. We explore this possibility using a first-order kinetic model of solute absorption, where the relative magnitude of reaction to diffusion is characterized by the Damkohler number Da. When the particle is remote from the wall, it is repelled from it with a velocity that scales inversely with the square of distance. The opposite extreme, when the ratio δ of separation distance

to particle size is small, results in the anomalous scaling $\delta^{\frac{\sqrt{1+2Da}-1}{2}}$ of the solute concentration in the narrow gap separating the particle and wall. This irrational power may only be obtained by asymptotic matching with solute transport outside the gap. For $Da < 4$ the self-propulsion speed possesses the same scaling, being set by the large pressures forming in the gap through a lubrication-type mechanism. For $Da > 4$ the particle velocity is $O(\delta)$, set by the flow in the region outside outside the gap. Solute advection is subdominant to diffusion in both the remote and near-contact limits, and accordingly affects neither the above scaling nor the resulting approximations.

6:29PM E25.00005 Traction Reveals Nature of Wall-effects for Microswimmers near Boundaries, XINHUI SHEN, MARCOS MARCOS¹, Nanyang Technological University, HENRY C. FU, University of Utah — The flow field due to a low-Reynolds number swimmer swimming in the vicinity of a planar boundary has been frequently studied using image systems of flow singularities. However, it can also be represented by an integral of the traction on the boundary. We show that examining the traction pattern on the boundary caused by a swimmer provides insights into determining when far-field multipole models are accurate. We investigate the instantaneous swimming velocity and traction induced by a three-sphere swimmer placed near a solid planar wall quantitatively. When the swimmer is far from the wall, the effect of the wall can be accurately represented using the image of a force dipole, but near the wall, a system of singularities reflecting the internal structure of the swimmer is necessary. We find that the instantaneous traction reflects these limits, and furthermore can be used to determine the range of validity of the far-field approximation. We also investigate the time-averaged velocity and traction. In the far field, the image of a quadrupole accurately represents the effect of the boundary, and the traction is also quadrupolar, while in the near field, the traction shows the influence of the internal structure of the swimmer.

¹The full name of this author is Marcos.

6:42PM E25.00006 Maximizing the propulsive thrust of a driven filament at low Reynolds number through non-uniform flexibility, ZHIWEI PENG, GWYNNE ELFRING, University of British Columbia, ON SHUN PAK, Santa Clara University — In the low Reynolds number regime, periodic boundary actuation of a rigid filament leads to a reciprocal motion and hence produces zero propulsive thrust. Introducing flexibility into the filament results in filament deformation enabling propulsion in the absence of inertia. For a given actuation frequency and filament length, an optimal bending stiffness of the filament can be determined to produce the largest propulsive force. However, the possibility of further improving the propulsion by allowing variable flexibility along the filament remains largely unexplored. In this work, we perform a theoretical investigation of flexibility distributions that can maximize propulsive thrust of a driven filament at low Reynolds number.

Sunday, November 20, 2016 5:37PM - 6:42PM –

Session E26 Reacting Flows: Multiphase E146 - Ann Karagozian, University of California, Los Angeles

5:37PM E26.00001 Periodic Partial Extinction Regime in Acoustically Coupled Fuel Droplet Combustion¹, MIGUEL PLASCENCIA QUIROZ, JOHN BENNEWITZ, ANDRES VARGAS, HYUNG SUB SIM, OWEN SMITH, ANN KARAGOZIAN, UCLA — This experimental study investigates the response of burning liquid fuel droplets exposed to standing acoustic waves, extending prior studies quantifying mean and temporal flame response to moderate acoustic excitation². This investigation explores alternative fuels exposed to a range of acoustic forcing conditions (frequencies and amplitudes), with a focus on ethanol and JP-8. Three fundamental flame regimes are observed: sustained oscillatory combustion, periodic partial extinction and reignition (PPER), and full extinction. Phase-locked OH* chemiluminescence imaging and local temporal pressure measurements allow quantification of the combustion-acoustic coupling through the local Rayleigh index G . As expected, PPER produces negative G values, despite having clear flame oscillations. PPER is observed to occur at low-frequency, high amplitude excitation, where the acoustic time scales are large compared with kinetic/reaction times scales for diffusion-limited combustion processes. These quantitative differences in behavior are determined to depend on localized fluid mechanical strain created by the acoustic excitation as well as reaction kinetics.

¹Supported by AFOSR Grant FA9550-15-1-0339

²Sevilla, et al., Comb. Flame 161, pp. 1604-1619, 2014

5:50PM E26.00002 Effects of Particle Additives on Acoustically Coupled Fuel Droplet Combustion¹, HYUNG SUB SIM, MIGUEL PLASCENCIA QUIROZ, ANDRES VARGAS, JOHN BENNEWITZ, OWEN SMITH, ANN KARAGOZIAN, UCLA — Addition of nanoscale particulates to liquid hydrocarbon fuels is suggested to have numerous benefits for combustion systems, although aggregation of metal nanoparticles can produce deleterious effects. The present experiments explore the effect of nano Aluminum (nAl) additives on the combustion of single liquid fuel droplets, with and without exposure of the droplets to standing acoustic waves. Building on prior studies², the present experiments quantify variations in the burning rate constant K for ethanol droplets with increasing concentrations of nAl in a quiescent environment. Burning fuel droplets that are continuously fed via a capillary as well as suspended (non-fed) droplets are examined. Nano Al is observed to create ejections of both particles and vapor toward the end of the burning period for non-fed droplets; this phenomenon is delayed when the droplet is replenished via continuous fuel delivery. Yet for the majority of conditions explored, increasing concentrations of nAl tend to reduce K . When ethanol droplets with nAl are exposed to standing waves, acoustic perturbations appear to delay particulate agglomeration, sustaining combustion for a longer period of time and increasing K .

¹Supported by AFOSR Grant FA9550-15-1-0339

²Sevilla, et al., Comb. Flame 161, pp. 1604-1619, 2014

6:03PM E26.00003 Characterization of transcritical and supercritical droplet vaporization regimes using computations¹, PAVAN GOVINDARAJU, DANIEL BANUTI, PETER MA, MURALIKRISHNA RAJU, MATTHIAS IHME, Stanford University — Mixing of liquid fuel with ambient gases plays an important role in engine combustion efficiency and emissions. The situation of cold liquid fuel injected into gas at very high pressure and temperature conditions creates special challenges for prediction of combustion characteristics. Among them, the important question is how the interface between cold liquid fuel and hot ambient responds at the pressures and temperatures specific to engines. The presentation will elaborate on the computational procedure used to simulate the injection of n-dodecane into N_2 and comparing interface characteristics with experimental data. This requires robust tools for predicting droplet evaporation, real fluid properties and molecular-dynamic simulations for validating surface tension characteristics. The effect of pyrolysis in the gas phase is considered and the influence of surface tension is examined. Finally, a comparison between theory, experiments and simulations is presented for transition in vaporization regimes.

¹NASA

6:16PM E26.00004 Computational Analysis of End-of-Injection Transients and Combustion Recessions, DORRIN JARRAHBASHI, SAYOP KIM, BENJAMIN W. KNOX, CAROLINE L. GENZALE, Georgia Institute of Technology, GEORGIA INSTITUTE OF TECHNOLOGY TEAM — Mixing and combustion of ECN Spray A after end of injection are modeled with different chemical kinetics models to evaluate the impact of mechanism formulation and low-temperature chemistry on predictions of combustion recession. Simulations qualitatively agreed with the past experimental observations of combustion recession. Simulations with the Cai mechanism show second-stage ignition in distinct regions near the nozzle, initially spatially separated from the lifted diffusion flame, but then rapidly merge with flame. By contrast, the Yao mechanism fails to predict sufficient low-temperature chemistry in mixtures upstream of the diffusion flame and combustion recession. The effects of the shape and duration of the EOI transient on the entrainment wave near the nozzle, the likelihood of combustion recession, and the spatiotemporal development of mixing and chemistry in near-nozzle mixtures are also investigated. With a more rapid ramp-down injection profile, a weaker combustion recession occurs. For extremely fast ramp-down, the entrainment flux varies rapidly near the nozzle and over-leaning of the mixture completely suppresses combustion recession. For a slower ramp-down profile complete combustion recession back toward the nozzle is observed.

6:29PM E26.00005 Numerical Simulation of Condensation of Sulfuric Acid and Water in a Large Two-stroke Marine Diesel Engine, J. H. WALTHER, N. KARVOUNIS, K. M. PANG, Technical University of Denmark — We present results from computational fluid dynamics simulations of the condensation of sulfuric acid (H_2SO_4) and water (H_2O) in a large two-stroke marine diesel engine. The model uses a reduced n-heptane skeletal chemical mechanism coupled with a sulfur subset to simulate the combustion process and the formation of SO_x and H_2SO_4 . Condensation is modeled using a fluid film model coupled with the Eulerian in-cylinder gas phase. The fluid film condensation model is validated against both experimental and numerical results. The engine simulations reveal that the fluid film has a significant effect on the sulfuric acid gas phase. A linear correlation is found between the fuel sulfur content and the sulfuric acid condensation rate. The initial in-cylinder water content is found not to affect the sulfuric acid condensation but it has a high impact on water condensation. The scavenging pressure level shows an inverse correlation between pressure and condensation rate due to change in the flame propagation speed. Finally, increasing the cylinder liner temperature significantly decreases water condensation but has a negligible influence on the condensation of sulfuric acid.

Sunday, November 20, 2016 5:37PM - 6:55PM –
Session E27 CFD: High Performance Computing E147-148 - Michael Dodd, University of Washington

5:37PM E27.00001 PSH3D fast Poisson solver for petascale DNS, DARREN ADAMS, University of Illinois, NCSA, MICHAEL DODD, ANTONINO FERRANTE, University of Washington, Seattle — Direct numerical simulation (DNS) of high Reynolds number, $Re \geq O(10^5)$, turbulent flows requires computational meshes $\geq O(10^{12})$ grid points, and, thus, the use of petascale supercomputers. DNS often requires the solution of a Helmholtz (or Poisson) equation for pressure, which constitutes the bottleneck of the solver. We have developed a parallel solver of the Helmholtz equation in 3D, PSH3D. The numerical method underlying PSH3D combines a parallel 2D Fast Fourier transform in two spatial directions, and a parallel linear solver in the third direction. For computational meshes up to 8192^3 grid points, our numerical results show that PSH3D scales up to at least 262k cores of Cray XT5 (Blue Waters). PSH3D has a peak performance $6\times$ faster than 3D FFT-based methods when used with the ‘partial-global’ optimization, and for a 8192^3 mesh solves the Poisson equation in 1 sec using 128k cores. Also, we have verified that the use of PSH3D with the ‘partial-global’ optimization in our DNS solver does not reduce the accuracy of the numerical solution of the incompressible Navier-Stokes equations.

5:50PM E27.00002 Towards mitigating Asynchronous Computing effects in largescale simulations, ANKITA MITTAL, SHARATH GIRIMAJI, Texas AM Univ — Synchronization of processing elements (PEs) in massively parallel simulations has shown to significantly affect scalability of scientific applications. Relaxing this synchronization among PEs (asynchronous) conserves the stability condition but severely affects the accuracy reducing the average error to first-order regardless of the original scheme. At the present time, several approaches are under consideration to improve the order of asynchronous computations. In this work, we propose to modify the original governing equation to obtain a Proxy-Equation which when solved asynchronously recovers the order of accuracy of the original numerical scheme. Performing 1D simulations for the Advection Diffusion Equation, we observe that the wave speed and the viscosity must be increased in the vicinity of PE boundaries in order to counteract the effect of asynchrony. In addition to recovering accuracy, this method shows lower magnitudes of average error when compared to existing asynchronytolerant methods. Similar results are also presented for a 1D viscous Burgers equation.

6:03PM E27.00003 Unstable phenomena of low speed compressible natural convection with open boundaries by multi-GPU implementation., WEI-HSIANG WANG, RIKEN Adv Inst for Computational Science, WU-SHUNG FU, Nation Chiao Tung University, Taiwan, MAKOTO TSUBOKURA, Kobe University, Japan — Unstable phenomena of low speed compressible natural convection are investigated numerically. Geometry contains parallel square plates or single heated plate with open boundaries is taken into consideration. Numerical methods of the Roe scheme, preconditioning and dual time stepping matching the DP-LUR method are used for low speed compressible flow. The absorbing boundary condition and modified LODI method is adopted to solve open boundary problems. High performance parallel computation is achieved by multi-GPU implementation with CUDA platform. The effects of natural convection by isothermal plates facing upwards in air is then carried out by the methods mentioned above. Unstable behaviors appeared upon certain Rayleigh number with characteristic length respect to the width of plates or height between plates.

6:16PM E27.00004 CODE BLUE: Three dimensional massively-parallel simulation of multi-scale configurations¹, DAMIR JURIC, LIMSI, CNRS, France, LYSE KAHOUADJI, Imperial College London, JALEL CHERGUI, LIMSI, CNRS, France, SEUNGWON SHIN, Hongik University, South Korea, RICHARD CRASTER, OMAR MATAR, Imperial College London — We present recent progress on BLUE, a solver for massively parallel simulations of fully three-dimensional multiphase flows which runs on a variety of computer architectures from laptops to supercomputers and on 131072 threads or more (limited only by the availability to us of more threads). The code is wholly written in Fortran 2003 and uses a domain decomposition strategy for parallelization with MPI. The fluid interface solver is based on a parallel implementation of a hybrid Front Tracking/Level Set method designed to handle highly deforming interfaces with complex topology changes. We developed parallel GMRES and multigrid iterative solvers suited to the linear systems arising from the implicit solution for the fluid velocities and pressure in the presence of strong density and viscosity discontinuities across fluid phases. Particular attention is drawn to the details and performance of the parallel Multigrid solver.

¹EPSRC UK Programme Grant MEMPHIS (EP/K003976/1)

6:29PM E27.00005 Discrete kinetic and lattice Boltzmann formulations for reaction cross-diffusion systems and their hyperbolic extensions in chemotaxis, PAUL DELLAR, University of Oxford — We present discrete kinetic and lattice Boltzmann formulations for reaction cross-diffusion systems, as commonly used to model microbiological chemotaxis and macroscopic predator-prey interactions, and their hyperbolic extensions with fluid-like persistence terms. For example, the canonical Patlak–Keller–Segal model for chemotaxis involves a flux of cells up the gradient of a chemical secreted by the cells, in addition to the usual down-gradient diffusive fluxes. Existing lattice Boltzmann approaches for such systems use finite difference approximations to compute the flux of cells due to the chemical gradient. The resulting coupling between, and necessary synchronisation of the evolution of, adjacent grid points greatly complicates boundary conditions, and efficient implementation on graphical processing units (GPUs). We present a kinetic formulation using cross-collisions between bases of moments for the two sets of distribution functions to couple the fluxes of the two species, from which we construct lattice Boltzmann algorithms using second-order Strang splitting. We demonstrate an efficient GPU implementation, and verify second-order spatial convergence towards spectral solutions for benchmark problems such as the finite-time blow-up in the Patlak–Keller–Segal model.

6:42PM E27.00006 LES of a bluff-body stabilized premixed flame using discontinuous Galerkin scheme, YU LV, MATTHIAS IHME, Stanford University, STANFORD UNIVERSITY TEAM — This talk focuses on the development of a high-order discontinuous Galerkin (DG) method for application to chemically reacting flows. To enable these simulations, several algorithmic aspects are addressed, including the time-integration of multi-step chemical reactions, the incorporation of detailed thermo-viscous transport properties, and the stabilization of high-order solution representation. This DG solver is applied in implicit LES of a turbulent bluff-body stabilized propane/air premixed flame. The simulation results for cold-flow and reacting conditions are reported and compared to experimental data.

Sunday, November 20, 2016 5:37PM - 6:55PM –

Session E28 Particle-laden Flows: Non-spherical Particles F149 - Greg Voth, Wesleyan University

5:37PM E28.00001 Measurements of inertial range scaling in rotations of rigid particles in turbulence¹, GREG VOTH, BRENDAN COLE, STEFAN KRAMEL, Wesleyan University — We measure the rotation rate of 3D-printed particles with sizes spanning the inertial range in a turbulent flow between oscillating grids. Tetrads, composed of four slender rods in tetrahedral symmetry, and triads, three slender rods in triangular planar symmetry, are tracked in a flow with $R_\lambda = 156$ and $R_\lambda = 214$ using four high-speed cameras. Tetrads rotate like spheres and triads rotate like disks. Measurements of tetrads' solid body 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. We observe Kolmogorov scaling for the mean square solid body rotation rate of tetrads, $\langle \Omega^2 \rangle \propto r^{-4/3}$, extending earlier work on rods by Parsa and Voth (PRL 2014) to particles that do not experience preferential orientation. The theory is extended to higher moments and intermittent scaling, but experiments do not yet resolve intermittency corrections. For triads, the solid-body rotation rate is preferentially aligned with the particle orientation, and we find that the preferential alignment of these large planar particles is quite different than was previously observed for small tracer disks.

¹Supported by NSF grant DMR-1508575

5:50PM E28.00002 Orientation statistics of non-spherical particles sedimenting in turbulence, STEFAN KRAMEL, LYDIA TIERNEY, WYATT REES, GREG A. VOTH, Wesleyan University, UDAYSHANKAR MENON, ANUBHAB ROY, DONALD L. KOCH, Cornell University — We study the sedimentation of non-spherical particles in turbulence. The particle orientation is determined by a competition between inertial torques causing a preferential alignment and turbulence randomizing the orientation. The relative importance is quantified by a settling number S_F defined as the ratio of the tumbling-rate from inertial torques and from turbulence. The experiments focus on the orientation statistics of particles formed from several slender arms, including fibers and particles with three arms in planar symmetry (triads), which allows us to study alignment of both fibers and disk-like particles. We measure the time-resolved 3D orientations of the particles along with the fluid velocity field around them in a vertical water tunnel. An active jet array with 40 individually controllable jets enables us to adjust the turbulence intensity and observe the transition from strongly aligned particles to randomized orientations as S_F is decreased. Results are compared to simulations and theory based on slender body theory.

6:03PM E28.00003 Rotation of non-spherical particles in turbulence of neutral buoyancy and large size¹, EVAN VARIANO, NIMISH PUJARA, ANKUR BORDOLOI, UC Berkeley — We explore the ways in which particle size and shape affects particle rotation. We consider rotation in the laboratory frame and in the particles local frame of reference. We use homogeneous isotropic turbulence whose Taylor microscale is similar to the particle size. Our data, as well as a new analysis of Stokes numbers, suggests that particle inertia prevents alignment with turbulent structures. The total angular velocity is nearly shape-independent, an observation which has been presented before. We offer an explanation for this behavior, based on an analysis of Jeffery's equations for inertia-free point ellipsoids.

¹Support from National Science Foundation and Army Research Office

6:16PM E28.00004 Drag reduction in turbulent channel laden with finite-size oblate spheroids.¹, MEHDI NIAZI ARDEKANI, KTH Royal Institute of Technology, PEDRO COSTA COLLABORATION, WIM-PAUL BREUGEM COLLABORATION, FRANCESCO PICANO COLLABORATION, LUCA BRANDT COLLABORATION — Suspensions of oblate rigid particles in a turbulent plane channel flow are investigated for different values of the particle volume fraction. We perform direct numerical simulations (DNS), using a direct-forcing immersed boundary method to account for the particle-fluid interactions, combined with a soft-sphere collision model and lubrication corrections for short-range particle-particle and particle-wall interactions. We show a clear drag reduction and turbulence attenuation in flows laden with oblate spheroids, both with respect to the single phase turbulent flow and to suspensions of rigid spheres. We explain the drag reduction by the lack of the particle layer at the wall, observed before for spherical particles. In addition, the special shape of the oblate particles creates a tendency to stay parallel to the wall in its vicinity, forming a shield of particles that prevents strong fluctuations in the outer layer to reach the wall and vice versa. Detailed statistics of the fluid and particle phase will be presented at the conference to explain the observed drag reduction.

¹Supported by the European Research Council Grant No. ERC-2013-CoG-616186, TRITOS. The authors acknowledge computer time provided by SNIC (Swedish National Infrastructure for Computing) and the support from the COST Action MP1305: Flowing matter

6:29PM E28.00005 Computational and Experimental Study of Spherocylinder Particles in Fluidized Beds¹, VINAY MAHAJAN, Delft Univ of Tech, HANS KUIPERS, Eindhoven Univ of Tech, JOHAN PADDING, Delft Univ of Tech, MULTIPHASE REACTORS GROUP, TU EINDHOVEN TEAM — Non-spherical particle flows are often encountered in fluidized process equipment. A coupled computational fluid dynamics(CFD) and discrete element method(DEM) approach has been extensively applied in recent years to study these flows at the particle scale. However, most of these studies focus on spherical particles while in reality, the constituent particles are seldom spherical. Particle shape can significantly affect the hydrodynamical response in fluidized beds. The drag force acting on a non-spherical particle can vary considerably with particle shape, orientation of the particle, Reynolds number and packing fraction. In this work, a CFD-DEM approach has been extended to model a lab scale quasi-2D fluidized bed of spherocylinder (rod-like) particles. These particles can be classified as Geldart D particles and have an aspect ratio of 4. Numerical results for the pressure drop, bed height and solid circulation patterns are compared with results from a complementary laboratory experiment. We also present results on particle orientations close to the confining walls, which provides interesting insight regarding the particle alignment. Thus the capability of the CFD-DEM approach to efficiently account for global bed dynamics in fluidized bed of rod-like particle is demonstrated.

¹This research work is funded by ERC grant

6:42PM E28.00006 On the preferential sampling of helicity by isotropic helicoids¹, LUCA BIFERALE, Department of Physics, University of Rome Tor Vergata and INFN , KRISTIAN GUSTAVSSON, Department of Physics, University of Rome Tor Vergata and Dept. Physics University of Gothenburg, RICCARDO SCATAMACCIA, Department of Physics, University of Rome Tor Vergata and INFN — We present a theoretical and numerical study on the motion of isotropic helicoids in complex flows. These are particles whose motion is invariant under rotations but not under mirror reflections of the particle. This is the simplest, yet unexplored, extension of the much studied case of small spherical particles. We show that heavy isotropic helicoids, due to the coupling between translational and rotational degrees of freedom, preferentially sample different helical regions in laminar or chaotic advecting flows. This opens the way to control and engineer particles able to track complex flow structures with potential applications to microfluidics and turbulence.

¹ERC AdG Grant NewTURB no. 339032

Sunday, November 20, 2016 5:37PM - 6:42PM –

Session E29 CFD: Lattice Boltzmann Methods F150 - Joseph Kocheemoolayil, STC Corp. in NASA Ames Research Center

5:37PM E29.00001 How good is the Lattice Boltzmann method?, JOSEPH KOCHEEMOOLAYIL, STC Corp. in NASA Ames Research Center, MICHAEL BARAD, CETIN KIRIS, NASA Ames Research Center — Conflicting opinions exist in literature regarding how efficient the lattice Boltzmann method is relative to high-order finite difference approximations of the Navier-Stokes equations on Cartesian meshes, especially at high Mach numbers. We address the question from the pragmatic viewpoint of a practitioner. Dispersion, dissipation and aliasing errors of various lattice Boltzmann models are systematically quantified. The number of floating point operations and memory required for a desired accuracy level are carefully compared for the two numerical methods. Turbulent kinetic energy budgets for several standard test cases such as the decaying Taylor-Green vortex problem are used to evaluate how effective the stabilization mechanisms necessary for lattice Boltzmann method at high Reynolds numbers are. Detailed comments regarding the cyclomatic complexity of the underlying software, scalability of the underlying algorithm on state-of-the-art high-performance computing platforms and wall clock times and relative accuracy for selected simulations conducted using the two approaches are also made.

5:50PM E29.00002 Lattice Boltzmann Models for Flows with Axial Symmetry and Mass and Momentum Sources without Cubic Velocity Errors

, FARZANEH HAJABDOLLAHI, KANNAN PREMNATH, University of Colorado Denver — Three-dimensional flows with axial symmetry arise in numerous applications, which can be solved more efficiently on a two-dimensional Cartesian coordinate system with appropriate source terms. Lattice Boltzmann (LB) method is promising recent development in CFD. However, existing LB models are not Galilean invariant (GI) due to the degeneracy of the resulting third-order longitudinal moments, which leads to cubic velocity truncation errors. This can lead to anisotropic stress tensor with velocity-dependent viscosities and numerical instability under high shear even with finer grids. In this investigation, we develop a new radius-weighted LB model for axisymmetric flows using a non-orthogonal moment basis with an extended moment equilibria and restore GI on standard lattices. Also, as another related example, we consider flows with mass and momentum sources, which are important in various contexts, including acoustics, reacting flows and flows undergoing phase change. To handle such problems, we develop a new LB model by incorporating sources in its zeroth and first order moments, with extended moment equilibria to eliminate the cubic velocity errors. Both the resulting new models will be validated for benchmark problems.

6:03PM E29.00003 An Improved Lattice Boltzmann Model for Non-Newtonian Flows with Applications to Solid-Fluid Interactions in External Flows

, SAAD ADAM, KANNAN PREMNATH, University of Colorado Denver — Fluid mechanics of non-Newtonian fluids, which arise in numerous settings, are characterized by non-linear constitutive models that pose certain unique challenges for computational methods. Here, we consider the lattice Boltzmann method (LBM), which offers some computational advantages due to its kinetic basis and its simpler stream-and-collide procedure enabling efficient simulations. However, further improvements are necessary to improve its numerical stability and accuracy for computations involving broader parameter ranges. Hence, in this study, we extend the cascaded LBM formulation by modifying its moment equilibria and relaxation parameters to handle a variety of non-Newtonian constitutive equations, including power-law and Bingham fluids, with improved stability. In addition, we include corrections to the moment equilibria to obtain an inertial frame invariant scheme without cubic-velocity defects. After performing its validation study for various benchmark flows, we study the physics of non-Newtonian flow over pairs of circular and square cylinders in a tandem arrangement, especially the wake structure interactions and their effects on resulting forces in each cylinder, and elucidate the effect of the various characteristic parameters.

6:16PM E29.00004 The flow around a flapping foil¹, FRANCISCO MANDUJANO, CARLOS MALAGA, Physics Department, School of Science, Universidad Nacional Autonoma de Mexico — The flow around a two-dimensional flapping foil immersed in a uniform stream is studied numerically using a Lattice-Boltzmann model, for Reynolds numbers between 100 and 250, and flapping Strouhal numbers between 0.01 and 0.6. The computation of the hydrodynamic force on the foil is related to the the wake structure. When the foil's is fixed in space, numerical results suggest a relation between drag coefficient behaviour and the flapping frequency which determines the transition from the von Kármán to the inverted von Kármán wake. When the foil is free of translational motion up-stream swimming at constant speed is observed at certain values of the flapping Strouhal.

¹This work was partially supported by UNAM-DGAPA-PAPIIT grant number IN115316

6:29PM E29.00005 LBM-DSMC Hybrid Method for Complex Out-of-Equilibrium

Flows , GIANLUCA DI STASO, HERMAN J.H. CLERCX, Eindhoven Univ of Tech, SAURO SUCCI, IAC-CNR, Rome, FEDERICO TOSCHI, Eindhoven Univ of Tech — Many complex flows are characterized by the simultaneous presence of a range of non-equilibrium and rarefaction effects in different regions of the flow field. We recently developed a Direct Simulation Monte Carlo (DSMC)-Lattice Boltzmann Method (LBM) hybrid scheme, based on domain decomposition technique and on Grads moments method, able to accurately and efficiently simulate such flows. While DSMC is employed to compute the flow field only where large non-equilibrium effects are present, the more computationally efficient LBM is employed wherever the non-equilibrium effects can be dealt with perturbatively, i.e. according to Navier-Stokes hydrodynamics. Here we present the results on the application of the hybrid method to complex three-dimensional flows, in particular to the flow around a microsphere and through a disk-shaped expansion channel. The solutions provided by the hybrid method are compared against full DSMC simulations and the computational gain guaranteed by the application of the hybrid method over the full DSMC is also demonstrated.

Sunday, November 20, 2016 5:37PM - 6:55PM –

Session E30 Granular Flows: Mixing, Segregation and Separation F151 - Richard Lueptow, Northwestern University

5:37PM E30.00001 Mixing dynamics of cutting and shuffling for granular materials¹

, RICHARD M. LUEPTOW, ZAFIR ZAMAN, MENGQI YU, PAUL P. PARK, JÚLIO M. OTTINO, PAUL B. UMBANHOWAR, Northwestern University — Chaotic dynamics has been shown to play a major role in fluid mixing, but the study of its relevance to granular flows has only recently begun. We utilize a simple 3D geometry, a half-filled spherical tumbler rotated alternately by $\leq \pi/2$ about two perpendicular horizontal axes, to develop a dynamical systems framework for granular mixing and non-mixing. In these systems, mixing can only occur during flow (from stretching due to shear and from collisional diffusion in the flowing layer) or by material separation intrinsic to the rotation protocol resulting from cutting and shuffling. In X-ray subsurface visualization experiments, surprisingly persistent ($O(100)$ iterations) non-mixing elliptical regions and larger non-mixing barriers occur as predicted by both a continuum model and an idealized theoretical model (with an infinitely thin flowing layer) based on the mathematics of piece-wise isometries. In these models, the stretching in the flowing layer vanishes as the flowing layer thickness decreases to reveal the underlying skeleton of the mixing. This dynamical systems perspective provides insight into mixing and non-mixing phenomena unique to granular materials.

¹Funded by NSF Grant CMMI-1435065.

5:50PM E30.00002 Stratification of size-bidisperse granular mixtures in a quasi-2D bounded heap with periodic flow modulation¹ , HONGYI XIAO, ZHEKAI DENG, PAUL UMBANHOWAR, JULIO OTTINO, RICHARD LUEPTOW, Northwestern University — Segregation of disperse granular materials in unsteady flows is ubiquitous in nature and industry, yet remains largely unexplored. In this study, unsteady flows are generated by feeding size-bidisperse granular mixtures onto a quasi-2D bounded heap using alternating feed rates, which results in stratified layers of large and small particles. The mechanism of stratification is investigated in detail using Discrete Element Method (DEM) simulations of the flow. During the transition from the slow to the fast feed rate, a segregating wedge propagates downstream and forms a large particle layer extending upstream. During the opposite transition, upstream segregated small particles relax downstream and form a small particle layer extending downstream. The transient kinematics from DEM simulations are quantified and used to inform a time-dependent continuum model that captures the interplay of advection, diffusion, and segregation in the flowing layer. The continuum model reproduces the principle characteristics of the stratification patterns observed in experiments and simulations.

¹Funded by NSF Grant CBET-1511450

6:03PM E30.00003 Streamwise transport in multi-component granular flows , CHRIS JOHNSON, Univ of Manchester, ANDREW HOGG, JEREMY PHILLIPS, Univ of Bristol — Thin free-surface avalanches of granular material occur widely in nature and industry. In these contexts the flows often contain particles of a wide range of sizes, which separate from one another due to particle size segregation. This segregation is troublesome in industry (where a well-mixed state is usually desired) and is an important mechanism for determining the runout and morphology of natural avalanches and debris flows. In this talk we develop a model for the spatial and temporal evolution of the particle size distribution in a granular flow, due to particle segregation, diffusion and advection processes. We use asymptotic solutions of this model to formulate equations governing the depth-integrated particle size distribution. These equations naturally extend the shallow-water models often used to predict the dynamics of monodisperse avalanches. Our modelling shows that particles in granular avalanches may strongly segregate in the direction of flow, even when the segregation is relatively weak compared to diffusive mixing and the avalanche is nearly homogeneous throughout its depth. We demonstrate this surprising phenomenon through comparison with discrete particle simulations.

6:16PM E30.00004 Application of methods for quantifying maximum potential segregation and actual segregation risk to design of powder blends , DAVID GOLDFARB, STEPHEN CONWAY, MICHAEL GENTZLER, Merck Research Labs — As described in a recent publication (Gentzler, Tardos, Michaels, *Powder Technology*, 2015), the tendency of pharmaceutical powders to demix due to segregation can threaten the content uniformity of solid dosage forms. Using the methodology established in this publication, examples of analysis and optimization of pharmaceutical formulations to evaluate the potential for segregation during formulation and reduce the risk of content uniformity issues upon scale-up are provided. Modification to active components and excipient properties are considered and a systematic risk assessment approach for multi-component blends emerges. Use of the measurements to understand excipient and raw material sensitivities in lieu of pilot and commercial-scale production tests is described. This approach has the potential for being readily applied to the study of the segregation risk potential outside the pharmaceutical industry.

6:29PM E30.00005 Segregating photoelastic particles in free-surface granular flows. , AMALIA L. THOMAS, NATHALIE M. VRIEND, University of Cambridge — We experimentally investigate bimodal avalanches of photoelastic discs between two narrow side-walls. We visualize the physical phenomena that occur during segregation and quantify the dynamic appearance of force chains within the bulk of the flow from fringe patterns using photoelastic theory. The photoelastic technique has been used in granular research for almost half a century and has been applied in a variety of quasi-steady systems. We have now adapted the technique to perform well within dynamic granular flows where collisions are short-lived and force chains are formed and broken continuously. Our photoelastic urethane discs are cast in-house to provide high-resolution fringe patterns and a high stress-optic coefficient. In addition we carried out stress relaxation tests to study the viscoelastic properties of the photoelastic material, and measured the speed of force transmission and dampening from a moving particle onto a static chain of particles. In our avalanche experiments, we also employ particle tracking and particle velocimetry techniques to measure the general flow field within the avalanche. The overall goal of our work is to investigate and quantify the influence of the distribution of forces on the fundamental processes that drive segregation.

6:42PM E30.00006 Hydrodynamics and Segregation in Poiseuille Flow of a Binary Granular Mixture , RONAK GUPTA, MEHEBOOB ALAM, Jawaharlal Nehru Centre, Jakkur PO, Bangalore, India — Steady State profiles of hydrodynamic fields have been computed for the Poiseuille flow of a dilute bi-disperse granular mixture using DSMC (direct simulation Monte Carlo) method. The effects of mass bidispersity and inelasticity are studied and it is found that species segregation follows a non-monotonic trend with increasing mass-ratio if the particles are inelastic. Mixture velocity shows a similar trend. Nonequipartition of granular temperature is expectedly enhanced with increasing mass-ratio and inelasticity, but is additionally a strong function of Knudsen number. Effort is made to compare simulation results with a continuum theory for dilute binary granular mixtures, with the aim being to check if theory is able to predict the novel segregation tendencies uncovered in DSMC simulations.

Sunday, November 20, 2016 5:37PM - 6:42PM –
Session E31 Experimental Techniques - Reactive Flows F152 - Douglas Kelley, University of Rochester

5:37PM E31.00001 Front tracking for characterizing and quantifying reactive mixing

, DOUGLAS KELLEY, THOMAS NEVINS, University of Rochester — Mixing in industrial chemical reactors involves complicated interactions between advection, reaction, and diffusion that are difficult to simulate or measure in detail. However, in large-Damköhler-number systems which show sharp fronts between reacted and unreacted regions, reactor dynamics might be more simply and usefully characterized in terms of the reaction fronts themselves. In fact, prior work has already shown that the reaction rate and material diffusivity can be calculated directly if front speed and front thickness are known. We have developed methods to optically track reaction fronts, measuring their speed and thickness throughout space and time. We will present such measurements in both simulation and experiment, consider their statistics, and discuss future efforts to characterize and quantify mixing in chemical reactors.

5:50PM E31.00002 Spontaneous Raman Scattering Measurements of Vibrational Non-Equilibrium in High-Speed Jets¹

, HEATH REISING, TIMOTHY HALLER, NOEL CLEMENS, PHILIP VARGHESE, The University of Texas at Austin — Vibrational non-equilibrium is detected and quantified in a high-speed jet using spontaneous Raman scattering. The non-equilibrium is induced by rapid mixing of the different temperature streams of the jet and coflow which are approximately 500 K and 1000 K, respectively. Simultaneous measurements of vibrational and rotational temperatures are made using fits of time-averaged high-resolution Stokes spectra of both N₂ and O₂ to high fidelity models of the spectrum. Independent measurements of these two species temperatures show good agreement in rotational temperature while the vibrational temperatures show only N₂ to have a strong non-equilibrium. This suggests that vibrational energy transfer between these two molecules is very inefficient at these conditions. Work is being conducted to extend the technique to single-shot measurements by employing a multiple-pass cell to increase the incident laser fluence in the measurement volume. This new capability will allow for statistics of vibrational temperature to be quantified. The instantaneous nature of the measurements will also allow the technique to be applied in regions of large temperature fluctuations, such as the base of a lifted turbulent jet flame, where time-average measurements are not valid.

¹This work was supported by funding from the Air Force Office of Scientific Research

6:03PM E31.00003 Application of computer vision in studying fire plume behavior of tilting flames

, AMIRHESSAM AMINFAR, JEANETTE COBIAN IIGUEZ , STEPHANIE PHAM , Univ of California - Riverside, JOEY CHONG, GLORIA BURKE, DAVID WEISE, USDA Forest Service, MARKO PRINCEVAC, Univ of California - Riverside — With the development in computer sciences especially in the field of computer vision, image processing has become an inevitable part of flow visualization. Computer vision can be used to visualize flow structure and to quantify its properties. We used a computer vision algorithm to study fire plume tilting when the fire is interacting with a solid wall. As the fire propagates to the wall the amount of air available for the fire to consume will decrease on the wall side. Therefore, the fire will start tilting towards the wall. Aspen wood was used for the fuel source and various configurations of the fuel were investigated. The plume behavior was captured using a digital camera. In the post processing, the flames were isolated from the image by using edge detection technics, making it possible to develop an algorithm to calculate flame height and flame orientation. Moreover, by using an optical flow algorithm we were able to calculate the speed associated with the edges of the flame which is related to the flame propagation speed and effective vertical velocity of the flame. The results demonstrated that as the size of the flame was increasing, the flames started tilting towards the wall. Leading to the conclusion that there should be a critical area of fire in which the flames start to tilt. Also, the algorithm made it possible to calculate a critical distance in which the flame will start orienting towards the wall

6:16PM E31.00004 Flame structure and chemiluminescence in premixed flames

, JOSE GRANA-OTERO, SIAMAK MAHMOUDI, University of Kentucky — The quantitative use of chemiluminescence requires the knowledge of the relationship between the concentration of excited species with flame properties such as the equivalency ratio, the burning rate or the heat release rate. With the aim of rigorously finding from first principles these relations we have analyzed, numerically and analytically, the distribution of the excited species OH* and CH* in steady hydrogen and methane planar premixed flames. Their mass fractions turn out to be extremely small; thus, a kinetic mechanism describing their dynamics in the flame can be obtained by simply adding the kinetic mechanism describing the excitation and de-excitation to the mechanism of the base flame. Due also to their small concentrations, the excited species are in steady state, facilitating a simple analytical description. The analyses show that OH*, both in hydrogen and methane flames, can be found broadly distributed downstream the preheat region, in a three-layer structure that is analytically described. The distribution of CH* is much simpler, being always in equilibrium with CH, whose concentration is in turn proportional to that of CH₄. As a result, CH* is confined to the methane consumption layer in lean flames, but broadly distributed in rich flames.

6:29PM E31.00005 Composition Independent Thermometry in Gaseous Combustion Using Spectral Lineshape Information¹

, DOMINIC ZELENAK, Graduate Student, North Carolina State University — Temperature is an important thermochemical property that holds the key to revealing 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 laser thermometry techniques that do not require prior knowledge of the local composition. Multiple methods for obtaining temperature are demonstrated, which make use of the spectral line broadening of an absorbing species (Kr) seeded into the flow. These techniques involve extracting the Doppler broadening from the Voight profile and utilizing compositional scaling of collisional broadening and shift to determine temperature. Doppler broadening-temperature scaling of two photon Kr-PLIF is provided. Lean-premixed and diffusion jet flames of CH₄ will serve as the test bed for experimentation, and validation of the two methods will be made using the corresponding temperature determined from Rayleigh scattering imaging with adiabatic mixing and unity Lewis number assumptions. A ratiometric dual lineshape thermometry method for turbulent flames will also be introduced.

¹AFOSR grant FA9550-16-1-0190 with Dr. Chiping Li as Program Manager

Sunday, November 20, 2016 5:37PM - 6:55PM –
Session E32 Rough Wall Turbulent Boundary Layers - III Oregon Ballroom 201 - Mitul Luhar,
University of Southern California

5:37PM E32.00001 A terrain-following modeling of wave boundary layers¹ , JIE YU,

Stony Brook University — Applying the method of conformal transformation, we put forward a terrain-following modeling approach for Stokes boundary layer flows. This complements the recent new development of the exact Floquet theory of water waves, that gives a complete basis of solutions for time harmonic potential flows over an arbitrarily periodic seabed. The theory applies for any given frequency, including the resonant waves. For a non-steep seabed profile, but not necessarily small undulation height comparing with the water depth, the analytical solutions can be obtained for the boundary layer velocities, bed shear stresses and rate of viscous dissipation, explicitly showing the variations both across the boundary layer and along the bed. For a relatively steep bed profile, a remedy is proposed that allows the velocity profiles to be locally determined across the boundary layer avoiding solving the 2-D differential equation for the vorticity. The modeling methodology is presented here for a constant viscosity, including the case of constant eddy viscosity, but can be extended to the case of variable eddy viscosity to improve turbulence modeling.

¹US National Science Foundation

5:50PM E32.00002 Energy amplification in turbulent flows over complex walls , MITUL LUHAR, University of Southern California

Many boundary layer flows in natural and manmade systems are characterized by the presence of complex walls (e.g. porous, rough, or patterned) that can substantially alter the near-wall turbulence. For example, the streaks and streamwise vortices prevalent in smooth-walled flows are often replaced by structures resembling Kelvin-Helmholtz vortices in flows over porous media and vegetation canopies. While stability analyses can reproduce some of these observations, they are limited in their ability to generate predictions for spectra and coherent structure in fully turbulent flows. The present effort seeks to address this limitation by extending the resolvent formulation to account for complex walls. Under the resolvent formulation, the turbulent velocity field is expressed as a linear superposition of propagating modes, identified via a gain-based decomposition of the Navier-Stokes equations. The presence of the complex substrate is modeled as a distributed body force, which alters the gain (i.e. energy amplification) and structure of the modes. Preliminary results show that this approach reproduces key observations from previous simulations and experiments of flow over porous media, vegetation canopies, as well as ripples with minimal computation.

6:03PM E32.00003 Experimental investigation of the effect of a singly-periodic perturbation on a rough-wall turbulent boundary layer¹ , JONATHAN MORGAN, BEVERLEY MCKEON,

Caltech — A 3D printed surface which is singly periodic in the streamwise and spanwise directions was placed in a turbulent boundary layer facility. The zero-pressure gradient boundary layer which developed over this singly periodic roughness was characterized with hot-wire anemometry. Compared to a canonical smooth-wall flow, the periodic roughness introduces through its boundary condition a static, singly-periodic fluctuation in mean velocity. From this linear introduction of a single-mode perturbation into the flow, the nonlinear effects of the perturbation on travelling modes can be tracked through statistics, spectra, and mean flow quantities to establish a link between roughness geometry and flow physics. Variation of the velocity power spectrum within the rough boundary layer as well as variation between smooth- and rough-wall boundary layers show the effect of the roughness to be concentrated at wavenumbers which correspond to the roughness wavelength. The effects of the roughness ultimately manifest nonlinearly as an altered Reynolds-stress field which changes the mean velocity profile of the boundary layer. Implications for more general roughness are discussed.

¹The authors gratefully acknowledge the support of the Office of Naval Research, grant N000141310739

6:16PM E32.00004 Study of transition mechanisms induced by an array of roughness elements¹ , PRAKASH SHRESTHA, GRAHAM V. CANDLER, Univ of Minn - Minneapolis, COMPUTATIONAL HYPERSONICS RESEARCH LAB TEAM

We study transition mechanisms of a Mach 5.92 laminar boundary layer due to an array of prismatic roughness elements using large-scale direct numerical simulations (DNS). We simulate a boundary layer tripped by arrays of different numbers of roughness elements, corresponding to experiments conducted at the Texas A & M University Actively Controlled Experimental (ACE) facility. We obtain solutions using a high-order, low-dissipation scheme for the convection terms in the Navier-Stokes equations. We perform separate 2D and 3D simulations. Flow parallel inflow acoustic disturbances are implemented in the 2D domain. We then interpolate spectral content obtained at 30 mm from the leading edge of the 2D domain to the inflow of the 3D domain. In the 3D domain, we compute optimal modes of pressure using dynamic mode decomposition (DMD). Using sparsity-promoting dynamic mode decomposition (SPDMD), we select the dominant modes to study the transition mechanisms. Recirculating vortices upstream and separated shear layers downstream of the roughness elements are observed to be the most dominant modes of transition. We compare streamwise mean mass flux and energy spectral densities at different streamwise locations to validate our simulations.

¹Office of Naval Research

6:29PM E32.00005 Analysis of turbulent heat and momentum transfer in a transitionally rough turbulent boundary layer , ALI DOOSTTALAB, SURANGA DHARMARATHNE, Texas Tech University, MURAT TUTKUN, IFE, Department of Process and Fluid Flow Technology, Norway, RONALD ADRIAN, Arizona State University, LUCIANO CASTILLO, Texas Tech University

A zero-pressure-gradient (ZPG) turbulent boundary layer over a transitionally rough surface is studied using direct numerical simulation (DNS). The rough surface is modeled as 24-grit sandpaper which corresponds to $k^+ \approx 11$, where k^+ is roughness height. Reynolds number based on momentum thickness is approximately 2400. The walls are isothermal and turbulent flow Prandtl number is 0.71. We simulate temperature as passive scalar. We compute the inner product of net turbulent force ($d(u_i u_i)/dx_i$) and net turbulent heat flux ($d(u_i \theta)/dx_i$) in order to investigate (i) the correlation between these vectorial quantities, (II) size of the projection of these fields on each other and (III) alignment of momentum and heat flux. The inner product in rough case results in larger projection and better alignment. In addition, our study on the vortices shows that surface roughness promotes production of vortical structures which affects the thermal transport near the wall.

6:42PM E32.00006 Internal interfaces over a step change in wall roughness¹ , R. JASON

HEARST, Univ of Southampton, RONALD E. HANSON, Applied Fluid Dynamics Inc., BHARATHRAM GANAPATHISUBRAMANI, Univ of Southampton — When flow encounters a step change in wall roughness, an internal boundary layer is formed near the wall. This internal layer grows with streamwise position and eventually dominates the entire boundary layer, returning it to equilibrium with the new boundary condition. It is well established that a canonical turbulent boundary layer is populated by patches of high and low velocity, referred to as uniform momentum zones (UMZs). The UMZs are separated by shear events. In this study, the characteristics of UMZs are examined as the flow transitions from one wall condition to another. Planar particle image velocimetry measurements were performed over both a rough-to-smooth ($R \rightarrow S$) and a smooth-to-rough ($S \rightarrow R$) step change in wall roughness. For the flow over a $R \rightarrow S$ change, the maximum wall normal position of the high momentum UMZs that populate the outer region of the boundary layer moves outward towards the free-stream, while the low momentum UMZs that are situated near the wall move closer to the wall. The talk will discuss the implication of these results as well as the results for the flow over a $S \rightarrow R$ step change in wall roughness.

¹ERC (Grant no. 277472), EPSRC (Grant no. EP/I037717/1), NSERC (post-doctoral fellowships)

Sunday, November 20, 2016 5:37PM - 6:55PM –

Session E33 Turbulence: Particle-laden Flows Oregon Ballroom 202 - Perry Johnson, Johns Hopkins University

5:37PM E33.00001 Restricted Euler dynamics along trajectories of small inertial particles in turbulence¹ , PERRY JOHNSON, CHARLES MENEVEAU, Johns Hopkins University — The fate of small particles in turbulent flows depends strongly on the surrounding fluid's velocity gradient properties such as rotation and strain-rates. For non-inertial (fluid) particles, the Restricted Euler model provides a simple, low-dimensional dynamical system representation of Lagrangian evolution of velocity gradients in fluid turbulence, at least for short times. Here we derive a new restricted Euler dynamical system for the velocity gradient evolution of inertial particles such as solid particles in a gas or droplets and bubbles in turbulent liquid flows. The model is derived in the limit of small (sub Kolmogorov scale) particles and low Stokes number. The system exhibits interesting fixed points, stability and invariant properties. Comparisons with data from Direct Numerical Simulations show that the model predicts realistic trends such as the tendency of increased straining over rotation along heavy particle trajectories and, for light particles such as bubbles, the tendency of severely reduced self-stretching of strain-rate.

¹ Supported by a National Science Foundation Graduate Research Fellowship Program under Grant No. DGE-1232825 and by a grant from The Gulf of Mexico Research Initiative.

5:50PM E33.00002 Simulation of finite size particles in turbulent flows using entropic lattice boltzmann method¹ , ABHINEET GUPTA, HERMAN J. H. CLERCX, FEDERICO TOSCHI, TU Eindhoven — Particle-laden turbulent flows occur in variety of industrial applications. While the numerical simulation of such flows has seen significant advances in recent years, it still remains a challenging problem. Many studies investigated the rheology of dense suspensions in laminar flows as well as the dynamics of point-particles in turbulence. Here we will present results on the development of numerical methods, based on the Lattice Boltzmann method, suitable for the study of suspensions of finite-size particles under turbulent flow conditions and with varying geometrical complexity. The turbulent flow is modeled by an entropic lattice Boltzmann method, and the interaction between particles and carrier fluid is modeled using bounce back rule. Direct contact and lubrication force models for particle-particle interactions and particle-wall interaction are taken into account to allow for a full four-way coupled interaction. The accuracy and robustness of the method is discussed by validating velocity profile in turbulent pipe flow, sedimentation velocity of spheres in duct flow and resistance functions of approaching particles. Results show that the velocity profiles and turbulence statistics can be significantly altered by the presence of the dispersed solid phase.

¹The author is supported by Shell-NWO computational sciences for energy research (CSER) Grant (12CSER034).

6:03PM E33.00003 Multiscale geometrical Lagrangian statistics: scale-dependent curvature and torsion angles in particle-laden turbulent flows¹ , KAI SCHNEIDER, I2M-CNRS, Centre de Mathématiques et d'Informatique, Aix-Marseille Université, Marseille, France, BENJAMIN KADOCH, IUSTI-CNRS, Aix-Marseille Université, Marseille, France, MAXIME BASSENNE, MAHDI ESMAILY-MOGHADAM, Center for Turbulence Research, Stanford University, Stanford, CA, USA, MARIE FARGE, LMD-IPSL-CNRS, Ecole Normale Supérieure, Paris, France, WOUTER BOS, LMFA-CNRS, Ecole Centrale de Lyon, Université de Lyon, Ecully, France — We present multiscale statistics of particle trajectories in isotropic turbulence and compare the behaviour of fluid and inertial particles. The directional change of inertial particles is quantified by considering the curvature angle for different time increments. Distinct scaling behaviors of the mean angle are observed for short, intermediate and long time lags. We also introduce the scale-dependent torsion angle, which quantifies the directional change of particles moving out of the plane. The influence of the Stokes and Reynolds numbers on the mean angles and on the probability distributions are analyzed. Finally, we assess the impact of LES and particle SGS modeling on those statistics.

¹MF and KS thankfully acknowledge financial support from CTR, Stanford.

6:16PM E33.00004 Effect of particle inertia on fluid turbulence in gas-solid disperse flow¹ , YOICHI MITO, Kitami Institute of Technology — The effect of particle inertia on the fluid turbulence in gas-solid disperse flow through a vertical channel has been examined by using a direct numerical simulation, to calculate the gas velocities seen by the particles, and a simplified non-stationary flow model, in which a uniform distribution of solid spheres of density ratio of 1000 are added into the fully-developed turbulent gas flow in an infinitely wide channel. The gas flow is driven downward with a constant pressure gradient. The frictional Reynolds number defined with the frictional velocity before the addition of particles, v_0^* , is 150. The feedback forces are calculated using a point force method. Particle diameters of 0.95, 1.3 and 1.9, which are made dimensionless with v_0^* and the kinematic viscosity, and volume fractions, ranging from 1×10^{-4} to 2×10^{-3} , in addition to the one-way coupling cases, are considered. Gravitational effect is not clearly seen where the fluid turbulence is damped by feedback effect. Gas flow rate increases with the decrease in particle inertia, that causes the increase in feedback force. Fluid turbulence decreases with the increase in particle inertia, that causes the increase in diffusivity of feedback force and of fluid turbulence.

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

6:29PM E33.00005 Statistics of relative velocities of heavy particles in turbulence¹

, DHRUBADITYA MITRA, AKSHAY BHATNAGAR, NORDITA, Stockholm, KRISTIAN GUSTAFSSON, BERNHARD MEHLIG, Dept. of Physics, University of Gothenburg — We consider heavy, inertial, passive, particles in homogeneous and isotropic turbulent flows. Using direct numerical simulations we study the statistics of relatives velocities of identical particles. We calculate the p^{th} order moments $m_p(R)$ of the collision velocity as a function of particle size. We find that, in agreement with theory, the moments show bifractal scaling behavior confirming the effects of caustics. We also compute the joint probability distribution functions (PDFs) of relative velocity and separation between two closeby particles and compare with theory at intermediate and large Stokes numbers.

¹Supported by the Knut and Alice Wallenberg Foundations under project "Bottlenecks for particle growth in turbulent aerosols" (Dnr. KAW 2014.0048)

6:42PM E33.00006 How long do particles spend in vortical regions in turbulent

flows?¹ , AKSHAY BHATNAGAR, NORDITA, Stockholm, ANUPAM GUPTA, University of Rome "Tor Vergata", Rome, Italy., DHRUBADITYA MITRA, NORDITA, Stockholm, RAHUL PANDIT, Centre for Condensed Matter Theory, Department of Physics, Indian Institute of Science, Bangalore, PRASAD PERLEKAR, TIFR Centre for Interdisciplinary Sciences, 21 Brundavan Colony, Narsingi, Hyderabad — We consider passive, heavy, inertial, particles (HIP) in three-dimensional, homogeneous, and isotropic turbulence. Whether a particle is in a vortical regions or not is determined by the two invariants of the (flow) velocity gradient matrix , Q and R , at the position of the particle. Using direct numerical simulations, we calculate the probability distribution functions (PDFs) of the first-passage-time of a tracer or a HIP in a vortical region. The corresponding PDF in two dimensions is known to show power-law tail. In three dimensions we find that the PDF possesses exponential tail with a characteristic time of the order of large-eddy-turnover-time of the flow.

¹partially supported by the Knut and Alice Wallenberg Foundation (DM and AB) under project "Bottlenecks for particle growth in turbulent aerosols" (Dnr. KAW 2014.0048)

Sunday, November 20, 2016 5:37PM - 6:55PM –

Session E34 Turbulence: Multiphase Flow Oregon Ballroom 203 - Don Bergstrom, University of Saskatchewan

5:37PM E34.00001 Numerical simulation of turbulent slurry flows , MOHAMMAD REZA HAGH-

GOO, Mechanical engineering Department, University of Saskatchewan, REYMOND J. SPITERI, Computer Science Department, University of Saskatchewan, DONLAD J. BERGSTROM, Mechanical engineering Department, University of Saskatchewan — Slurry flows, i.e., the flow of an agglomeration of liquid and particles, are widely employed in many industrial applications, such as hydro-transport systems, pharmaceutical batch crystallizers, and wastewater disposal. Although there are numerous studies available in the literature on turbulent gas-particle flows, the hydrodynamics of turbulent liquid-particle flows has received much less attention. In particular, the fluid-phase turbulence modulation due to the particle fluctuating motion is not yet well understood and remains challenging to model. This study reports the results of a numerical simulation of a vertically oriented slurry pipe flow using a two-fluid model based on the kinetic theory of granular flows. The particle stress model also includes the effects of frictional contact. Different turbulence modulation models are considered, and their capability to capture the characteristic features of the turbulent flow is assessed. The model predictions are validated against published experimental data and demonstrate the significant effect of the particles on the fluid-phase turbulence.

5:50PM E34.00002 Micro-bubbles and Micro-particles are Not Faithful Tracers of Turbulent Acceleration , CHAO SUN, Center for Combustion Energy and Department of Thermal Engineering, Tsinghua

University, China., VARGHESE MATHAI, Physics of Fluids Group, University of Twente, The Netherlands., ENRICO CALZAVARINI, Univ. Lille, CNRS, FRE 3723, LML, Laboratoire de Mecanique de Lille, F 59000 Lille, France., JON BRONS, DETLEF LOHSE, Physics of Fluids Group, University of Twente, The Netherlands. — We report on the Lagrangian statistics of acceleration of small (sub-Kolmogorov) bubbles and tracer particles with Stokes number $St \ll 1$ in turbulent flow. At decreasing Reynolds number, the bubble accelerations show deviations from that of tracer particles, i.e. they deviate from the Heisenberg-Yaglom prediction and show a quicker decorrelation despite their small size and minute St . Using direct numerical simulations, we show that these effects arise due to the drift of these particles through the turbulent flow. We theoretically predict this gravity-driven effect for developed isotropic turbulence, with the ratio of Stokes to Froude number or equivalently the particle drift-velocity governing the enhancement of acceleration variance and the reductions in correlation time and intermittency. Our predictions are in good agreement with experimental and numerical results. The present findings are relevant to a range of scenarios encompassing tiny bubbles and droplets that drift through the turbulent oceans and the atmosphere.

6:03PM E34.00003 Coalescence and Break-up of large, deformable droplets with different viscosities in turbulent channel flow¹ , ALESSIO ROCCON, University of Udine, FRANCESCO ZONTA, TU Wien, ALFREDO SOLDATI, Tu Wien, University of Udine — The dynamics of large, deformable droplets released in a turbulent channel flow is numerically analyzed. Pseudo-Spectral direct numerical simulations are based on the resolution of the coupled Navier-Stokes and Cahn-Hilliard equations (Phase-Field Model). The droplets have the same density but different viscosity compared to the surrounding fluid. We first focus on droplets coalescence and break-up rate. Two different dynamic are observed, depending on the Weber number We , (which measures the ratio between the inertial forces and the surface tension forces) and the viscosity ratio λ , (ratio between the viscosity of the drop and the continuous phase). For small We , droplets are only slightly deformed and their viscosity does not influence much the coalescence/break-up rate. For larger We , droplets are deformable and their viscosity can significantly alter the coalescence and break-up dynamics.

¹PAR FSC 2007-2013, Regione FVG Underwater Blue Efficiency

6:16PM E34.00004 Viscosity stratified fluids in turbulent channel flow , ALFREDO SOLDATI,

University of Udine; TU Wien, SOMAYEH AHMADI, ALESSIO ROCCON, University of Udine, FRANCESCO ZONTA, TU Wien — Direct Numerical Simulation (DNS) is used to study the turbulent Poiseuille flow of two immiscible liquid layers inside a rectangular channel. A thin liquid layer (fluid 1) flows on top of a thick liquid layer (fluid 2), such that their thickness ratio is $h_1/h_2 = 1/9$. The two liquid layers have the same density but different viscosities (viscosity-stratified fluids). In particular, we consider three different values of the viscosity ratio $\lambda = \nu_1/\nu_2$: $\lambda = 1$, $\lambda = 0.875$ and $\lambda = 0.75$. Numerical Simulations are based on a Phase Field method to describe the interaction between the two liquid layers. Compared with the case of a single phase flow, the presence of a liquid-liquid interface produces a remarkable turbulence modulation inside the channel, since a significant proportion of the kinetic energy is subtracted from the mean flow and converted into work to deform the interface. This induces a strong turbulence reduction in the proximity of the interface and causes a substantial increase of the volume-flowrate. These effects become more pronounced with decreasing λ .

6:29PM E34.00005 In-situ Measurements of Cloud Droplet Dynamics¹, JAN MOLACEK, GHOLAMHOSSEIN BAGHERI, Max Planck Institute for Dynamics and Self-Organisation, Goettingen, HAITAO XU, Tsinghua University, Beijing, China, EBERHARD BÖDENSCHATZ, Max Planck Institute for Dynamics and Self-Organisation, Goettingen — We present an in-situ experiment investigating the dynamics of cloud droplets and its dependence on the turbulent flow properties. This dynamics plays a major role in the rate of growth of cloud particles by coalescence and the resulting precipitation rate. The experiment takes place at a mountain research station at an altitude of 2650m, and makes use of a movable platform that can travel with the mean wind velocity over a distance of 5m and at speeds of up to 7.5m/s. Moving with mean velocity enables us to follow individual cloud particles over longer intervals, thus improving the quality of the statistics. Simultaneous measurements of other variables such as droplet size distribution and humidity fluctuations are done in order to develop a complete picture of the microphysical conditions within clouds. Preliminary results will be presented and discussed.

¹We gratefully acknowledge the generous support from the Bavarian State Ministry of the Environment and Consumer Protection via the VAO project

6:42PM E34.00006 Determining the Discharge Rate from a Submerged Oil Leaks using ROV Video and CFD study¹, PANKAJ SAHA, ORISE-Postdoc Fellow, National Energy Technology Laboratory, FRANK SHAFFER, MEHRDAD SHAHNAM, USDOE National Energy Technology Laboratory, OMER SAVAS, U.C. Berkeley, Department of Mechanical Engineering, DAVE DEVITES, OHMSETT Mar Inc, TIMOTHY STEFFECK, DOI Bureau of Safety and Environmental Enforcement — The current paper reports a technique to measure the discharge rate by analyzing the video from a Remotely Operated Vehicle (ROV). The technique uses instantaneous images from ROV video to measure the velocity of visible features (turbulent eddies) along the boundary of an oil leak jet and subsequently classical theory of turbulent jets is imposed to determine the discharge rate. The Flow Rate Technical Group (FRTG) Plume Team developed this technique that manually tracked the visible features and produced the first accurate government estimates of the oil discharge rate from the Deepwater Horizon (DWH). For practical application this approach needs automated control. Experiments were conducted at UC Berkeley and OHMSETT that recorded high speed, high resolution video of submerged dye-colored water or oil jets and subsequently, measured the velocity data employing LDA and PIV software. Numerical simulation have been carried out using experimental submerged turbulent oil jets flow conditions employing LES turbulence closure and VOF interface capturing technique in OpenFOAM solver. The CFD results captured jet spreading angle and jet structures in close agreement with the experimental observations.

¹The work was funded by NETL and DOI Bureau of Safety and Environmental Enforcement (BSEE)

Sunday, November 20, 2016 5:37PM - 6:55PM –
Session E35 Turbulence: Turbulent/Non-turbulent Interface Oregon Ballroom 204 - Ivan Marusic, University of Melbourne

5:37PM E35.00001 Scale-dependent entrainment velocity and scale-independent net entrainment in a turbulent axisymmetric jet¹, JIMMY PHILIP, University of Melbourne, DHIREN MISTRY, JAMES DAWSON, Norwegian University of Science and Technology, IVAN MARUSIC, University of Melbourne — The net entrainment in a jet is the product of the mean surface area (\bar{S}) and the mean entrainment velocity, $\bar{V} \bar{S}$, where, $\bar{V} = \alpha U_c$ with α the entrainment coefficient and U_c the mean centreline velocity. Instantaneously, however, entrainment velocity (v) at a point on the interface is the difference between the interface and the fluid velocities, and the total entrainment $\int v ds = V S$, where S is the corrugated interface surface area and V the area averaged entrainment velocity. Using time-resolved multi-scale PIV/PLIF measurements of velocity and scalar in an axisymmetric jet at $Re = 25000$, we evaluate V and S directly at the smallest resolved scales, and by filtering the data at different scales (Δ) we find their multi-scales counterparts, V_Δ and S_Δ . We show that $\bar{V} \bar{S} = V_\Delta S_\Delta = V S$, independent of the scale. Furthermore, S is found to have a fractal dimension $D_3 \approx 2.32 \pm 0.1$. Independently, we find that $V_\Delta \sim \Delta^{0.31}$, indicating increasing entrainment velocity with increasing length scale. This is consistent with a constant net entrainment across scales, and suggests α as a scale-dependent quantity.

¹Engineering and Physical Sciences Research Council (research grant no. EP/I005879/1), David Crighton Fellowship from the DAMTP, Univ of Cambridge, and the Australian Research Council.

5:50PM E35.00002 Scalar transport across the turbulent/non-turbulent interface in jets: Schmidt number effects¹, TIAGO S. SILVA, CARLOS B. DA SILVA, Inst Superior Tecnico (IST), IDMEC TEAM — The dynamics of a passive scalar field near a turbulent/non-turbulent interface (TNTI) is analysed through direct numerical simulations (DNS) of turbulent planar jets, with Reynolds numbers ranging from $142 \leq Re_\lambda \leq 246$, and Schmidt numbers from $0.07 \leq Sc \leq 7$. The steepness of the scalar gradient, as observed from conditional profiles near the TNTI, increases with the Schmidt number. Conditional scalar gradient budgets show that for low and moderate Schmidt numbers a *diffusive superlayer* emerges at the TNTI, where the scalar gradient diffusion dominates, while the production is negligible. For low Schmidt numbers the growth of the turbulent front is commanded by the molecular diffusion, whereas the scalar gradient convection is negligible.

¹The authors acknowledge the Laboratory for Advanced Computing at University of Coimbra for providing HPC, computing, consulting resources that have contributed to the research results reported within this paper. URL <http://www.lca.uc.pt>

6:03PM E35.00003 Evolution of the turbulent/non-turbulent interface in the near field of an axisymmetric jet, JAMES DAWSON, DHIREN MISTRY, Norwegian University of Science and Technology (NTNU) — We characterise the near-field evolution of an axisymmetric jet by considering the multi-scale topology of the turbulent/non-turbulent interface (TNTI). Using planar laser-induced fluorescence data from a high Reynolds number jet we implement a multi-scale methodology to evaluate the fractal dimension of the TNTI as a function of streamwise distance. We show that the streamwise evolution of the fractal dimension, D_f , of the TNTI reaches a plateau just beyond the potential core which was measured to be $x/d \approx 4.5$ in the current experiment. Downstream of the potential core we show that $D_f \approx 0.33$, which agrees with recently reported values of D_f measured in fully-developed turbulent flows, such as the far-field of a jet and in turbulent boundary layers. The onset of this fractal behaviour also coincides with evidence of flow homogeneity based on the radial auto-correlation functions of axial and radial velocity fluctuations. These results indicate that the flow-field about the TNTI beyond the potential core exhibits a hierarchy of scales (turbulent cascade) that is characteristic of fully-developed turbulence.

6:16PM E35.00004 The turbulent/non-turbulent interface in viscoelastic fluids , JOO

MELO, CARLOS B. DA SILVA, Inst Superior Tecnico (IST), IDMEC TEAM — The dynamics of the enstrophy in shear free turbulent/non-turbulent interfaces (TNTI) is analysed through direct numerical simulations (DNS) using the Finitely Extensible Nonlinear Elastic constitutive equations closed with the Peterlin approximation (FENE-P). The Reynolds number and the Deborah number of the DNS range between $116 \leq Re_\lambda \leq 182$ and $0.11 \leq De \leq 1.23$, respectively. A new term emerges in the enstrophy transport equation for viscoelastic fluids - the *viscoelastic production* - which competes with the enstrophy diffusion and enstrophy production for the build up of enstrophy near the TNTI, particularly for high Deborah numbers. While for low Deborah numbers the viscoelastic production contributes to a depletion of vorticity inside the turbulent region, this effect is reversed at the higher Deborah number configurations.

6:29PM E35.00005 The role of the scalar and enstrophy flux in entrainment processes

, DHIREN MISTRY, JAMES R. DAWSON, NTNU — Turbulent entrainment is a multi-stage, multi-scale process that describes the growth of a turbulent region of flow. Ultimately, turbulent entrainment is achieved through viscous diffusion of vorticity, and molecular diffusion in the presence of scalars, with irrotational and unmixed regions of the flow at the smallest scales. We do not fully understand how these small-scale processes are coupled to or modulated by the large-scales of turbulence. This is partly because the mean entrainment rates in turbulent shear flows can be determined by considering large-scales quantities only. We present experimental evidence that the large-scale flux of enstrophy and scalar towards the turbulent/non-turbulent interface (TNTI) coincides with a local increase in the entrainment velocity along the TNTI. This is achieved using a passive scalar ($Sc \gg 1$) to identify the TNTI, and a time-resolved interface-tracking method to measure the local entrainment velocity. Our results indicate that the both scalar and enstrophy fluxes towards the TNTI increase the vorticity and scalar gradients increasing the local rates of diffusion. These results show how local processes of small-scale diffusion are modulated by the large-scale turbulence.

6:42PM E35.00006 High Reynolds numbers scaling of the turbulent/non-turbulent interface , CARLOS BETTENCOURT DA SILVA, TIAGO S. SILVA, Inst Superior Tecnico (IST), IDMEC TEAM — The scaling of the turbulent/non-turbulent interface (TNTI) at high Reynolds numbers is assessed using new direct numerical simulations (DNS) of turbulent planar jets (PJET) and shear free turbulence (SFT) with Reynolds numbers ranging from $142 \leq Re_\lambda \leq 300$. The thickness of the turbulent sublayer (TSL), where the enstrophy production dominates over enstrophy diffusion, is of the order of the Taylor micro-scale, and is roughly one order of magnitude larger than the Kolmogorov micro-scale for these Reynolds numbers, however it clearly scales with the Kolmogorov micro-scale, at sufficiently high Reynolds numbers. It is argued the same scaling should be observed in TNTI from mixing layers, wakes and boundary layers, provided the Reynolds number is sufficiently high.

Sunday, November 20, 2016 5:37PM - 6:55PM –

Session E36 Drops: Superhydrophobic: Icophobicity Portland Ballroom 251 - Peichun Amy Tsai, University of Alberta

5:37PM E36.00001 Universal wetting transition of an evaporating water droplet on superhydrophobic surfaces , PEICHUN AMY TSAI, ADRIEN BUSSONNIER, MASOUD BIGDELI, University of Alberta, DI-YEN CHUEH, Academia Sinica, Taiwan, QINGXIA LIU, University of Alberta, PEILIN CHEN, Academia Sinica, Taiwan — An evaporating water droplet on a superhydrophobic surface undergoes a wetting transition from a heterogeneous wetting (Cassie-Baxter) to homogeneous wetting (Wenzel) state. The critical transition is manifested by a sudden decrease of contact angle, when "Fakir water drop permeates the minute hydrophobic cavities. This breakdown of superhydrophobicity would hinder various applications of self-cleaning, low-frictional, and potentially ice-phobic properties of superhydrophobic materials. In this work, we experimentally investigate such wetting transition using hydrophobic nanostructures. With a theoretical model, we find a universal criterion of the critical contact angle at the transition point. The prediction of critical contact angle, which solely depends on the geometrical parameters of the hydrophobic pillars, agree well with various data for both micro- and nano-structures.

5:50PM E36.00002 Icophobia of Leaves , H. PIROUZ KAVEHPOUR, University of California, Los Angeles, ELIKA T. SHIRAZI, Oakwood School, ELAHEH ALIZADEH-BIRJANDI, University of California, Los Angeles — Ice adhesion and excessive accumulation on exposed structures and equipment are well known to cause serious problems in cold-climate regions; therefore, the development of coatings that can resist icing can solve many challenges in various areas of industry. This work was inspired by nature and ice-resistivity and superhydrophobicity of plants leaves. Kale is an example of a plant that can be harvested in winter. It shows superhydrophobic behavior, which is normally known as an advantage for cleaning the leaves, but we were able to show that its surface structure and high contact angle of water drops on kale leaves could delay the ice formation process making it a good candidate for an ice-repellent coating. We have performed in-depth experimental analyses on how different plants can prevent icing, and contact angle measurements and scanning electron microscopy (SEM) of the leaves were taken to further mimic their surface morphology.

6:03PM E36.00003 Crystal deposition patterns from evaporating sessile drops on superhydrophobic and liquid impregnated surfaces , SAMANTHA MCBRIDE, SUSMITA DASH, KRIPA VARANASI, Massachusetts Institute of Technology, VARANASI GROUP TEAM — Accelerated corrosion and scale buildup near oceans is partially due to deposition of salty sea mist onto ships, cars, and building structures. Many corrosion preventative measures are expensive, time intensive, and/or have negative impacts on the environment. One solution is the use of specific surfaces that are engineered for scale resistance. In this work, we show that we can delay crystallization and reduce scale adhesion on specifically engineered liquid impregnated surfaces (LIS). The low contact angle hysteresis of the LIS results in a sliding contact line of the saline droplet during evaporation, and the elevated energy barrier of the smooth liquid interface delays crystallization. Experiments conducted on surfaces with different wettability also demonstrate the corresponding influence in controlling salt crystal polymorphism.

6:16PM E36.00004 Designing icophobic surfaces by passively sustaining liquid film at ice-substrate interface , TOM ZHAO, PAUL JONES, NEELESH PATANKAR, Northwestern University — Ice formation poses a significant barrier to transportation, energy generation and transport, gas extraction, etc. We propose to design icophobic surfaces that reduce ice formation and lower ice adhesion by sustaining a film of liquid water at the interface between bulk ice and the substrate. The liquid layer is in phase equilibrium with the surrounding bulk ice, and thus exists without constant energy input. Using molecular dynamic simulations, we show this liquid film can be maintained indefinitely by exploiting the phenomena of interfacial premelting and the freezing point depression of ice confined in surface texture due to the Gibbs Thomson effect. We demonstrate the reduction of both the work and strength of ice adhesion as a function of surface wettability and geometric parameters of the surface texture.

6:29PM E36.00005 Dynamics of Defrosting on Hydrophobic and Superhydrophobic Surfaces, KEVIN MURPHY, Virginia Tech, WILLIAM MCCLINTIC, KEVIN LESTER, PATRICK COLLIER, Center for Nanophase Materials Science, JONATHAN BOREYKO, Virginia Tech — It has recently been demonstrated that frost can grow in a suspended Cassie state on nanostructured superhydrophobic surfaces, which has implications for enhanced defrosting rates. However, to date there have been no direct comparisons of the defrosting kinetics of Cassie frost versus frost on conventional surfaces. Here, we fabricate a hybrid aluminum plate where half of the top face exhibits a superhydrophobic nanostructure while the other half is smooth and hydrophobic. By growing frost to varying thicknesses and melting at several tilt angles, we reveal the advantages and disadvantages of each surface with regards to the extent and speed of the shedding of melt water. For sufficiently thick frost layers, the Cassie state of frost on the superhydrophobic surface uniquely enabled the rapid and effective shedding of melt water even at low tilt angles. On the other hand, the hydrophobic surface was more effective at removing very thin frost sheets, as the reduced contact angle of water on the surface facilitated the coalescence of droplets to grow the melt water beyond the capillary length for gravitational removal. Therefore, the utilization of superhydrophobic versus hydrophobic surfaces for defrosting applications depends upon the context of the system conditions.

6:42PM E36.00006 An Experimental Investigation on Bio-inspired Icephebic Coatings for Aircraft Icing Mitigation¹, HUI HU, HAIXING LI, RYE WALDMAN, Iowa State University — By leveraging the Icing Research Tunnel available at Iowa State University (ISU-IRT), a series of experimental investigations were conducted to elucidate the underlying physics pertinent to aircraft icing phenomena. A suite of advanced flow diagnostic techniques, which include high-speed photographic imaging, digital image projection (DIP), and infrared (IR) imaging thermometry, were developed and applied to quantify the transient behavior of water droplet impingement, wind-driven surface water runback, unsteady heat transfer and dynamic ice accreting process over the surfaces of airfoil/wing models. The icephobic performance of various bio-inspired superhydrophobic coatings were evaluated quantitatively at different icing conditions. The findings derived from the icing physics studies can be used to improve current icing accretion models for more accurate prediction of ice formation and accretion on aircraft wings and to develop effective anti-/de-icing strategies for safer and more efficient operation of aircraft in cold weather.

¹The research work is partially supported by NASA with grant number NNX12AC21A and National Science Foundation under award numbers of CBET-1064196 and CBET- 1435590.

Sunday, November 20, 2016 5:37PM - 6:42PM – Session E37 Drops: Rupture Portland Ballroom 252 - Andrew Wollman, Portland State University

5:37PM E37.00001 Self-similar breakup of a retracting liquid cone, FREDERIK BRASZ, ALEXIS BERNY, JAMES BIRD, Boston University — When a fluid filament breaks up due to the Rayleigh-Plateau instability, a thin thread typically pinches off from a nearly spherical drop. Depending on its shape, this thread can break up again while it retracts to form satellite and even sub-satellite droplets. Past studies have modeled the shape of the retracting filament as a cone, yet the dynamics of nearly inviscid retracting cones are known to be stable, preventing any further filament breakup. Here we show that under certain finite perturbations, retracting conical liquid filaments can become unstable and break up into a cascade of self-similar droplets. Combining numerical simulations and experiments, we explore whether or not a conical filament is likely to break up based on cone angle and initial perturbation. We expect our results to be relevant in applications in which satellite bubbles or droplets are important, such as in modeling the flux of aerosols from the ocean to the atmosphere.

5:50PM E37.00002 The Many Fates of Retracting Newtonian Filaments, CHRISTOPHER ANTHONY, SUMEET THETE, MICHAEL HARRIS, OSMAN BASARAN, Purdue University — The retraction of Newtonian filaments plays a central role in applications as diverse as inkjet printing and atomization where formation of satellite droplets is undesirable. In order to avoid satellite drop production, filaments formed after drop, jet, or sheet breakup should contract to spheres without undergoing further pinch-off. Therefore, it is important to understand all of the dynamical responses that can arise during filament recoil. To accomplish this goal, we use high accuracy simulations to analyze the retraction of Newtonian filaments in a passive ambient fluid. Previously, Notz and Basaran described the fate of low-viscosity filaments. More recent works by Hoepffner and Pare on intermediate viscosity filaments and by Lohse et al. on high viscosity filaments have greatly enhanced our understanding of filament recoil. Unfortunately, taking all of these works in aggregate does not provide a comprehensive picture of filament dynamics. Here, we overcome the deficiencies of these earlier studies to provide a comprehensive analysis of filament recoil and arrive at a complete phase diagram of the system response. While doing so, we also uncover a new mode of filament breakup that has been missed by earlier investigators.

6:03PM E37.00003 Scaling During Drop Formation and Filament (Thread) Breakup, BRAYDEN WAGONER, SUMEET THETE, OSMAN BASARAN, School of Chemical Engineering, Purdue University — Many free surface flows such as drop formation, filament (thread) breakup, and drop coalescence are important in applications as diverse as ink jet printing, atomization, and emulsion science and technology. A common feature of these flows is that they all exhibit finite time singularities. When a liquid filament undergoes capillary thinning and tends toward pinch-off, it is instructive to monitor how certain quantities, such as the threads radius, vary with time remaining until the pinch-off singularity. Experimental determination of this so-called scaling behavior of thread radius and other quantities is important for testing scaling theories and the accuracy of numerical simulations of free surface flows. Conversely, the experimental measurements can be used to develop new theories when none are available. In this talk, we will present some novel ways of experimentally measuring scaling behaviors. The results will be highlighted in terms of experiments involving the formation and breakup of drops and filaments of (a) simple or pure Newtonian fluids and also (b) particle-laden liquids or suspensions containing non-Brownian particles.

6:16PM E37.00004 The role of surfactants in drop formation and thread breakup, PRITISH KAMAT, BRAYDEN WAGONER, SUMEET THETE, OSMAN BASARAN, Purdue University — The ability of surfactants to adsorb onto and lower the surface tension of water-air and water-oil interfaces is exploited in industrial applications, nature, and everyday life. An important example is provided by drop formation where a thinning liquid thread connects an about-to-form globular, primary drop to the rest of the liquid that remains on the nozzle when the primary drop falls from it. Surfactants can affect pinch-off in two ways: first, by lowering surface tension they lower capillary pressure (which equals, to highest order, surface tension divided by thread radius), and second, as surfactant concentration along the interface can be non-uniform, they cause the interface to be subjected to a gradient of surface tension, or Marangoni stress. By means of high-accuracy simulations and supporting experiments, we clarify the role played by surfactants on drop formation and thread breakup.

6:29PM E37.00005 Capillary breakup of fluid threads within confinement¹ , GUOQING

HU, CHUNDONG XUE, XIAODONG CHEN, Institute of Mechanics, Chinese Academy of Sciences — Fluid thread breakup is a widespread phenomenon in nature, industry, and daily life. Driven by surface tension (or capillarity) at low flow-rate condition, the breakup scenario is usually called capillary instability or Plateau-Rayleigh instability. Fluid thread deforms under confinement of ambient fluid to form a fluid neck. Thinning of the neck at low flow-rate condition is quasistatic until the interface becomes unstable and collapses to breakup. Underlying mechanisms and universalities of both the stable and unstable thinning remain, however, unclear and even contradictory. Here we conduct new numerical and experimental studies to show that confined interfaces are not only stabilized but also destabilized by capillarity at low flow-rate condition. Capillary stabilization is attributed to confinement-determined internal pressure that is higher than capillary pressure along the neck. Two origins of capillary destabilization are identified: one is confinement-induced gradient of capillary pressure along the interface; the other is the competition between local capillary pressure and internal pressure.

¹This work was supported by National Natural Science Foundation of China (Grant No. 11402274, 11272321, and 11572334).

Sunday, November 20, 2016 5:37PM - 6:55PM –

Session E38 Flow Instability: Thermal Portland Ballroom 255 - Lou Kondic, New Jersey Institute of Technology

5:37PM E38.00001 Nucleation type instabilities in partially wetting nanoscale nematic liquid films , MICHAEL LAM, LINDA CUMMINGS, LOU KONDIC, New Jersey Institute of Technology — Nucleation type instabilities are studied in nematic liquid crystal (NLC) films with thicknesses less than a micrometer. Within the framework of the long wave approximation, a 4th order nonlinear partial differential equation is proposed for the free surface height. Unlike simple fluids, NLC molecules have a dipole moment which induces an elastic response due to deformation in the bulk of the fluid. The model includes the balance between the bulk elasticity energy and the anchoring (boundary) energy at the substrate and free surface, and van der Waals' intermolecular forces, by means of a structural disjoining pressure. In this presentation, we focus on two-dimensional flow and present simulation results for a flat film with a localized perturbation. We are interested in the morphology of the dewetted film as a function of the initial film thickness. We will show that there exists a range of film thicknesses within the linearly unstable flat film regime where stability analysis does not explain the morphology of the dewetted film. Marginal stability criterion (MSC) is used to derive an analytical expression for the velocity at which a perturbation propagates into the unstable flat film. Finally, we discuss the degree to which MSC can be used to explain the observed morphology.

5:50PM E38.00002 High-frequency vibration of heated liquid layer covered by insoluble surfactant¹ , ALEXANDER MIKISHEV, Sam Houston State University, ALEXANDER NEPOMNYASHCHY, Technion — We study the influence of high-frequency vertical vibration on thin liquid layer with insoluble surfactant adsorbed on the free surface. The layer is subjected to a vertical temperature gradient (the layer is heated either from below or from above). We perform the linear analysis of Marangoni instability. The system is characterized by monotonic and oscillatory modes. The characteristic frequency of long surfactant-induced Marangoni waves is $O(\varepsilon^2)$, where ε is the scale of wavenumber, hence the frequency of external vibrations of order one can be considered as a high frequency. The threshold of the onset of Marangoni convection is shifted by the vibration. Applying a long-wave approach we obtain a system of weakly nonlinear equations describing dynamics near the threshold. The standard Floquet method helps to investigate the excitation of short scale Marangoni waves.

¹ERAU 13373

6:03PM E38.00003 Instabilities of manometric fluid films on a thermally conductive substrate¹ , LOU KONDIC, NANYI DONG, NJIT — We consider thin fluid films placed on thermally conductive substrates and exposed to time-dependent spatially uniform heat source. The evolution of the films is considered within the long-wave framework in the regime such that both fluid/substrate interaction, modeled via disjoining pressure, and Marangoni forces, are relevant. The main finding is that when self-consistent computation of the temperature field is carried out, a complex interplay of different instability mechanisms results. This includes either monotonous or oscillatory dynamics of the free surface. In particular, we find that the oscillatory behavior is absent if the film temperature is assumed to be slaved to the current value of the film thickness. The results are discussed within the context of liquid metal films, but are of relevance to dynamics of any thin film involving variable temperature of the free surface, such that the temperature and the film interface itself evolve on comparable time scales.

¹Supported by NSF Grant No. CBET-1604351

6:16PM E38.00004 Evaporative suppression of Rayleigh-Taylor instability in pure and binary mixtures¹ , RANGA NARAYANAN, DIPIN PILLAI, University of Florida Gainesville — The classic configuration of an interface between a liquid lying above its vapor is well-known to be unstable due to the Rayleigh-Taylor instability. We study this heavy-over-light configuration in the presence of evaporation. For this, a model configuration of a liquid lying above its vapor confined between two flat plates is chosen. The system is heated from the vapor side by maintaining the temperature of the plate in contact with the vapor higher than that in contact with the liquid. A weighted residual-integral boundary layer model is developed for this system. We show that strong evaporation can linearly stabilize Rayleigh-Taylor instability. Interestingly, when evaporation is weak and the system is linearly unstable, it can still evolve nonlinearly to a steady interface configuration that sustains a stable layer of vapor. In the presence of weak evaporation, the interface reaches near the bottom plate. Under such conditions, the system exhibits features of a pure Rayleigh-Taylor. Inertia is shown to slow down the rate at which the interface reaches the steady state. It also results in non-monotonic evolution of the interface. The study is extended to the case of a binary mixture with solutal Marangoni taken into account.

¹NSF 0968313, FSGC NNX15 005

6:29PM E38.00005 Nonlinear dynamics and three-dimensional stable patterns in Rayleigh-Taylor unstable condensing liquid layers: improved one- and 1.5-sided models

, TAO WEI, FEI DUAN, Nanyang Technological University — Three-dimensional patterns of condensing layers suspended on cooled substrates are investigated with an introduced vapor boundary layer (VBL), to which the changes in gas composition and temperature are confined. The interfacial vapor transport equation incorporates convective and diffusive fluxes, coupled with a long-wave evolution equation for interface location. Our work extends that of Kanatani [1] on sessile evaporating films to Rayleigh-Taylor unstable condensing layers with nonlinear theory and involves effects of vapor recoil, gravity combined with buoyancy in condensate and heat flux in VBL. The general framework is termed as 1.5-sided model, which can reduce to a one-sided model. An extended 1.5-sided basic state is proposed, whose stability is investigated with linear stability analysis and nonlinear simulation. With one-sided model, finite-time rupture is found, suggesting that destabilizations of vapor recoil and negative gravity combined with buoyancy prevail over stabilizations of mass gain, thermocapillarity and surface tension. This is in sharp contrast to 1.5-sided model, in which such a layer can be stabilized by convection and diffusion of vapor on interface, and rupture is suppressed as stable pattern even with induced destabilization of thermocapillarity.

6:42PM E38.00006 Oscillatory instability of a self-rewetting film driven by thermal modulation

, WILLIAM BATSON, YEHUDA AGNON, ALEX ORON, Technion-Israel Institute of Technology — Here we consider the self-rewetting fluids (SRWFs) that exhibit a well-defined minimum surface tension with respect to temperature, in contrast to those where surface tension decreases linearly. Utilization of SRWFs has grown significantly in the past decade, due to observations that heat transfer is enhanced in applications such as film boiling and pulsating heat pipes. With similar applications in mind, we investigate the dynamics of a thin SRWF film which is subjected to a temperature modulation in the bounding gas. A model is developed within the framework of the long-wave approximation, and a time-averaged thermocapillary driving force for destabilization is uncovered for SRWFs that results from the nonlinear surface tension. Linear analysis of the nonlinear PDE for the film thickness is used to determine the critical conditions at which this driving force destabilizes the film, and numerical integration of this evolution equation reveals that linearly unstable perturbations saturate to regular periodic solutions (when the modulational frequency is set properly). Properties of these flows such as bifurcation and long-domain flows, where multiple unstable linear modes interact, will also be discussed.

Sunday, November 20, 2016 5:37PM - 6:55PM –

Session E39 Bio: Medical Devices Portland Ballroom 256 - On Shun Pak, Santa Clara University

5:37PM E39.00001 Influence of Slippery Pacemaker Leads on Lead-Induced Venous Occlusion

, WEIGUANG YANG, Stanford University, SAGAR BHATIA, DAYNA OBENAUF, MAX RESSE, Santa Clara University, MAHDI ESMAILY-MOGHADAM, JEFFREY FEINSTEIN, Stanford University, ON SHUN PAK, Santa Clara University — The use of medical devices such as pacemakers and implantable cardiac defibrillators have become commonplace to treat arrhythmias. Pacing leads with electrodes are used to send electrical pulses to the heart to treat either abnormally slow heart rates, or abnormal rhythms. Lead induced vessel occlusion, which is commonly seen after placement of pacemaker or ICD leads, may result in lead malfunction and/or SVC syndrome, and makes lead extraction difficult. The association between the anatomic locations at risk for thrombosis and regions of venous stasis have been reported previously. The computational studies reveal obvious flow stasis in the proximity of the leads, due to the no-slip boundary condition imposed on the lead surface. With the advent of recent technologies capable of creating slippery surfaces that can repel complex fluids including blood, we explore computationally how local flow structures may be altered in the regions around the leads when the no-slip boundary condition on the lead surface is relaxed using various slip lengths. The findings evaluate the possibility of mitigating risks of lead-induced thrombosis and occlusion by implementing novel surface conditions (i.e. theoretical coatings) on the leads.

5:50PM E39.00002 Computational device design: measuring esophageal distensibility using EndoFLIP¹

, SHASHANK ACHARYA, Department of Mechanical Engineering, Northwestern University, WENJUN KOU, PETER J. KAHRILAS, JOHN E. PANDOLFINO, Feinberg School of Medicine, Northwestern University, NEELESH A. PATANKAR, Department of Mechanical Engineering, Northwestern University — Characterizing the strength of sphincters in the human body is valuable from a diagnostic and surgical standpoint. We develop a numerical model for the EndoFLIP device (Endolumenal Functional Lumen Imaging Probe) that is crucial to the biomechanical study of the Lower Esophageal Sphincter (LES). The simulations demonstrate how the device operates *in vivo*. From this model, we suggest additional use cases for the device that can give insight into the state of the esophageal wall. Currently, the device measures a single steady quantity (distensibility) that is calculated from pressure and area. Our analysis shows that by capturing and analyzing spatio-temporal pressure variations during peristalsis, the effectiveness of the contractions and health of the surrounding tissue can be quantified. Furthermore, there is an opportunity to validate tissue models by comparing dilation results with clinical data from the device.

¹This work is supported by the Cabell Fellowship at Northwestern University

6:03PM E39.00003 Prediction and optimization of the recovery rate in centrifugal separation of platelet-rich plasma (PRP)¹

, LINFENG PIAO, HYUNGMIN PARK, Seoul National University, CHRIS JO, Seoul National University Hospital — We present a theoretical model of the recovery rate of platelet and white blood cell in the process of centrifugal separation of platelet-rich plasma (PRP). For the practically used conditions in the field, the separation process is modeled as a one-dimensional particle sedimentation; a quasi-linear partial differential equation is derived based on the kinematic-wave theory. This is solved to determine the interface positions between supernatant-suspension and suspension-sediment, used to estimate the recovery rate of the plasma. While correcting the Brown's hypothesis (1989) claiming that the platelet recovery is linearly proportional to that of plasma, we propose a new correlation model for prediction of the platelet recovery, which is a function of the volume of whole blood, centrifugal acceleration and time. For a range of practical parameters, such as hematocrit, volume of whole blood and centrifugation (time and acceleration), the predicted recovery rate shows a good agreement with available clinical data. We propose that this model is further used to optimize the preparation method of PRP that satisfies the customized case.

¹Supported by a grant (MPSS-CG-2016-02) through the Disaster and Safety Management Institute funded by Ministry of Public Safety and Security of Korean government.

6:16PM E39.00004 Multiscale modeling of a Chemofilter device for filtering chemotherapy toxins from blood¹, NAZANIN MAANI, SAMAN BEYHAGHI, University of Wisconsin Milwaukee, DARYL YEE, California Institute of Technology, MICHEAL NOSONOVSKY, University of Wisconsin Milwaukee, JULIA GREER, California Institute of Technology, STEVEN HETTS, University of California San Francisco, VITALIY RAYZ, University of Wisconsin Milwaukee — **Purpose:** Chemotherapy drugs injected intra-arterially to treat cancer can cause systemic toxic effects. A catheter-based Chemofilter device, temporarily deployed in a vein during the procedure can filter excessive drug from the blood thus reducing chemotherapy side-effects. CFD modeling is used to design the membrane of the Chemofilter in order to optimize its hemodynamic performance. **Methods:** Multiscale approach is used to model blood flow through the Chemofilter. The toxins bind to the Chemofilter's membrane formed by a lattice of numerous micro cells deployed in a blood vessel of much larger size. A detailed model of the flow through a 2x2 microcell matrix with periodic boundary conditions is used to determine the permeability of the membrane. The results are used to simulate the flow through the whole device modeled as a uniform porous membrane. The finite-volume solver Fluent is used to obtain the numerical solution. **Results:** The micro cell matrix has a porosity of 0.92. The pressure drop across the resolved microcells was found to be 630 Pa, resulting in the permeability of $6.21 \times 10^{-11} \text{ m}^2$ in the normal direction. These values were used to optimize the device geometry in order to increase the contact area of the membrane, while minimizing its obstruction to the flow.

¹NIH NCI R01CA194533

6:29PM E39.00005 Stochastic Model of Clogging in a Microfluidic Cell Sorter, THOMAS FAI, CHRIS RYCROFT, Harvard University — Microfluidic devices for sorting cells by deformability show promise for various medical purposes, e.g. detecting sickle cell anemia and circulating tumor cells. One class of such devices consists of a two-dimensional array of narrow channels, each column containing several identical channels in parallel. Cells are driven through the device by an applied pressure or flow rate. Such devices allows for many cells to be sorted simultaneously, but cells eventually clog individual channels and change the device properties in an unpredictable manner. In this talk, we propose a stochastic model for the failure of such microfluidic devices by clogging and present preliminary theoretical and computational results. The model can be recast as an ODE that exhibits finite time blow-up under certain conditions. The failure time distribution is investigated analytically in certain limiting cases, and more realistic versions of the model are solved by computer simulation.

6:42PM E39.00006 An application of the focused liquid jet: needle free drug injection system¹, AKIHITO KIYAMA, CHIHIRO KATSUTA, SENNOSUKE KAWAMOTO, NANAMI ENDO, AKANE TANAKA, YOSHIOUKI TAGAWA, Tokyo Univ of Agri & Tech — Recently, a focused liquid jet draws great attention since it can be applied to various applications (e. g. Ink jet printing, medical devices). In our research, in order to discuss its applicability for a needle-free drug injection system, we shoot a focused liquid jet (Tagawa, et al., Phys. Rev. X, 2012) to an animal skin with very high-speed. Previously, the penetration of this jet into a gelatin and an artificial skin has been performed in order to model of the jet penetration process (Tagawa, et al., Lab. Chip., 2013). However, experiment for jet injection into the animal skin has not been conducted yet. In this presentation, we inject ink as the liquid jet into the skin of the hairless rat. We observe the top/back view and the cross-sectional view of the injected (ink-stained) skin. We capture the stained area of the skin in order to find characteristics of the jet penetration. We discuss the criteria for the jet penetration into the skin.

¹This work was supported by JSPS KAKENHI Grant Numbers JP26709007, JP16J08521

Sunday, November 20, 2016 5:37PM - 6:55PM –
Session E40 Magnetohydrodynamics: Magnetic Fields Portland Ballroom 253-258-254-257 -
Oleg Zikanov, University of Michigan - Dearborn

5:37PM E40.00001 Metal pad instabilities in liquid metal batteries¹, OLEG ZIKANOV, Univ of Michigan - Dearborn — Strong variations between the electrical conductivities of electrolyte and metal layers in a liquid metal battery indicate the possibility of 'metal pad' instabilities. Deformations of the electrolyte-metal interfaces cause strong perturbations of electric currents, which, hypothetically, can generate Lorentz forces enhancing the deformations. We investigate this possibility using two models: a mechanical analogy and a two-dimensional linearized approximation. It is found that the battery is prone to instabilities of two types. One is similar to the sloshing-wave instability observed in the Hall-Héroult aluminum reduction cells. Another is new and related to the interactions of current perturbations with the azimuthal magnetic field induced by the base current.

¹Financial support was provided by the U.S. National Science Foundation (Grant CBET 1435269)

5:50PM E40.00002 Resonance of a Metal Drop under the Effect of Amplitude-Modulated High Frequency Magnetic Field¹, JIAHONG GUO, Shanghai Institute of Applied Mathematics and Mechanics, Shanghai University, Shanghai, China, ZUOSHENG LEI, Shanghai Key Laboratory of Modern Metallurgy & Material Processing, Shanghai University, Shanghai, China, HONGDA ZHU, Shanghai Institute of Applied Mathematics and Mechanics, Shanghai University, Shanghai, China, LIJIE ZHANG, Shanghai Key Laboratory of Modern Metallurgy & Material Processing, Shanghai University, Shanghai, China, MAGNETIC HYDRODYNAMICS(SIAMM) TEAM, MAGNETIC MECHANICS AND ENGINEERING(SMSE) TEAM — The resonance of a sessile and levitated drop under the effect of high frequency amplitude-modulated magnetic field (AMMF) is investigated experimentally and numerically. It is a new method to excite resonance of a metal drop, which is different from the case in the presence of a low-frequency magnetic field. The transient contour of the drop is obtained in the experiment and the simulation. The numerical results agree with the experimental results fairly well. At a given frequency and magnetic flux density of the high frequency AMMF, the edge deformations of the drop with an azimuthal wave numbers were excited. A stability diagram of the shape oscillation of the drop and its resonance frequency spectrum are obtained by analysis of the experimental and the numerical data. The results show that the resonance of the drop has a typical character of parametric resonance.

¹The National Natural Science Foundation of China (No. 51274237 and 11372174)

6:03PM E40.00003 Geometrical shock dynamics of fast magnetohydrodynamic shocks¹, WOUTER MOSTERT, DALE I. PULLIN, Graduate Aerospace Laboratories, California Institute of Technology, RAVI SAMTANEY, Mechanical Engineering, King Abdullah University of Science and Technology, VINCENT WHEATLEY, School of Mechanical and Mining Engineering, University of Queensland — We extend the theory of geometrical shock dynamics (GSD, Whitham 1958), to two-dimensional fast magnetohydrodynamic (MHD) shocks moving in the presence of nonuniform magnetic fields of general orientation and strength. The resulting generalized area-Mach number rule is adapted to MHD shocks moving in two spatial dimensions. A partially-spectral numerical scheme developed from that of Schwendeman (1993) is described. This is applied to the stability of plane MHD fast shocks moving into a quiescent medium containing a uniform magnetic field whose field lines are inclined to the plane-shock normal. In particular, we consider the time taken for an initially planar shock subject to an initial perturbed magnetosonic Mach number distribution, to first form shock-shocks.

¹Supported by KAUST OCRM Award No. URF/1/2162-01

6:16PM E40.00004 Mixed convection in liquid metal flow in a horizontal duct with strong axial magnetic field¹, XUAN ZHANG, OLEG ZIKANOV, University of Michigan - Dearborn — The work is motivated by design of self-cooled liquid-metal breeder blankets for Tokamak fusion reactors. Thermal convection caused by non-uniform internal heating in a liquid metal flow in a horizontal duct with strong axial magnetic field is analyzed numerically. Axial magnetic field is considered strong enough (the Hartmann number up to 10^4 corresponding to typical reactor condition) to suppress the streamwise variation of the flow, so a two-dimensional fully developed flow is studied. Duct walls are assumed to be thermally and electrically insulated. The non-uniform internal heat deposited by captured neutrons is fully diverted by the mean flow. Realistically high Grashof (up to 10^{11}) and Reynolds (up to 10^6) numbers are considered. It is found that the state of the flow is strongly affected by the vertical stable stratification developing in response to the streamwise growth of mean temperature. Two flow regimes are identified: the regime with developed transverse convection at moderate Grashof numbers, and the regime, in which convection is suppressed at high Grashof numbers.

¹Financial support was provided by the U.S. National Science Foundation (Grant CBET 1435269) and by the University of Michigan Dearborn.

6:29PM E40.00005 Effects of magnetic and kinetic helicities on the growth of magnetic fields in laminar and turbulent flows by helical-Fourier decomposition¹, MORITZ LINKMANN, GANAPATI SAHOO, University of Rome Tor Vergata, MAIRI MCKAY, ARJUN BERERA, University of Edinburgh, LUCA BIFERALE, University of Rome Tor Vergata — We perform an analytical and numerical study of incompressible homogeneous conducting fluids by Fourier-helical decomposition of the equations of magnetohydrodynamics (MHD) and a subsequent reduction of the number of degrees of freedom. From the stability properties of the most general subset of interacting velocity and magnetic fields on a closed Fourier triad, we make predictions on the large-scale magnetic-field growth depending on the distribution of magnetic and kinetic helicities among the three wavenumbers. In the kinematic dynamo regime we predict the formation of a large-scale magnetic component with a magnetic helicity of opposite sign with respect to the kinetic helicity, a sort of triadic-by-triad α -effect in Fourier space, while in presence of strong small-scale magnetic helicity we predict an inverse cascade of magnetic helicity. We confirm these predictions through a series of Direct Numerical Simulations, either seeding different magnetic helical components in a strongly helical flow (turbulent/laminar) or directly injecting helical magnetic fluctuations at small scales. Our results show that important dynamical features of MHD flows can be predicted from an analytically tractable dynamical system derived directly from the MHD equations.

¹ERC ADG NewTURB 2013

6:42PM E40.00006 Azimuthal swirl in liquid metal electrodes and batteries, RAKAN ASHOUR, DOUGLAS KELLEY, Univ of Rochester — Liquid metal batteries consist of two molten metals with different electronegativity separated by molten salt. In these batteries, critical performance related factors such as the limiting current density are governed by fluid mixing in the positive electrode. In this work we present experimental results of a swirling flow in a layer of molten lead-bismuth alloy driven by electrical current. Using in-situ ultrasound velocimetry, we show that poloidal circulation appears at low current density, whereas azimuthal swirl becomes dominant at higher current density. The presence of thermal gradients produces buoyant forces, which are found to compete with those produced by current injection. Taking the ratio of the characteristic electromagnetic to buoyant flow velocity, we are able to predict the current density at which the flow becomes electromagnetically driven. Scaling arguments are also used to show that swirl is generated through self-interaction between the electrical current in the electrode with its own magnetic field.

Monday, November 21, 2016 8:00AM - 10:10AM – Session G1 Instabilities at Soft Interfaces | A105 - Camille Duprat, LadHyX, Ecole Polytechnique

8:00AM G1.00001 Sedimentation of slender elastic filaments in a viscous liquid, VERONICA RASPA, LadHyX, Ecole polytechnique, ANKE LINDNER, OLIVIA DU ROURE, PMMH, ESPCI, CAMILLE DUPRAT, LadHyX, Ecole polytechnique — We explore experimentally the dynamics of slender flexible filaments sedimenting in a viscous fluid at low Reynolds number. The observed deformations and dynamics result from a balance between viscous, elastic and gravitational forces on the slender body and thus are characterized by a dimensionless elasto-gravity number. We present measurements of the filaments stationary shape, velocities and trajectories for different initial conditions and filament characteristics (i.e: density, bending rigidity, size). In particular, we observe bending and reorientation of the filament, and investigate the conditions under which the filament can buckle. The introduction of elasticity broadens the spectrum of accessible sedimentation stationary states, compared to those appearing for their rigid counterparts where nor bending or buckling are allowed.

8:13AM G1.00002 Viscous fingering in an elastic channel¹ , ANDREW L. HAZEL, LUCIE DUCLOUÉ,

ANNE JUEL, University of Manchester, UK — We investigate experimentally the fingering instability of a flat, steadily propagating interface in a Hele-Shaw channel, where the top boundary has been replaced by an elastic membrane. In order to create a steadily propagating flat front, we exploit the reopening modes of fluid-filled elasto-rigid channels. The collapsed upper boundary reopens through the steady propagation of a wide finger, when air is injected from one end at a constant flow rate. For high levels of collapse and high finger speed, the tip of the finger becomes flat, creating a leading edge normal to the direction of propagation, which in turn is subject to a smaller scale viscous fingering instability. By modifying the cross-sectional geometry of the channel, we can actuate the finger shape to observe a variety of small-scale fingering phenomena including growth in a direction normal to the propagation and dendrite formation. The instability of the flat front exhibits constant-length fingers, very similar to the stubby fingers observed in radial compliant Hele-Shaw cells, and reminiscent of the printers instability travel with the front. We investigate the geometry of those fingers in terms of the speed of the front, and the geometry of the reopening region.

¹The financial support of the Leverhulme Trust is gratefully acknowledged.

8:26AM G1.00003 Crustal fingering: solidification on a moving interface , XIAOJING

FU, Massachusetts Institute of Technology, JOAQUIN JIMENEZ-MARTINEZ, MARK PORTER, Los Alamos National Laboratory, LUIS CUETO-FELGUEROZO, Technical University of Madrid, Madrid, Spain, RUBEN JUANES, Massachusetts Institute of Technology — Viscous fingering—the hydrodynamic instability that takes place when a less viscous fluid displaces a more viscous fluid—is a well known phenomenon. Motivated by the formation of gas hydrates in seafloor sediments and during the ascent of gas bubbles through ocean water, here we study the interplay of immiscible viscous fingering with solidification of the evolving unstable interface. We present experimental observations of the dynamics of a bubble of Xenon in a water-filled and pressurized Hele-Shaw cell. The evolution is controlled by two processes: (1) the formation of a hydrate “crust” around the bubble, and (2) viscous fingering from bubble expansion. To reproduce the experimental observations, we propose a phase-field model that describes the nucleation and thickening of a porous solid shell on a moving gas-liquid interface. We design the free energy of the three-phase system (gas-liquid-hydrate) to rigorously account for interfacial effects, mutual solubility, and phase transformations (hydrate formation and disappearance). We introduce a pseudo-plasticity model with large variations in viscosity to describe the plate-like rheology of the hydrate shell. We present high-resolution numerical simulations of the model, which illustrate the emergence of complex “crustal fingering” patterns as a result of gas fingering dynamics modulated by hydrate growth at the interface.

8:39AM G1.00004 Elastic Suppression of Viscous Fingering , GUNNAR PENG, JOHN LISTER,

University of Cambridge — Consider peeling an elastic tape or beam away from a rigid base to which it is stuck by a film of viscous liquid. The peeling motion requires air to invade the viscous liquid and is thus susceptible to the Saffman–Taylor fingering instability. We analyse the fundamental travelling-wave solution and show that the advancing air–liquid interface remains linearly stable at higher capillary numbers than in a standard Hele-Shaw cell. A short-wavelength expansion yields an analytical expression for the growth rate which is valid for all unstable modes throughout the parameter space, allowing us to identify and quantify four distinct physical mechanisms that each help suppress the instability. Applying our method to the experiments by Pihler-Puzovic et al. (2012) reveals that the radial geometry and time-variation stabilize the system further.

8:52AM G1.00005 Viscous fingering of a draining suspension , YUN CHEN, FRANK MALAMBRI,

SUNGYON LEE, Texas A&M University — The Saffman-Taylor viscous fingering arises when a viscous oil is withdrawn from a Hele-Shaw cell that is filled with a less viscous fluid. When particles are introduced into the draining fluid, new behaviors emerge, which are unobserved in the well-established pure oil case. We experimentally investigate the particle-modified inward fingering for varying particle concentrations. In particular, the fingering growth rate and number of fingers are experimentally quantified and are shown to be directly affected by the presence of particles. The physical mechanism of the particle-modified fingering is also discussed.

9:05AM G1.00006 Capillary pinch-off of a viscous suspension. , JORIS CHATEAU, ELISABETH

GUZZELLI, HENRI LHUISSIER, Aix Marseille Univ. & CNRS — We study how the presence of non-Brownian, non-colloidal and neutrally buoyant beads suspended in a Newtonian liquid affects the break-up of a capillary thread. Both unstable capillary bridges and threads stretched behind a dripping drop are considered. On the early stage of pinch-off, the suspension behaves as a continuous medium. On the later stage, when the neck diameter reaches a few particle sizes, the pinch-off accelerates continuously until the thinning rate of a pure liquid thread is recovered. We will discuss how these two regimes and their transition depend on the particle volume fraction, size and wettability, as well as on the pinch-off configuration.

9:18AM G1.00007 The dynamics of semiflexible actin filaments in simple shear flow¹

, YANAN LIU, ANKE LINDNER, OLIVIA DU ROURE, PMMH, ESPCI, Paris — The rheological properties of complex fluids made of particles in a suspended fluid depend on the behavior of microscopic particles in flow. A first step to understand this link is to investigate the individual particle dynamics in simple shear flows. A rigid rod will perform so-called Jeffery orbits, however when the rod becomes flexible and Brownian, the behavior in terms of deformation and migration is still to be fully understood. We chose here to address this situation by studying experimentally the behavior of semiflexible polymers. We use actin filaments and combine fluorescent labeling techniques, microfluidic devices to carry out controlled systematical experiments. Different dynamics are observed as a function of the elasto-viscous number, comparing viscous forces to elastic restoring forces $\zeta = (8\pi\eta\dot{\gamma}L^4)/(Lpk_B T)$. The bending modulus of the actin filaments is given by its persistence length $L_p = 17 \pm 1 \mu\text{m}$. When increasing the elasto-viscous number we subsequently observe tumbling, buckling, and bending under flow. Those observations seem to be in good agreement with recent numerical simulations. At the same time, actin filaments fluctuate due to Brownian motion and these fluctuations can modify the individual dynamics of actin filaments.

¹ERC PaDy No.682367

9:31AM G1.00008 Swelling-induced surface instabilities in growing poroelastic polymer networks , MATTHEW G. HENNESSY, Imperial College London, ALESSANDRA VITALE, Politecnico di Torino, JOAO T. CABRAL, OMAR K. MATAR, Imperial College London — The swelling that occurs when a deformable polymer network absorbs solvent can generate large compressive stresses which, in turn, can lead to a rich variety of surface instabilities. In this talk, we will discuss recent experiments by our group which suggest that the growth of a polymer network by photopolymerisation and the onset of swelling-induced surface instabilities can simultaneously occur and drive the self-assembly of complex three-dimensional structures. In addition, we will present a theoretical model of photopolymersation that captures the growth, swelling, and mechanical response of the polymer network. The model is based on an Eulerian formulation of nonlinear poroelasticity. The transport of monomer is described by a generalisation of Darcy's law that accounts for flow due to gradients in the pressure and composition. A combination of asymptotic analysis and finite-element simulations is used to explore the coupling between growth and instability as well as the resulting surface morphologies.

9:44AM G1.00009 Patterns in swelling hydrogels, CHRIS MACMINN, Department of Engineering Science, University of Oxford, THIBAULT BERTRAND, Department of Mechanical Engineering and Materials Science, Yale University, JORGE PEIX-INHO, Laboratoire Ondes et Milieux Complexes, CNRS & Université de Normandie, SHOMEEK MUKHOPADHYAY, Department of Geology and Geophysics, Yale University — Swelling is a process in which a porous material spontaneously grows by absorbing additional pore fluid. Polymeric hydrogels are highly deformable materials that can experience very large volume changes during swelling. This allows a small amount of dry gel to absorb a large amount of fluid, making gels extremely useful in applications from moisture control to drug delivery. However, a well-known consequence of these extreme volume changes is the emergence of a striking morphological instability. We study the transient mechanics of this instability here by combining a theoretical model with a series of simple experiments, focusing on the extent to which this instability can be controlled by manipulating the rate of swelling.

9:57AM G1.00010 Lubricated wrinkles, OUSMANE KODIO, IAN GRIFFITHS, DOMINIC VELLA, Mathematical Institute, University of Oxford — We investigate the problem of an elastic beam above a thin viscous layer. The beam is subject to a fixed end-to-end displacement, which will ultimately cause it to adopt the Euler-buckled state. However, additional liquid must be drawn in to allow this buckling. In the interim, the beam forms a wrinkled state with wrinkles coarsening over time. This problem has been studied experimentally by Vandeparre *et al.* *Soft Matter* (2010), who provided a scaling argument suggesting that the wavelength, λ , of the wrinkles grows according to $\lambda \sim t^{1/6}$. However, a more detailed theoretical analysis shows that, in fact, $\lambda \sim (t/\log t)^{1/6}$. We present numerical results to confirm this and show that this result provides a better account of previous experiments.

Monday, November 21, 2016 8:00AM - 9:57AM – Session G2 Wind Turbines: LES A106 - James Brasseur, University of Colorado

8:00AM G2.00001 Fundamental Distinctions in Physics underlying Nonsteady Forcings of Wind Turbine Power vs. Drivetrain by Atmospheric Turbulence, JAMES BRASSEUR, U Colorado, ADAM LAVELY, TARAK NANDI, Penn State — Whereas the primary function of a wind turbine (WT) is the generation of electricity, wind farm profitability is decreased both by integrated losses in power and increases in premature failures of drivetrain components resulting from energetic nonsteady aerodynamic forcings of WT rotors by atmospheric and wake turbulence. Here we contrast the physics underlying dominant nonsteady atmospheric turbulence forcings of the bending moments in the WT rotor plane (torque/power) vs. the out-of-plane bending moments (OPBM) that underlie premature drivetrain component failure. Using an advanced actuator line model of the 5 MW NREL and the 1.5 MW GE wind turbine rotors embedded within a high-fidelity spectral LES of a typical daytime convective atmospheric boundary layer, we show that (1) the physics underlying large torque vs. OPBM fluctuations are associated with fundamentally different turbulence eddy characteristics and (2) nonsteady response centers on 4 characteristic time scales associated advection of eddies and load response of blades cutting through internal turbulence eddy structure. *Supported by DOE. Computer resources by NSF/XSEDE.*

8:13AM G2.00002 Wind-farms in shallow conventionally neutral boundary layers: effects of transition and gravity waves on energy budget¹, JOHAN MEYERS, DRIES ALLAERTS, KU Leuven, Mechanical Engineering, Celestijnenlaan 300A, B3001 Leuven — Conventionally neutral boundary layers (CNBL) often arise in offshore conditions. In these situations the neutral boundary layer is capped by a strong inversion layer and a stably stratified free atmosphere aloft. We use large-eddy simulations to investigate the interaction between a CNBL and a large wind farm. Following the approach of Allaerts & Meyers (2015) [1], a set of equilibrium CNBLs are produced in a precursor simulation, with a height of approx. 300, 500, and 1000m, respectively. These are used at the inlet of a large wind-farm with a fetch of 15 km, and 20 rows of turbines. We find that above the farm, an internal boundary layer (IBL) develops. For the two lower CNBL cases, the IBL growth is stopped by the overlying capping inversion. Moreover, the upward displacement of the CNBL excites gravity waves in the inversion layer and the free atmosphere above. For the lower CNBL cases, these waves induce significant pressure gradients in the farm. A detailed energy budget analysis of the CNBL is further presented. [1] Allaerts Dries, Meyers Johan (2015). Large eddy simulation of a large wind-turbine array in a conventionally neutral atmospheric boundary layer. *Physics of Fluids* 27 (6), art.nr. 065108

¹The authors acknowledge support from the European Research Council (FP7-Ideas, grant no. 306471).

8:26AM G2.00003 Spectra and Large-Scale Structures in a Turbulent Boundary Layer Interacting with Wind Turbine Arrays¹, YULIA PEET, TANMOY CHATTERJEE, Arizona State University — Wind Turbine Array Boundary Layer is a relatively simple, yet useful theoretical conceptualization to study very large wind farms in an atmospheric boundary layer. In this talk, we investigate the length scales of eddies involved in the power generation in these very large, “infinite” wind farms by analyzing the spectra of the turbulent flux of the mean kinetic energy from Large Eddy Simulations (LES). A goal is to provide a fundamental understanding of the dynamic behavior, the size, the scaling laws and the anisotropic structure of the energy containing eddies responsible for power generation from the wind turbines. Large-scale structures with an order of magnitude bigger than the turbine rotor diameter are shown to have substantial contribution to wind power. The study is performed with a Spectral Element LES code with the recently implemented near-wall model and the actuator line model to represent the effect of rotating wind turbine blades. In this presentation, we also explore an idea of a “multiscale” wind farm, where larger and smaller turbines are arranged in a symbiotic way, with smaller turbines helping to harvest additional power from the wakes of the larger turbines, inspired by the findings of the spectral analysis in uniform wind farms.

¹NSF CBET 13358568 award

8:39AM G2.00004 Boundary-layer flow and power output in large wind farms during transition from neutral to stable conditions¹, DRIES ALLAERTS, JOHAN MEYERS, KU Leuven, Mechanical Engineering, Celestijnenlaan 300A, B3001 Leuven, Belgium — In wind farms, power deficits are directly related to ambient turbulence levels. Power deficits will therefore increase during the transition from a daytime, conventionally neutral boundary layer (CNBL) to the stable boundary layer (SBL) at night. Besides turbulent decay, a multitude of effects occurs during this transition. For instance, low-level jets may cause strong winds at high elevations, while the velocity near the surface generally decreases. Consequently, Coriolis forces induce a change in wind direction, which alters the apparent wind-farm layout in streamwise direction. In this study, we perform LES of a large onshore wind farm in the late-afternoon transition from an equilibrium CNBL to a surface-cooled SBL. The results of two different cooling rates are compared with the wind-farm performance in the CNBL. The power output decrease during the transition, with faster decrease for stronger surface cooling. However, the initial decrease is dominated by the reduction in wind speed, and the relative power deficits do not increase. Further, considerable wake deflection occurs, and a spatially heterogeneous distribution of temperature and heat flux is observed.

¹The authors acknowledge support from the European Research Council (FP7–Ideas, grant no. 306471).

8:52AM G2.00005 Yaw control for power optimization of an array of turbines: large eddy simulations¹, UMBERTO CIRI, MARIO ROTEA, STEFANO LEONARDI, The University of Texas at Dallas — Nowadays, advanced control systems are highly sought for the efficient operation of large clusters of wind turbines. The main objective is to mitigate wake interactions thus increasing annual energy production and/or limiting fatigue loads. Several control strategies have been proposed: generator torque, blade pitch angle and turbine yaw angle. Specifically, the introduction of a misalignment between the rotor plane and the wind direction (i.e. a non-zero yaw angle) causes the wake to laterally displace. Consequently, this phenomenon can potentially be exploited to avoid or reduce waked operations in aligned turbines configurations. However, the successful use of this strategy requires proper coordination between the individual machines in order to identify the optimal yaw angles. Because of the complex mechanisms which are expected to occur in this kind of flow, modeling inaccuracies may have a major impact on the results. As a consequence, a model-free approach is pursued, namely a Nested Extremum Seeking Control, coupled with Large-Eddy Simulations to assess the impact on performances of this control strategy, devise optimal settings and identify key interactions.

¹This work is supported by NSF Award IIP 1362033, NSF IR/D program (while Dr. Rotea is serving at the NSF), NSF grant n. 1243482. TACC is acknowledged for computational resources.

9:05AM G2.00006 A numerical investigation of the role of the turbine rotor scale and the nacelle on wake meandering¹, DANIEL FOTI, University of Minnesota, XIAOLEI YANG, Stonybrook University, LIAN SHEN, University of Minnesota, FOTIS SOTIROPOULOS, Stonybrook University — Recent analysis of a hydrokinetic turbine (Kang et al. J. Fluid Mech., 2014) and laboratory scale wind turbine (Foti et al. Phys. Rev. Fluids, 2016) reveal that the turbine nacelle has a considerable effect on the turbulence kinetic energy and wake meandering. However, the role of the nacelle on wake meandering for utility-scale wind turbines has not been fully investigated. In this work, a numerical investigation using large eddy simulations of four wind turbines with rotor diameters ranging from laboratory to utility scale reveals similar turbulent structures in the far wake and a comparable wake meandering Strouhal number regardless of rotor size. By reconstructing the wake meandering with three dimensional spatio-temporal filtering process, first proposed in Foti et al. (Phys. Rev. Fluids, 2016), the statistics of the dynamics of the wake meandering are quantified in terms of amplitude and wavelength. Results indicate that the wavelength of wake meandering can be properly scaled by rotor diameter of the turbines for both simulations with and without a nacelle model. The meandering amplitude, on the other hand, is larger for the simulation with a nacelle. This is further quantitative evidence that a nacelle model is imperative to accurately capturing wake meandering.

¹This work was supported by Department of Energy (DE-EE0002980, DE-EE0005482 and DE-AC04-94AL85000), and Sandia National Laboratories

9:18AM G2.00007 Effect of nacelle on the wake meandering in Horns Rev wind farm¹, XIAOLEI YANG, Stony Brook University, DANIEL FOTI, University of Minnesota, FOTIS SOTIROPOULOS, Stony Brook University — Turbine wake meandering has considerable effects on the velocity deficit and turbulence intensity in the wake. However, the mechanism for wake meandering is still not well understood and low-order models cannot take into account the wake meandering effects accurately. A recent work by Kang, Yang and Sotiropoulos (Journal of Fluid Mechanics 744 (2014): 376-403) showed that the nacelle has a significant effect on the wake meandering of a hydrokinetic turbine. To examine the nacelle contributions to wake meandering and wake interactions in utility-scale wind farms, we simulate the atmospheric turbulent flow over the Horns Rev wind farm using large-eddy simulation with actuator type models. In a preliminary simulation on a coarse grid using actuator line model for turbine blades without a nacelle model, the computed power shows overall good agreement with field measurements. Fine grid simulations using an actuator surface model for turbine blades with and without a nacelle model are being carried out. The corresponding results will be presented with analysis on wake meandering dynamics using the technique proposed by Horward et al. (Physics of Fluids 27 (2015): 075103) and Foti et al. (Physical Review Fluids, accepted).

¹This work was supported by DOE (DE-AC04-94AL85000), Xcel Energy (grant RD4-13) and Sandia National Laboratories. Computational resources were provided by National Renewable Energy Laboratory and University of Minnesota Supercomputing Institute.

9:31AM G2.00008 Performance of a wind turbine over a ridged terrain¹, CHRISTIAN SANTONI, UMBERTO CIRI, STEFANO LEONARDI, The University of Texas at Dallas — Performance of wind turbines is affected by their interaction with the topography. Low momentum flow from the terrain may impinge the turbine resulting in fatigue loads that may reduce durability. However, at the same time it may promote the transport of momentum and kinetic energy into the wake improving the power production on the downstream turbines. In order to address how the topography affects the flow, Large Eddy Simulations of a wind turbine located on a wavy surface are performed. The height variation of the topography is described by a sinusoidal wave. Two different amplitudes were considered, $0.10D$ and $0.05D$, where D is the rotor diameter. The wavelength has been kept constant to $3D$. The effect of the relative position of rotor and terrain geometry was assessed by placing the turbine either at the crest or at the trough of the undulated wall. NREL-5MW turbine blades were modeled using the actuator line model whereas the tower, nacelle and topography using the immersed boundary method. A simulation of a wind turbine on a flat terrain was performed as reference case. The performance of the turbine was evaluated in terms of the power production and blade load fluctuations, as well as for the energy entrainment into the wake of the turbine.

¹The numerical simulations were performed on XSEDE TACC under Grant No. CTS070066. This work was supported by the National Science Foundation, grant number IIA-1243482 (the WINDINSPIRE project).

9:44AM G2.00009 Unsteady Flow in Different Atmospheric Boundary Layer Regimes and Its Impact on Wind-Turbine Performance, IMAN GOHARI, Ph.D. Candidate, ARTEM KOROBENKO, Ph.D. , JINHUI YAN, Ph.D. Candidate, YURI BAZILEVS, SUTANU SARKAR, Professor — Wind is a renewable energy resource that offers several advantages including low pollutant emission and inexpensive construction. Wind turbines operate in conditions dictated by the Atmospheric Boundary Layer (ABL) and that motivates the study of coupling ABL simulations with wind turbine dynamics. The ABL simulations can be used for realistic modeling of the environment which, with the use of fluid-structure interaction, can give realistic predictions of extracted power, rotor loading, and blade structural response. The ABL simulations provide inflow boundary conditions to the wind-turbine simulator which uses arbitrary Lagrangian-Eulerian variational multiscale formulation. In the present work, ABL simulations are performed to examine two different scenarios: (i) A neutral ABL with zero heat-flux and inversion layer at 350m, in which the wind turbine experiences maximum mean shear; (2) A shallow ABL with the surface cooling-rate of -1 K/hr, in which the wind turbine experiences maximum mean velocity at the low-level-jet nose height. We will discuss differences in the unsteady flow between the two different ABL conditions and their impact on the performance of the wind turbine cluster in the coupled ABL-wind turbine simulations.

Monday, November 21, 2016 8:00AM - 10:10AM –

Session G3 Vortex Dynamics: Mechanisms and Plates B110-111 - Melissa Green, Syracuse University

8:00AM G3.00001 Identification and tracking of hairpin vortex auto-generation in turbulent wall-bounded flow, YANGZI HUANG, MELISSA GREEN, Syracuse Univ — Hairpin vortices have been widely accepted as component structures of turbulent boundary layers. Their properties (size, vorticity, energy) and dynamic phenomena (origin, growth, breakdown) have been shown to correlate to the complex, multi-scaled turbulent motions observed in both experiments and simulations. As established in the literature, the passage of a hairpin vortex creates a wall-normal ejection of fluid, which encounters the high-speed freestream resulting in near-wall shear and increased drag. A previously generated simulation of an isolated hairpin vortex is used to study the auto-generation of a secondary vortex structure. Eulerian methods such as the Q criterion and 2 function, as well as Lagrangian methods are used to visualize the three-dimensional hairpin vortices and the auto-generation process. The circulation development and wall-normal location of both primary and secondary hairpin heads are studied to determine if there is a correlation between the strength and height of the primary hairpin vortex with the secondary hairpin vortex auto-generation.

8:13AM G3.00002 Vortex Clusters and Their Time Evolution in High- Reynolds-Number Turbulence¹, TAKASHI ISHIHARA, Nagoya University, ATSUYA UNO, RIKEN AICS, KOJI MORISHITA, MITSUO YOKOKAWA, Kobe University, YUKIO KANEDA, Aichi Institute of Technology — Time series data (with a time interval of $4\tau_\eta$) obtained by high-resolution direct numerical simulations (DNSs) of forced incompressible turbulence in a periodic box, with a maximum of 12288^3 grid points and Taylor micro-scale Reynolds numbers R_λ up to 2300, are used to study the vortex dynamics in high Reynolds number (Re) turbulent flows. Here τ_η is the Kolmogorov time scale. A visualization method to handle such large-scale data was developed for this study. In the high Re turbulence generated by the DNS, we observed the dynamics of tube-like vortex clusters of various sizes, which are constructed by strong micro vortices. For example, we observed the generation of the tube-like clusters of various sizes and the processes of their merging and breakdown. We also observed layer-like vortex clusters of the order of the integral length scale forming shear layers in the high Re turbulence.

¹This research used computational resources of the K computer and other computers of the HPCI system provided by the AICS and the ITC of Nagoya University through the HPCI System Research Project (Project ID:hp150174, hp160102).

8:26AM G3.00003 Vortex reconnection in the K-type transitional channel flow¹, YAOMIN ZHAO, YUE YANG, SHIYI CHEN, Peking Univ — Vortex reconnection, as the topological change of vortex lines or surfaces, is a critical process in transitional flows, but is challenging to accurately characterize in shear flows. We apply the vortex-surface field (VSF), whose isosurface is the vortex surface consisting of vortex lines, to study vortex reconnection in the K-type temporal transition in channel flow. Based on the VSF, both qualitative visualization and quantitative analysis are used to investigate the reconnection between the hairpin-like vortical structures evolving from the opposite channel halves. The incipient vortex reconnection is characterized by the vanishing minimum distance between a pair of vortex surfaces and the reduction of vorticity flux through the region enclosed by the VSF isolines on the spanwise symmetric plane. In addition, we find that the surge of the wall friction coefficient begins at the identified reconnection time, which is discussed with the induced velocity during reconnection and the Biot-Savart law.

¹This work has been supported in part by the National Natural Science Foundation of China (Grant Nos. 11522215 and 11521091), and the Thousand Young Talents Program of China.

8:39AM G3.00004 Thermodynamic effects during vortex reconnection, JEAN-PIERRE HICKEY, University of Waterloo — Thermodynamic and compressibility effects during vortex reconnection are studied using fully-resolved, high-order direct numerical simulations. The bridging and reconnection of coherent structures is believed to play a key role in the understanding of the turbulent energy cascade as it provides a mechanism for the energy transfer from the large scale down to the dissipation scale. Previous studies have focused on the incompressible cases to understand the hydrodynamics of reconnection process. Here, the focus lies on the thermodynamic effects, more specifically, the temperature and density changes and concomitant thermophysical variations resulting from the large pressure gradients at the time of reconnection. The initial setup replicates the anti-parallel vortex configuration proposed by Melander and Hussain (CTR Summer program, 1988). The simulations are conducted with a third-order in time, sixth-order compact finite difference (in space) schemes for the solution of the fully compressible, Navier-Stokes equations at Reynolds numbers (circulation/viscosity) from 1000 to 5000. The results will highlight the importance of the thermodynamic effects during the reconnection process and the dependence on the local Mach number of the flow.

8:52AM G3.00005 Vortex breakdown in simple pipe bends, JESSE AULT, SANGWOO SHIN, HOWARD STONE, Princeton University — Pipe bends and elbows are one of the most common fluid mechanics elements that exists. However, despite their ubiquity and the extensive amount of research related to these common, simple geometries, unexpected complexities still remain. We show that for a range of geometries and flow conditions, these simple flows experience unexpected fluid dynamical bifurcations resembling the bubble-type vortex breakdown phenomenon. Specifically, we show with simulations and experiments that recirculation zones develop within the bends under certain conditions. As a consequence, fluid and particles can remain trapped within these structures for unexpectedly-long time scales. We also present simple techniques to mitigate this recirculation effect which can potentially have impact across industries ranging from biomedical and chemical processing to food and health sciences.

9:05AM G3.00006 Vortex-breakdown and wall-separation states in swirling flows in a straight pipe, YUXIN ZHANG, ZVI RUSAK, Rensselaer Polytechnic Institute, SHIXIAO WANG, The University of Auckland, New Zealand — The appearance of vortex-breakdown and wall-separation states in various incoming swirling flows to a straight circular pipe is investigated. Fixed-in-time profiles of the axial and circumferential velocities and of the azimuthal vorticity are prescribed at the pipe inlet. A parallel flow state is set at the pipe outlet. Following the theory of Wang & Rusak (1997), the outlet state of the steady flow problem is determined for a long pipe by solutions of the columnar (axially-independent) Squire-Long equation. For each of the incoming flows studied, these solutions include the base columnar flow state, a decelerated flow along the centerline, an accelerated flow along the centerline, a vortex-breakdown state and a wall-separation state. These theoretical predictions are numerically realized by flow simulations based on the unsteady flow equations. The simulations shed light on the base flow stability and the dynamics of initial perturbations to the various states. The present study extends all the six bifurcation diagrams of solutions studied in Leclaire & Sipp (2010), who stopped the development of branches of steady states once breakdown and wall-separation states first appear.

9:18AM G3.00007 Characterization of Vortex Development on a Pitching Plate¹

KEVIN WABICK, JAMES BUCHHOLZ, The University of Iowa — The formation and evolution of leading-edge vortices (LEVs) is ubiquitous on a broad range of aerodynamic structures and natural fliers, and can have a significant impact on aerodynamic loads. The formation of LEVs is considered on a pitching flat plate at a chord-based Reynolds number of 10^4 with varying dimensionless pitch rate through characterization of the sources and sinks of vorticity that contribute to their growth and evolution. The effect of pitch rate is examined, and the flow field evolution and measured fluxes of vorticity are compared to prior measurements on a purely plunging plate, to isolate the effects of rotation on the factors contributing to vortex strength and development.

¹This work was supported by the Air Force Office of Scientific Research through grant number FA9550-16-1-0107.

9:31AM G3.00008 ABSTRACT WITHDRAWN —

9:44AM G3.00009 Evaluating vortex sheet models for separated flow past a flat plate

MONIKA NITSCHE, University of New Mexico, LING XU, University of Michigan — Numerical studies of separated flows using the full governing equations are computationally expensive. In practice, low order point vortex or vortex sheet models are often used instead. These models are based on simple algorithms to satisfy the Kutta condition at sharp edges. Here, we use highly resolved direct numerical simulations of flow past a finite flat plate to evaluate vortex sheet models for separation. We obtain values for the shed circulation, vortex trajectory and vortex sizes as a function of time and Reynolds number, for accelerated flow past a flat plate at an angle to the incoming flow. We then compare the viscous results with results from a vortex sheet model and determine the extent to which the model reproduces the flow.

9:57AM G3.00010 Vortex dynamics and surface pressure fluctuations on a normal flat plate¹

ARMAN HEMMATI², DAVID H. WOOD, ROBERT J. MARTINUZZI, SIMON W. FERRARI, YAOPING HU, University of Calgary — The effect of vortex formation and interactions on surface pressure fluctuations is examined in the wake of a normal flat plate by analyzing Direct Numerical Simulations at $Re=1200$. A novel local maximum score-based 3D method is used to track vortex development in the region close to the plate where the major contributions to the surface pressure are generated. Three distinct vortex shedding regimes are identified by changes in the lift and drag fluctuations. The instances of maximum drag coincide with impingement of newly formed vortices on the plate. This results in large and concentrated areas of rotational and strain contributions to generation of pressure fluctuations. Streamwise vortex straining and chordwise stretching are correlated with the large ratios of streamwise to chordwise normal stresses and regions of significant rotational contribution to the pressure. In contrast at the minimum drag, the vorticity field close to the plate is disorganized, and vortex roll-up occurs farther downstream. This leads to a uniform distribution of pressure.

¹This study was supported by Alberta Innovates Technology Futures (AITF) and Natural Sciences and Engineering Research Council of Canada (NSERC)

²Currently at Princeton University

Monday, November 21, 2016 8:00AM - 9:57AM —

Session G4 Acoustics III: Thermoacoustics

B112 - Timothy Lieuwen, Georgia Institute of Technology

8:00AM G4.00001 Analytical Model for Axial-Azimuthal Thermoacoustic Modes in an Annular Combustor

VISHAL ACHARYA, TIMOTHY LIEUWEN, Georgia Inst of Tech — Recent advances in theoretical models for azimuthal modes have highlighted their potential in accurately capturing the physics with minimal cost when compared to detailed simulations. Such models for annular combustors have considered multiple burners, effects of the plenum as well as effects of azimuthal mean flow to name a few. However, in all these models, only the azimuthal modes have been considered and as such cannot capture axial-azimuthal coupling of modes. In this paper, we consider an extension of these models by considering the axial extent of the annular combustor with a generic impedance boundary condition at the combustor exit. The inclusion of the axial mode is of practical relevance to combustors where axial-azimuthal modal coupling controls the Thermoacoustic instability of the system.

8:13AM G4.00002 Generation of indirect combustion noise by compositional inhomogeneities

LUCA MAGRI, JEFF O'BRIEN, MATTHIAS IHME, Center for Turbulence Research, Stanford University — The generation of indirect combustion noise in nozzles and turbine stages is commonly attributed to temperature inhomogeneities and vorticity fluctuations. Here, compositional inhomogeneities in a multi-component gas mixture are shown to produce indirect noise both theoretically and numerically. The chemical potential function is introduced as an additional acoustic source mechanism. The contribution of the compositional noise is compared to the entropy noise and direct noise by considering subsonic, supersonic and shocked nozzles downstream of the combustor exit. It is shown that the compositional noise is dependent on the local mixture composition and can exceed entropy noise for fuel-lean conditions and supersonic/shocked nozzle flows. This suggests that compositional indirect combustion noise may require consideration with the implementation of advanced combustion concepts in gas turbines, including low-emissions combustors, high-power-density engine cores, or compact burners.

8:26AM G4.00003 LES Investigation of Core Noise Mechanisms inside a Combustor-Nozzle System

JEFFREY O'BRIEN, Stanford University - Center for Turbulence Research, FRIEDRICH BAKE, German Aerospace Center (DLR) - Institute of Propulsion Technology, JEONGLAE KIM, MATTHIAS IHME, Stanford University - Center for Turbulence Research — The aim of the work is to expand knowledge of core noise physics through the study of a representative aviation-type combustor with converging-diverging nozzle attached to the exhaust. First, a fully compressible LES of the entire flowpath is performed and validated against experimental measurements. From this calculation, the time history of the flow is sampled in a plane near the nozzle entrance to construct a library of representative fluctuations that are potential precursors to the direct & indirect noise observed at the nozzle outlet. This data is then used as an inflow for a series of separate nozzle simulations in which fluctuations in pressure, temperature ("hot spots"), and mixture composition are imposed separately to isolate their effect on the emitted noise. This methodology allows quantitative investigation of core-noise physics that lower-order models do not, including: the effect of non-linearity of high-amplitude perturbations, superposition of forcing types, the impact of the spatial structure of the perturbations, and the restriction to low-frequency perturbations and calorically perfect gas assumption. The calculations also represent the first time variations in mixture composition have been shown to induce downstream noise in a high-fidelity, 3D simulation.

8:39AM G4.00004 Bulk viscosity effects on ultrasonic thermoacoustic instability

JEFFREY LIN, Stanford University, CARLO SCALO, Purdue University, LAMBERTUS HESSELINK, Stanford University — We have carried out unstructured fully-compressible Navier-Stokes simulations of a minimal-unit traveling-wave ultrasonic thermoacoustic device in looped configuration. The model comprises a thermoacoustic stack with 85% porosity and a tapered area change to suppress the fundamental standing-wave mode. A bulk viscosity model, which accounts for vibrational and rotational molecular relaxation effects, is derived and implemented via direct modification of the viscous stress tensor, $\tau_{ij} \equiv 2\mu \left[S_{ij} + \frac{\lambda}{2\mu} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right]$, where the bulk viscosity is defined by $\mu_b \equiv \lambda + \frac{2}{3}\mu$. The effective bulk viscosity coefficient accurately captures acoustic absorption from low to high ultrasonic frequencies and matches experimental wave attenuation rates across five decades. Using pressure-based similitude, the model was downscaled from total length $L = 2.58$ m to 0.0258 m, corresponding to the frequency range $f = 242 - 24200$ Hz, revealing the effects of bulk viscosity and direct modification of the thermodynamic pressure. Simulations are carried out to limit cycle and exhibit growth rates consistent with linear stability analyses, based on Rott's theory.

8:52AM G4.00005 Nonlinear modeling of thermoacoustically driven energy cascade

PRATEEK GUPTA, CARLO SCALO, Purdue Univ, GUIDO LODATO, INSA de Rouen — We present an investigation of nonlinear energy cascade in thermoacoustically driven high-amplitude oscillations, from the initial weakly nonlinear regime to the shock wave dominated limit cycle. We develop a first principle based quasi-1D model for nonlinear wave propagation in a canonical minimal unit thermoacoustic device inspired by the experimental setup of Biwa et al. (JASA, 2011). Retaining up to quadratic nonlinear terms in the governing equations, we develop model equations for nonlinear wave propagation in the proximity of differentially heated no-slip boundaries. Furthermore, we discard the effects of acoustic streaming in the present study and focus on nonlinear energy cascade due to high amplitude wave propagation. Our model correctly predicts the observed exponential growth of the thermoacoustically amplified second harmonic, as well as the energy transfer rate to higher harmonics causing wave steepening. Moreover, we note that nonlinear coupling of local pressure with heat transfer reduces thermoacoustic amplification gradually thus causing the system to reach limit cycle exhibiting shock waves. Throughout, we verify the results from the quasi-1D model with fully compressible Navier-Stokes simulations.

9:05AM G4.00006 Gas dynamics of heat-release-induced waves in supercritical fluids: revisiting the Piston Effect, MARIO TINDARO MIGLIORINO, CARLO SCALO, Purdue University — We investigate a gasdynamic approach to the modeling of heat-release-induced compression waves in supercritical fluids. We rely on highly resolved one-dimensional fully compressible Navier-Stokes simulations of CO₂ at pseudo-boiling conditions in a closed duct inspired by the experiments of Miura et al., Phys. Rev. E, vol. 74, 2006. Near-critical fluids exhibit anomalous variations of thermodynamic variables taken into account by adopting the Peng-Robinson equation of state and Chung's Method. An idealized heat source is applied, away from the boundaries, resulting in the generation of compression waves followed by contact discontinuities bounding a region of hot expanding fluid. For higher heat-release rates such compressions are coalescent with distinct shock-like features (i.e. non-isentropicity and propagation Mach numbers measurably greater than unity) and a non-uniform post-shock state, not present in ideal gas simulations, caused by the highly nonlinear equation of state. Thermoacoustic effects are limited to: (1) a one-way/one-time thermal-to-acoustic energy conversion, and (2) cumulative non-isentropic bulk heating due to the resonating compression waves, resulting in what is commonly referred to as the Piston Effect.

9:18AM G4.00007 The nature of combustion noise: Stochastic or chaotic?¹, VIKRANT GUPTA, None, MIN CHUL LEE, Incheon National University, LARRY K. B. LI, Hong Kong University of Science & Technology — Combustion noise, which refers to irregular low-amplitude pressure oscillations, is conventionally thought to be stochastic. It has therefore been modeled using a stochastic term in the analysis of thermoacoustic systems. Recently, however, there has been a renewed interest in the validity of that stochastic assumption, with tests based on nonlinear dynamical theory giving seemingly contradictory results: some show combustion noise to be stochastic while others show it to be chaotic. In this study, we show that this contradiction arises because those tests cannot distinguish between noise amplification and chaos. We further show that although there are many similarities between noise amplification and chaos, there are also some subtle differences. It is these subtle differences, not the results of those tests, that should be the focus of analyses aimed at determining the true nature of combustion noise. Recognizing this is an important step towards improved understanding and modeling of combustion noise for the study of thermoacoustic instabilities.

¹This work was supported by the Research Grants Council of Hong Kong (Project No. 16235716 and 26202815).

9:31AM G4.00008 Forced synchronization of thermoacoustic oscillations in a ducted flame¹, YU GUAN, VIKRANT GUPTA, The Hong Kong University of Science and Technology, KARTHIK KASHINATH, Lawrence Berkeley National Lab, LARRY K.B. LI, The Hong Kong University of Science and Technology — Forced synchronization is a process in which a self-excited system subjected to external forcing starts to oscillate at the forcing frequency f_f in place of its own natural frequency f_n . There are two motivations for studying this in thermoacoustics: (i) to determine how external forcing could be used to control thermoacoustic oscillations, which are harmful to many combustors; and (ii) to better understand the nonlinear interactions between self-excited hydrodynamic and thermoacoustic oscillations. In this experimental study, we examine the response of a ducted premixed flame to harmonic acoustic forcing, for two natural states of the system: (1) a state with periodic oscillations at f_1 and a marginally stable mode at f_2 ; and (2) a state with quasiperiodic oscillations at two incommensurate frequencies f_1 and f_2 . When forcing the periodic state, we find that the forcing amplitude required for lock-in increases linearly with $|f_f - f_1|$ and that the marginally stable mode becomes excited when $f_f \approx f_2$. When forcing the quasiperiodic state, we find that the system locks into the forcing when $f_f \approx f_1$ or f_2 or $1/2(f_1 + f_2)$. These findings should lead to improved control of periodic and aperiodic thermoacoustic oscillations in combustors.

¹This work was supported by the Research Grants Council of Hong Kong (Project No. 16235716 and 26202815).

9:44AM G4.00009 Numerical Study of Energy Conversion in the Taconis Oscillations by Tracing Fluid Particles, SHIZUKO ADACHI, School of Business and Commerce, Tokyo International University, KATSUYA ISHII, Department of Information Science, Ochanomizu University — Temporal evolution of physical properties of spontaneous thermoacoustic oscillations of a helium gas in a closed cylindrical tube is obtained by solving the axisymmetric compressible Navier-Stokes equations. The ratio of the wall temperature of the hot part near both ends to that of the cold central part is 15. We trace fluid particles which start from various points in a closed tube for a fundamental mode oscillation of a standing wave and a second mode oscillation. Work done by the fluid particles is numerically estimated. Fluid particles drift in the closed tube while oscillating. Work done by fluid particles moving in a hot region during one cycle is negative in both excitation modes. Displacement of fluid particles moving in the finite temperature gradient region is large. They do not give a cyclic transformation in a pressure-volume diagram. Work done by fluid particles moving near the tube axis from the cold part to the hot part is positive. Work done by fluid particles moving in a cold region during one cycle is positive but the amount of the work is smaller than that in the hot region.

Monday, November 21, 2016 8:00AM - 9:44AM –

Session G5 Compressible Flow: Shock Interactions B113 - Veronica Eliasson, University of California, San Diego

8:00AM G5.00001 Optimization on the focusing of multiple shock waves., SHI QIU,

University of Southern California, VERÓNICA ELIASSEN, University of Southern California & University of California, San Diego — Focusing of multiple shock waves can lead to extreme thermodynamic conditions, which are desired for applications like shock wave lithotripsy and inertial confinement fusion. To study shock focusing effects, multiple energy sources have been placed in a circular pattern around an intended target, while the distance between each source and the target is fixed. All the sources are set to release the same amount of energy at the same time in order to create multiple identical shock waves. The object is to optimize the thermodynamic conditions at the target by rearranging the initial placement of each source. However, dealing with this optimization problem can be challenging due to the high computational cost introduced by solving the Euler equations. To avoid this issue, both numerical and analytical methods have been applied to handle shock focusing more efficiently. A numerical method, an approximate theory named Geometrical Shock Dynamics (GSD), has been utilized to describe the motion of shock. Using an analytical method, a transition curve between regular and irregular reflection has been employed to predict shock interactions. Results show that computational cost can be reduced dramatically by combining GSD and a transition curve. In addition, optimization results based on varying initial setups is discussed.

8:13AM G5.00002 Unsteadiness of a shock train in Mach 2.0 flow, ROBIN HUNT, JAMES

DRISCOLL, MIRKO GAMBA, University of Michigan — Experimental observations of the progression of flow unsteadiness within a shock train are presented. A downstream control valve is used to generate a shock train in the constant area test section of a wind tunnel with a freestream Mach number of 2.0. Even with nominally constant boundary conditions the shock train exhibits inherent unsteady motion about the time average position. At the conditions presented the shocks can be displaced by up to 0.35 duct heights. Better knowledge of the shock train's dynamics may allow us to introduce control algorithms to reduce the system's unsteadiness and thus minimize the associated mechanical and thermal loads. An edge detection algorithm is applied to the instantaneous frames of high speed Schlieren movies to track the location of morphological features within the shock system. Simultaneously, high speed pressure transducers record the pressure fluctuations along the bottom wall of the duct. The results indicate a complex frequency dependent dynamical system. A strong component of the dynamics involves a disturbance traveling upstream through the boundary layer. Once the disturbance reaches the leading shock foot the shocks respond in order with the most upstream shock moving first.

8:26AM G5.00003 Shock-Capturing Simulations of Multi-Fluid Shock-Turbulence

Interactions, YIFENG TIAN, FARHAD JABERI, Michigan State University, DANIEL LIVESCU, Los Alamos National Laboratory, ZHAORUI LI, Texas AM University — The interaction between an isotropic multi-fluid turbulence with a planar shock wave is studied using turbulence resolved shock-capturing simulations. This problem is an extension of the canonical Shock-Turbulence Interaction (STI), with the effects of strong density variations (from compositional changes) taken into consideration. To establish shock-capturing simulation as a reliable method for studying STI, LIA convergence tests are conducted. These tests are consistent with previous DNS studies and indicate that LIA limits can be approximated at relatively high Reynolds numbers and low turbulent Mach numbers when the separation between numerical shock thickness and turbulent length scales is adequate. When variable density effects are introduced, turbulence structure is modified more by the normal shock, with a differential distribution of turbulent statistics in regions with different densities, resulting in a strong mixing asymmetry in the post-shock region. Turbulence achieves similar axisymmetric two-dimensional local state right after the shock wave in the multi-fluid case, but has a faster return to three-dimensional isotropic structure when compared to the single-fluid case. The characteristics of post-shock thermodynamic fluctuations are also affected and are dominated by shock strength fluctuations that result from the compositional changes.

8:39AM G5.00004 Shock wave-free interface interaction, ROMAN FROLOV, PETER MINEV, ROUS-

LAN KRECHETNIKOV, University of Alberta — The problem of shock wave-free interface interaction has been widely studied in the context of compressible two-fluid flows using analytical, experimental, and numerical techniques. While various physical effects and possible interaction patterns for various geometries have been identified in the literature, the effects of viscosity and surface tension are usually neglected in such models. In our study, we apply a novel numerical algorithm for simulation of viscous compressible two-fluid flows with surface tension to investigate the influence of these effects on the shock-interface interaction. The method combines together the ideas from Finite Volume adaptation of invariant domains preserving algorithm for systems of hyperbolic conservation laws by Guermond and Popov and ADI parallel solver for viscous incompressible NSEs by Guermond and Minev. This combination has been further extended to a two-fluid flow case, including surface tension effects. Here we report on a quantitative study of how surface tension and viscosity affect the structure of the shock wave-free interface interaction region.

8:52AM G5.00005 Predictive Analytical Model for Isolator Shock-Train Location in a Mach 2.2 Direct-Connect Supersonic Combustion Tunnel, JOE LINGREN, LEON VANSTONE,

KELLEY HASHEMI, The University of Texas at Austin, SIVARAM GOGINENI, Spectral Energies LLC, JEFFREY DONBAR, Air Force Research Laboratory, MARUTHI AKELLA, NOEL CLEMENS, The University of Texas at Austin — This study develops an analytical model for predicting the leading shock of a shock-train in the constant area isolator section in a Mach 2.2 direct-connect scramjet simulation tunnel. The effective geometry of the isolator is assumed to be a weakly converging duct owing to boundary-layer growth. For some given pressure rise across the isolator, quasi-1D equations relating to isentropic or normal shock flows can be used to predict the normal shock location in the isolator. The surface pressure distribution through the isolator was measured during experiments and both the actual and predicted locations can be calculated. Three methods of finding the shock-train location are examined, one based on the measured pressure rise, one using a non-physics-based control model, and one using the physics-based analytical model. It is shown that the analytical model performs better than the non-physics-based model in all cases. The analytic model is less accurate than the pressure threshold method but requires significantly less information to compute. In contrast to other methods for predicting shock-train location, this method is relatively accurate and requires as little as a single pressure measurement. This makes this method potentially useful for unstart control applications.

9:05AM G5.00006 The pressure impulse of a laser-induced underwater shock wave¹,

YOSHIYUKI TAGAWA, SHOTA YAMAMOTO, KEISUKE HAYASAKA, MASAHIRO KAMEDA, Tokyo Univ of Agri & Tech — We investigate the pressure impulse, the time integral of pressure evolution, of a laser-induced underwater shock wave. We simultaneously observe plasma formation, shock-wave expansion, and pressure in water using a combined measurement system that obtains high-resolution nanosecond-order image sequences. Remarkably, pressure impulse is found to distribute symmetrically for a wide range of experimental parameters even when the shock waves are emitted from an elongated plasma. In contrast, distribution of pressure peak is found to be non-spherically-symmetric. We rationalize aforementioned results by considering the structure of the underwater shock wave as a collection of multiple spherical shock waves originated from point-like plasmas in an elongated region.

¹This work was supported by JSPS KAKENHI Grant Number JP26709007

9:18AM G5.00007 ABSTRACT WITHDRAWN —

9:31AM G5.00008 Experimental and numerical investigation of cylindrical and hemispherical jet formation, MATTHEW BETNEY, PETA FOSTER, TIM RINGROSE, THOMAS EDWARDS, BRETT TULLY, HUGO DOYLE, NICHOLAS HAWKER, First Light Fusion Ltd., FIRST LIGHT FUSION LTD. TEAM — This paper presents a detailed investigation of the formation of jets in cylindrical and spherical cavities, when impacted by shocks at extreme pressures. As the shock pressure increases the effects of material strength lessen in proportion. Beyond a certain magnitude the behaviour is referred to as hydrodynamic. In this domain both cylindrical and spherical cavities involute to form jets, which go on to strike the leeward cavity wall, compressing the cavity contents to high pressures and temperatures. In this study, the jet formation process is isolated by cutting hemispherical and half-cylindrical cavities from the rear side of PMMA and copper blocks. This allows direct measurement of the jet speed and shape using high speed imaging, providing data against which numerical models may be compared. Shock waves at pressures of up to 30 GPa are formed in the targets by the impact of projectiles from a two-stage light gas gun, at velocities of up to 7 km/s. Numerically, the jet formation process is modelled using our in-house front-tracking code. This code uses Lagrangian hypersurfaces to model the interfaces between different media, with an underlying Eulerian mesh used to model the bulk flow. Detailed comparisons between numerical and experimental results are presented.

Monday, November 21, 2016 8:00AM - 9:57AM —

Session G6 Aerodynamics: Flapping Wings B114 - Karen Mullenens, Ecole Polytechnique Federale de Lausanne

8:00AM G6.00001 Effect of Oscillatory Plunging Motion on Airfoil Boundary Layer and Wake Behavior¹, MARK AGATE, JESSE LITTLE, Department of Aerospace and Mechanical Engineering, The University of Arizona, ANDREAS GROSS, New Mexico State University, HERMANN FASEL, Department of Aerospace and Mechanical Engineering, The University of Arizona — The effects of small amplitude ($0.030 < A/c < 0.048$) high frequency ($0.61 < \pi fc/U_\infty < 0.70$) plunging motion of the X-56A airfoil are examined at $Re=200,000$ for three angles of attack. Two angles of attack were chosen at pre-stall conditions and one angle of attack was selected to study post-stall effects. Static stall of the airfoil is 12.25° and the examined angles are 10° , 12° , and 14° . The purpose of this research is to examine the aerodynamic influence of structural motion when the wing is vibrating close to its eigenfrequency near static stall. The aerodynamic characteristics generated by the plunging motion are considered with specific focus on the laminar separation bubble near the leading edge. For the cases examined, the static lift is greatly exceeded. At the plunging case of 10° angle of attack, experimental results are very similar to those obtained from Theodorsen's Theory. For the 12° plunging case, lift exceeds that predicted by Theodorsen's Theory and the leading edge bubble bursts during the oscillation cycle. At the static stall condition of 14° , plunging periodically reattaches the flow and the bubble bursting is much more significant.

¹U.S. Air Force Office of Scientific Research (FA9550-14-1-0184)

8:13AM G6.00002 Circulation Produced by a Flapping Wing During Stroke Reversal¹, MATTHEW BURGE, MATTHEW RINGUETTE, State University of New York at Buffalo — We investigate the circulation behavior of the 3D flow structures formed during the stroke-reversal of a 2-degree-of-freedom flapping wing in hover. Previous work has related circulation peaks to the unsteady wing kinematics and forces. However, information from experiments detailing contributions from the multiple, 3D flow structures is lacking. The objective of this work is to quantitatively study the spanwise circulation as well as the spanwise flow which advects vorticity in the complex loop topology of a flapping wing during stroke reversal. We analyze the flow features of a scaled wing model using multi-plane stereo digital particle image velocimetry in a glycerin-water mixture. Data plane locations along the wing span are inspired by the time-resolved behavior of the 3D vortex structures observed in our earlier flow visualization studies. As with our prior work, we vary dimensionless parameters such as the pitching reduced frequency to understand their effect on the circulation. This research provides insight into the vortex dynamics produced by the coupled rotational and pitching wing motions during stroke reversal, when lift generation is challenging.

¹This work is supported by the National Science Foundation, Award Number 1336548, supervised by Dr. Dimitrios Papavassiliou.

8:26AM G6.00003 Decomposing the aerodynamic forces of low-Reynolds flapping airfoils¹, MANUEL MORICHE, MANUEL GARCIA-VILLALBA, OSCAR FLORES, Universidad Carlos III de Madrid — We present direct numerical simulations of flow around flapping NACA0012 airfoils at relatively small Reynolds numbers, $Re = 1000$. The simulations are carried out with TUCAN, an in-house code that solves the Navier-Stokes equations for an incompressible flow with an immersed boundary method to model the presence of the airfoil. The motion of the airfoil is composed of a vertical translation, heaving, and a rotation about the quarter of the chord, pitching. Both motions are prescribed by sinusoidal laws, with a reduced frequency of $k = 1.41$, a pitching amplitude of 30deg and a heaving amplitude of one chord. Both, the mean pitch angle and the phase shift between pitching and heaving motions are varied, to build a database with 18 configurations. Four of these cases are analysed in detail using the force decomposition algorithm of Chang (1992) and Martín Alcántara et al.(2015). This method decomposes the total aerodynamic force into added-mass (translation and rotation of the airfoil), a volumetric contribution from the vorticity (circulatory effects) and a surface contribution proportional to viscosity. In particular we will focus on the second, analysing the contribution of the leading and trailing edge vortices that typically appear in these flows.

¹This work has been supported by the Spanish MINECO under grant TRA2013-41103-P. The authors thankfully acknowledge the computer resources provided by the Red Espaola de Supercomputacion

8:39AM G6.00004 Modelling forces and flow features in flapping wings: a POD based approach.¹, MARCO RAIOLA, STEFANO DISCETTI, ANDREA IANIRO, Universidad Carlos III De Madrid — A novel POD-based approach to decompose the aerodynamic forces acting on a flapping wing along with the most relevant flow features is proposed. The method is applied to experimental data including PIV and force measurements at $Re = 3600$ and $St = 0.2$. An actuated 2D flapping wing with a NACA 0012 airfoil is designed to produce independent heaving and pitching motion. The wing is equipped with a 6 Degrees-Of-Freedom balance, providing aerodynamic force measurements. Planar PIV measurements are carried out to obtain a phase-locked flow features description in the wing near field. The PIV phase-averaged flow fields are transformed into flow fields in the reference frame fixed with respect to the moving wing. The POD performed on the vorticity field provides a time basis, constituted by the vorticity time coefficients, on which it is possible to project both the flow fields and the forces in order to assess the force contribution of each POD mode. The force generation is mostly ascribed to the first 4 modes. A satisfactory description of the measured forces is achieved through a truncation to the first 6 modes. A more detailed analysis of the flow field projections is useful to determine the force generation mechanism.

¹This work has been supported by the Spanish MINECO under grant TRA2013-41103-P.

8:52AM G6.00005 Numerical study of tandem flapping wings hovering near ground, SRINIDHI N G, VENGADESAN S, Indian Inst of Tech-Madras — The ground effect on tandem elliptical foils hovering in an inclined stroke plane is studied using immersed boundary projection method. The computations are carried out at a low Reynolds number, $Re = 100$, in a quiescent fluid at different heights from the ground. The effect of phase relationship, ψ , between the fore- and hindwings on force variation is studied. Flow induced by the rebound vortices changes the effective angle of attack (AoA) of the wings and influences the force generation. In some cases, the shed vortices merge with the rebound vortices and create a sustained recirculating vortex which has a significant effect on the force generation of the forewing. In counter-stroking ($\psi = 180^\circ$) and in-phase stroking ($\psi = 0^\circ$), the rebound vortices increase the effective AoA of the forewing and increase the lift coefficients; interestingly, for $\psi = 90^\circ$, such an increase in forces is not observed. Except for the cases with $\psi = 90^\circ$, time-averaged vertical force coefficient of the forewing is always greater than the hindwing. For selected cases, backward in time finite-time Lyapunov exponent (FTLE) ridges are used in conjunction with vorticity contours to gain more insight into the vorticity dynamics.

9:05AM G6.00006 Linearized propulsion theory of flapping airfoils revisited¹, RAMON FERNANDEZ-FERIA, Universidad de Malaga (Spain) — A vortical impulse theory is used to compute the thrust of a plunging and pitching airfoil in forward flight within the framework of linear potential flow theory. The result is significantly different from the classical one of Garrick that considered the leading-edge suction and the projection in the flight direction of the pressure force. By taking into account the complete vorticity distribution on the airfoil and the wake the mean thrust coefficient contains a new term that generalizes the leading-edge suction term and depends on Theodorsen function $C(k)$ and on a new complex function $C_1(k)$ of the reduced frequency k . The main qualitative difference with Garrick's theory is that the propulsive efficiency tends to zero as the reduced frequency increases to infinity (as $1/k$), in contrast to Garrick's efficiency that tends to a constant ($1/2$). Consequently, for pure pitching and combined pitching and plunging motions, the maximum of the propulsive efficiency is not reached as $k \rightarrow \infty$ like in Garrick's theory, but at a finite value of the reduced frequency that depends on the remaining non-dimensional parameters. The present analytical results are in good agreement with experimental data and numerical results for small amplitude oscillations.

¹Supported by the Ministerio de Economia y Competitividad of Spain Grant No. DPI2013-40479-P

9:18AM G6.00007 Mean, coherent and stochastic flow structure interactions in the near-wake of an oscillatory foil, FIRAS SIALA, Oregon State University, JAMES LIBURDY, Oreogn State University — Particle image velocimetry measurements are conducted to investigate the transport mechanism of flow kinetic energy in the near-wake of an oscillating foil at a reduced frequency of 0.18–0.2. Velocity triple decomposition is used to decompose the flow into mean, coherent and stochastic fields, and the kinetic energy evolution equations are utilized to study the energy exchange between the three components of the flow fields. The results show that the leading edge vortex (LEV) is responsible in extracting the majority of the free stream kinetic energy via the coherent shear strain. Furthermore, a scale-based model that characterizes the energy content of the LEV is developed. It is shown that coherent kinetic energy produced by the mean rate of strain scales remarkably well with the LEV energy content estimated by the model.

9:31AM G6.00008 Effect of Mean Angle of Attack Modulation on Dynamic Stall, KYLE HEINTZ, THOMAS CORKE¹, University of Notre Dame — Wind tunnel experiments at $M = 0.2$ were conducted on a cambered airfoil instrumented with surface pressure transducers that was oscillated with two independent frequencies. The primary input, f_1 , corresponds to a range of reduced frequencies, while the slower, secondary input, f_2 , drives the modulation of the mean angle of attack, thus varying the stall-penetration angle, α_{pen} . Various combinations transitioned different regimes of dynamic stall from "light" to "deep". Results suggest that when α_{pen} is falling between consecutive cycles, the aerodynamic loads do not fully recover to the values seen when α_{pen} is rising, even though the airfoil recedes to $\alpha_{pen} < 0$ during each oscillation. The experimental data is presented in terms of load coefficients, aerodynamic damping, and their phase relationships to pitch angle.

¹APS Fellow

9:44AM G6.00009 The role of wing kinematics of freely flying birds downstream the wake of flapping wings, KRISHNAMOORTHY KRISHNAN, Student, ROI GURKA, Asso. Professor — Avian aerodynamics has been a topic of research for centuries. Avian flight features such as flapping, morphing and maneuvering make bird aerodynamics a complex system to study, analyze and understand. Aerodynamic performance of the flapping wings can be quantified by measuring the vortex structures present in the downstream wake. Still, the direct correlation between the flapping wing kinematics and the evolution of wake features need to be established. In this present study, near wake of three bird species (western sandpiper, European starling and American robin) have been measured experimentally. Long duration, time-resolved, particle image velocimetry technique has been used to capture the wake properties. Simultaneously, the bird kinematics have been captured using high speed camera. Wake structures are reconstructed from the collected PIV images for long chord distances downstream. Wake vorticities and circulation are expressed in the wake composites. Comparison of the wake features of the three birds shows similarities and some key differences are also found. Wing tip motions of the birds are extracted for four continuous wing beat cycle to analyze the wing kinematics. Kinematic parameters of all the three birds are compared to each other and similar trends exhibited by all the birds have been observed. A correlation between the wake evolutions with the wing motion is presented. It was found that the wings' motion generates unique flow patterns at the near wake, especially at the transition phases. At these locations, a drastic change in the circulation was observed.

Monday, November 21, 2016 8:00AM - 9:57AM –

Session G7 Flow Control: Separation and Reattachment B115 - Jesse Little, University of Arizona

8:00AM G7.00001 Spatially Distributed Forcing for Boundary Layer Separation Control on a Wall Mounted Hump¹, DAVID BORGmann, ARTH PANDE, JESSE LITTLE, University of Arizona, RENE WOSZIDLO, The Boeing Company — Numerous successful efforts on controlling flow separation have been demonstrated using spatially distributed actuators. These include both steady and unsteady forcing from discrete locations in the vicinity of separation. Despite this, there are many open questions on the actual flow control mechanism. A canonical hump model is used to investigate these physics in a subsonic wind tunnel. Reynolds number independence is achieved above 0.72×10^6 and testing is performed up to 2.2×10^6 . The efficacy of discrete steady jets is studied as a function of spacing, momentum coefficient, velocity ratio and mass flux. Highly-resolved surface pressure data for the controlled flow are compared to an inviscid solution establishing a figure of merit. Results indicate the inviscid limit is reached for a momentum coefficient of 1% with actuator spacing of 0.5% span. A comparison of steady discrete jets with sweeping jets actuators of equivalent cross-sectional area is undertaken. Surface flow visualization and PIV are employed to extract detailed information on the baseline and controlled flow field. This importance of establishing critical baseline features is also discussed with respect to establishing proper boundary conditions for accompanying numerical simulations.

¹Supported by The Boeing Company

8:13AM G7.00002 Exploring active flow control for efficient control of separation on an Ahmed model¹, JONATHAN MCNALLY, FARRUKH ALVI, Florida State University — Active flow control is applied to an Ahmed model with a rear slant angle of 25° , where a typical flow field consists of a three-dimensional separation region on the rear slant of the bluff body. Linear arrays of discrete microjets, previously proven to effectively control this separation, are investigated further. A principal aim of this experimental study is to examine the sensitivity of control as the actuator location is shifted with respect to the separation location. Aerodynamic force and surface pressure measurements, combined with the velocity field obtained using particle image velocimetry, provide a measure of control efficacy and insight into the interaction of jet arrays with the local flow field, including the separating shear layer. An energy balance is conducted to characterize control efficiency for multiple positions over a range of microjet array blowing conditions. Results show that moving the actuator array further into the separation region requires higher microjet momentum to obtain a desired aerodynamic benefit. An empirical relationship is also developed for determining the required jet velocity as a function of position by relating the jet penetration distance to local flow features and length scales.

¹Partial support by FCAAP and NSF.

8:26AM G7.00003 Proportional feedback control of flow over a hemisphere¹, JUNGIL LEE, JINHYUK YUN, Ajou University, DONGGUN SON, Korea Atomic Energy Research Institute — In the present study, a proportional feedback control is applied to laminar flow over a hemisphere at $Re = 300$ to reduce its lift fluctuations. As a control input, blowing/suction is distributed on the surface of hemisphere before the separation, and its strength is linearly proportional to the transverse velocity at a sensing location in the centerline of the wake. To determine the optimal sensing location, we introduce a correlation function between the lift force and the time derivative of sensing velocity. The optimal proportional gains for the proportional control are obtained for the sensing locations considered. It is shown that the present control successfully attenuates the velocity fluctuations at the sensing location, resulting in the reduction of lift fluctuations of hemisphere.

¹Supported by NRF program (2014R1A1A1002671)

8:39AM G7.00004 Active Flow Control on a Generic Trapezoidal Wing Planform¹, ISRAEL WYGNANSKI, JESSE LITTLE, BERNHARD ROENTSCH, SEBASTIAN ENDRIKAT, University of Arizona — Fluidic oscillators are employed to increase the lift and improve longitudinal stability of a generic trapezoidal wing having aspect ratio of 1.15 and taper ratio of 0.27. Actuation is applied along the flap hinge which spans the entire wing and is parallel to the trailing edge. Experiments are conducted at a Reynolds number of 1.7×10^6 for a wide range of incidence (-8° to 24°) and flap deflection angles (0° to 75°). Baseline flow on the deflected flap is directed inboard prior to boundary layer separation, but changes to outboard with increasing incidence and flap deflection. The attached spanwise flow can be redirected using a sparse distribution of fluidic oscillators acting as a fluidic fence. However, the majority of lift enhancement and pitch break improvement is accomplished using a more dense distribution of actuators which attaches separated flow to the flap. Integral force and moment results are supported by surface flow visualization, pressure sensitive paint and PIV which reveal unique flow features such as a hinge vortex analogous to the leading edge vortex on a forward swept wing and the possible existence of an absolute instability in a plane parallel to the highly deflected flap.

¹Supported by U.S. Office of Naval Research (N00014-14-1-0387)

8:52AM G7.00005 Active Control of Airfoil Boundary Layer Separation and Wake using Ns-DBD Plasma Actuators¹, CLAUDIA DURASIEWICZ, JORGE CASTRO MALDONADO, JESSE LITTLE, University of Arizona — Nanosecond pulse driven dielectric barrier discharge (ns-DBD) plasma actuators are employed to control boundary layer separation and the wake of a NACA 0012 airfoil having aspect ratio of three. Ns-DBD plasma actuators are known to operate via a thermal mechanism in contrast to ac-DBDs which are momentum-based devices. Nominally 2D forcing is applied to the airfoil leading edge with pulse energy of 0.35 mJ/cm. Experiments are conducted at a Reynolds number of 0.74×10^6 primarily at 18° incidence which is well within the stalled regime. Baseline and controlled flow fields are studied using surface pressure measurements, constant temperature anemometry (CTA) and PIV. Forcing at a dimensionless frequency of $F^+ = fc/U_\infty = 1.14$ results in reattachment of nominally separated flow to the airfoil surface. Lower frequency forcing is less optimal for separation control, but produces strong fluctuations in the wake which are intended for use in the study of vortex body interaction in the future. Actuation below $F^+ = 0.23$ shows behavior consistent with an impulse-like response while forcing in the range $0.23 < F^+ < 0.92$ produces a single dominant frequency in the wake. Spanwise uniformity of the wake is documented using CTA at various locations downstream ($x/c < 7$).

¹Supported by U.S. Army Research Office (W911NF-14-1-0662)

9:05AM G7.00006 Passive Boundary Layer Separation Control on a NACA2415 Airfoil at High Reynolds Numbers, AGASTYA PARIKH, MARCUS HULTMARK, Princeton University — The design and analysis of a passive flow control system for a NACA2415 airfoil is undertaken. There exists a vast body of knowledge on airfoil boundary layer control with the use of controlled mass flux, but there is little work investigating passive mass flux-based methods. A simple duct system that uses the upper surface pressure gradient to force blowing near the leading edge and suction near the trailing edge is proposed and evaluated. 2D RANS analyses at $Re_c \approx 1.27 \times 10^6$ were used to generate potential configurations for experimental tests. Initial computational results suggest drag reductions of approximately 2–7% as well as lift increases of 4–5% at $\alpha = 10.0^\circ$ and $\alpha = 12.5^\circ$. A carbon composite-aluminum structure model that implements the most effective configurations, according to the CFD predictions, has been designed and fabricated. Experiments are being performed to evaluate the CFD results and the feasibility of the duct system.

9:18AM G7.00007 Flow Physics of Synthetic Jet Interactions on a Sweptback Model with a Control Surface, MARIANNE MONASTERO, MICHAEL AMITAY, Rensselaer Polytechnic Institute — Active flow control using synthetic jets can be used on aerodynamic surfaces to improve performance and increase fuel efficiency. The flowfield resulting from the interaction of the jets with a separated crossflow with a spanwise component must be understood to determine actuator spacing for aircraft integration. The current and previous work showed adjacent synthetic jets located upstream of a control surface hingeline on a sweptback model interact with each other under certain conditions. Whether these interactions are constructive or destructive is dependent on the spanwise spacing of the jets, the severity of separation over the control surface, and the magnitude of the spanwise flow. Measuring and understanding the detailed flow physics of the flow structures emanating from the synthetic jet orifices and their interactions with adjacent jets of varying spacings is the focus of this work. Wind tunnel experiments were conducted at the Rensselaer Polytechnic Institute Subsonic Wind Tunnel using stereo particle image velocimetry (SPIV) and pressure measurements to study the effect that varying the spanwise spacing has on the overall performance. Initial SPIV data gave insight into defining and understanding the mechanisms behind the beneficial or detrimental jets interactions.

9:31AM G7.00008 Nonlinear optimal control of bypass transition in a boundary layer flow, DANDAN XIAO, GEORGE PAPADAKIS, Imperial College London — Bypass transition is observed in a flat-plate boundary-layer flow when high levels of free stream turbulence are present. This scenario is characterized by the formation of streamwise elongated streaks inside the boundary layer, their break down into turbulent spots and eventually fully turbulent flow. In the current work, we perform DNS simulations of control of bypass transition in a zero-pressure-gradient boundary layer. A non-linear optimal control algorithm is developed that employs the direct-adjoint approach to minimise a quadratic cost function based on the deviation from the Blasius velocity profile. Using the Lagrange variational approach, the distribution of the blowing/suction control velocity is found by solving iteratively the non-linear Navier-Stokes and its adjoint equations in a forward/backward loop. The optimisation is performed over a finite time horizon during which the Lagrange functional is to be minimised. Large values of optimisation horizon result in instability of the adjoint equations. The results show that the controller is able to reduce the turbulent kinetic energy of the flow in the region where the objective function is defined and the velocity profile is seen to approach the Blasius solution. Significant drag reduction is also achieved.

9:44AM G7.00009 The Dynamics of Controlled Flow Separation within a Diverter Duct Diffuser¹, C. J. PETERSON, B. VUKASINOVIC, A. GLEZER, Georgia Institute of Technology — The evolution and receptivity to fluidic actuation of the flow separation within a rectangular, constant-width, diffuser that is branched off of a primary channel is investigated experimentally at speeds up to $M = 0.4$. The coupling between the diffuser's adverse pressure gradient and the internal separation that constricts nearly half of the flow passage through the duct is controlled using a spanwise array of fluidic actuators on the surface upstream of the diffuser's inlet plane. The dynamics of the separating surface vorticity layer in the absence and presence of actuation are investigated using high-speed particle image velocimetry combined with surface pressure measurements and total pressure distributions at the primary channel's exit plane. It is shown that the actuation significantly alters the incipient dynamics of the separating vorticity layer as the characteristic cross stream scales of the boundary layer upstream of separation and of the ensuing vorticity concentrations within the separated flow increase progressively with actuation level. It is argued that the dissipative (high frequency) actuation alters the balance between large- and small-scale motions near separation by intensifying the large-scale motions and limiting the small-scale dynamics. Controlling separation within the diffuser duct also has a profound effect on the global flow. In the presence of actuation, the mass flow rate in the primary duct increases 10% while the fraction of the diverted mass flow rate in the diffuser increases by more than 45% at 0.7% actuation mass fraction.

¹Supported by the Boeing Company.

Monday, November 21, 2016 8:00AM - 10:10AM – Session G8 Nonlinear Dynamics: Turbulence and Transition to Turbulence B116 - Jae Sung Park, University of Wisconsin-Madison

8:00AM G8.00001 Temporal coherence of turbulent dynamics in minimal channel flow and its connection to exact coherent states¹, JAE SUNG PARK², MICHAEL GRAHAM, University of Wisconsin-Madison — The dynamics of the turbulent near-wall region is known to be dominated by coherent structures. These near-wall coherent structures are observed to burst in an intermittent way, exporting turbulent kinetic energy to the rest of the flow. In addition, they are closely related to invariant solutions known as exact coherent states (ECS), some of which display nonlinear critical layer dynamics. In this study, temporal coherence in minimal channel flow relevant to burst and critical layer dynamics is investigated. The turbulence displays frequencies very close to the critical layer frequency displayed by an ECS family recently identified in the channel flow geometry. The bursting frequency is predominant near the wall, while the critical layer frequency becomes predominant over the bursting frequency as we move away from the wall. In particular, the critical layer frequency becomes more prominent near the channel center and at higher Reynolds number. Finally, turbulent bursts are classified into strong and relatively weak classes with respect to an intermittent approach to a lower branch ECS. The relationship between the strong burst class and the instability of the lower branch ECS 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).

²Will be joining the University of Nebraska-Lincoln as an assistant professor in January 2017.

8:13AM G8.00002 Enhanced energy fluxes via phase precession in forced Burgers equation¹, BRENDAN MURRAY, MIGUEL BUSTAMANTE, CASL, University College Dublin, MICHELE BUZZICOTTI, LUCA BIFERALE, Dept. Physics University of Rome Tor Vergata — We present a study of phase dynamics in the non-linear forced Burgers' equation. We uncover a connection between energy flux across scales and the evolution of triad phase combinations in Fourier space². As this energy is dissipated at small scales, real-space shock structures are associated with entangled correlations amongst the phase precession dynamics and the amplitude evolution of triads in Fourier space. We compute precession frequencies of the triad phases, which show a non-Gaussian distribution with multiple peaks and fat tails, with significant correlation between precession frequencies and amplitude growth. The observed fat tails and non-zero precession frequencies are two key criteria for enhancing energy fluxes via precession resonance³. We search for this resonance by varying the forcing strength and frequency and, additionally, by modifying the dimension of the underlying system via fractal Fourier decimation.

¹Supported by COST (European Cooperation in Science and Technology, Action MP1305) and SFI (Science Foundation Ireland, research grant no. 12/IP/1491).

²M. Buzzicotti, B. P. Murray, L. Biferale and M. D. Bustamante, *EPJE* **39**(3), (2016)

³ M. D. Bustamante, B. Quinn, and D. Lucas, *Phy. Rev. Let.* **113**(8), (2014)

8:26AM G8.00003 A variational method for the identification of nonlinear energy transfers responsible for intermittent fluctuations in turbulent shear flows, MOHAMMAD FARAZMAND, THEMIS SAPSIS, Massachusetts Inst of Tech-MIT — It is believed that the intermittent fluctuations in turbulent shear flows are triggered by the energy transfer to the mean flow via nonlinear inertial interactions. However, because of the vast range of active spatial and temporal scales, identifying the responsible interactions is not straightforward. We show that the responsible modes can be formulated as the (initially unknown) solutions of an appropriate constrained variational problem. The variational problem can be solved at a low computational cost, and the solution is the nontrivial mode with instantaneously maximal transfer of energy to the mean flow. We demonstrate the application of this variational method on a direct numerical simulation of a shear flow with external body forcing.

8:39AM G8.00004 Energy cascade and irreversibility in reversible shell models of turbulence¹, MASSIMO DE PIETRO, University of Roma Tor Vergata, MASSIMO CENCINI, Institute of Complex Systems-CNR, Rome, LUCA BIFERALE, University of Roma Tor Vergata, GUIDO BOFFETTA, University of Torino — Dissipation breaks the time reversibility of the Navier-Stokes equation. It has been conjectured that forced-dissipated Navier-Stokes equations are equivalent to a modified version of the equations in which the dissipative term is modified such as to preserve the time-inversion symmetry. This can be realized choosing a velocity dependent viscosity in such a way to preserve a global quantity, e.g. energy or enstrophy. Here we present results on shell models of turbulence where time reversibility is restored following the mechanism originally suggested. We show that when the time-dependent viscosity is chosen such as to conserve enstrophy, the resulting reversible dynamics exhibit an energy cascade, sharing the same features of the standard irreversible cascade.

¹We acknowledge funding from ERC ADG NewTURB No. 339032

8:52AM G8.00005 One-way spatial integration of Navier-Stokes equations: stability of wall-bounded flows, GEORGIOS RIGAS, TIM COLONIUS, Caltech, AARON TOWNE, Stanford University, MICHAEL BEYAR, Boeing Research & Technology — For three-dimensional flows, questions of stability, receptivity, secondary flows, and coherent structures require the solution of large partial-derivative eigenvalue problems. Reduced-order approximations are thus required for engineering prediction since these problems are often computationally intractable or prohibitively expensive. For spatially slowly evolving flows, such as jets and boundary layers, a regularization of the equations of motion sometimes permits a fast spatial marching procedure that results in a huge reduction in computational cost. Recently, a novel one-way spatial marching algorithm has been developed by Towne & Colonius (JCP 300: 844-861, 2015). The new method overcomes the principle flaw observed in Parabolized Stability Equations (PSE), namely the *ad hoc* regularization that removes upstream propagating modes. The one-way method correctly parabolizes the flow equations based on estimating, in a computationally efficient way, the local spectrum in each cross-stream plane and an efficient spectral filter eliminates modes with upstream group velocity. Results from the application of the method to wall-bounded flows will be presented and compared with predictions from the full linearized compressible Navier-Stokes equations and PSE.

9:05AM G8.00006 Exact Navier-stokes Solutions in a Compressible 2D Open Cavity Flow, JAVIER OTERO, ATI SHARMA, University of Southampton, RICHARD SANDBERG, University of Melbourne — In very simple geometries and always assuming an incompressible flow, researchers have sought to understand the flow physics by looking for steady or periodic flow solutions. These solutions exactly satisfy the governing equations, and determine the physics of the flow. In the current investigation we perform for the first time this type of analysis in a compressible flow and in a complex geometry. In particular, we focus on a 2D laminar inflow open cavity flow at $Re = 2000$, which is simulated using an in-house compressible DNS code. Initially, an exact periodic flow solution is found at $M = 0.5$, which shows a novel noise generation mechanism that we explain in detail. This periodic flow solution is continued across Mach number, covering from $M = 0.25$ to $M = 0.8$. At the lower end of the range, the periodic solution ceases to exist due to the low compressibility of the system and leads to a steady state. This steady solution can be seen as the bifurcation point between the family of steady and periodic solutions.

9:18AM G8.00007 Near critical swirling flow of a viscoelastic fluid, NGUYEN LY, ZVI RUSAK, JOHN TICHY, Rensselaer Polytechnic Institute, SHIXIAO WANG, U. Auckland, New Zealand — The interaction between flow inertia and elasticity in high Re , axisymmetric, and near-critical swirling flows of a viscoelastic fluid in a finite-length straight circular pipe is studied. The viscous stresses are described by the Giesekus constitutive model. The application of this model to columnar streamwise vortices is first investigated. Then, a nonlinear small-disturbance analysis is developed from the governing equations of motion. It explores the complicated interactions between flow inertia, swirl, and fluid viscosity and elasticity. An effective Re that links between steady states of swirling flows of a viscoelastic fluid and those of a Newtonian fluid is revealed. The effects of the fluid viscosity, relaxation time, retardation time and mobility parameter on the flow development and on the critical swirl for the appearance of vortex breakdown are explored. Decreasing the ratio of the viscoelastic characteristic times from one increases the critical swirl for breakdown. Increasing the Weissenberg number from zero or increasing the fluid mobility parameter from zero cause a similar effect. Results may explain changes in the appearance of breakdown zones as a function of swirl level that were observed in Stokes *et al.* (2001) experiments, where Boger fluids were used.

9:31AM G8.00008 Subcritical Transition to Turbulence in Couette-Poiseuille flow, JOSE EDUARDO WESFREID, LUKASZ KLOTZ, PMMH/ESPCI (Paris-France) — We study the subcritical transition to turbulence in the plane Couette-Poiseuille shear flow with zero mean advection velocity. Our experimental configuration consists on one moving wall of the test section (the second one remains stationary), which acts like a driving force for the flow, imposing linear streamwise velocity profile (Couette) and adverse pressure gradient in the streamwise direction (Poiseuille) at the same time. This flow, which had only been studied theoretically up to now, is always linearly stable. The transition to turbulence is forced by a very well controlled finite-size perturbation by injection, into the test section, of a water jet during a very short time. Using PIV technique, we characterized quantitatively the initial development of the triggered turbulent spot and compared its energy evolution with the theoretical predictions of the transient growth theory. In addition, we show results concerning the importance of nonlinearities, when waviness of streaks in streamwise direction induced self-sustained process in the turbulent spot. We also measured precisely the large-scale flow which is generated around the turbulent spot and studied its strength as a function of the Reynolds number.

9:44AM G8.00009 Experimental and numerical study of direct laminar-turbulent transition in Taylor-Couette flow¹, CHRISTOPHER J. CROWLEY, MICHAEL KRYGIER, Georgia Institute of Technology, DANIEL BORRERO-ECHEVERRY, Willamette University, ROMAN O. GRIGORIEV, MICHAEL F. SCHATZ, Georgia Institute of Technology — The transition to turbulence in large aspect ratio Taylor-Couette flow (TCF) occurs via a sequence of supercritical bifurcations of stable flow states (e.g. spiral vortices, interpenetrating spirals (IPS), and wavy interpenetrating spirals). We previously reported the discovery of a direct laminar-turbulent transition in a TCF system with counter-rotating cylinders ($Re_o = -1000$, $Re_i \approx 640$) and a small aspect ratio ($\Gamma = 5.26$) as Re_i is slowly increased. This transition is mediated by an unstable IPS state. As Re_i is decreased, the turbulent flow first relaminarizes into an intermediate, stable IPS state, before returning to circular Couette flow. In this talk we will present the study of this transition experimentally using tomographic PIV and direct numerical simulations with realistic boundary conditions, and show that it is both highly repeatable and that it shows hysteresis. The transition between both the IPS and turbulent states exhibits statistics consistent with chaotic attractor transitioning to a chaotic repeller. The IPS state is accessed from a subcritical transition and is inaccessible when the inner cylinder is originally accelerated on the way up to turbulence, suggesting that a finite amplitude perturbation is required to reach it.

¹This work is supported in part by the Army Research Office (Contract W911NF-16-1-0281)

9:57AM G8.00010 Connections between large-domain Newtonian turbulence and minimal-channel exact coherent states¹, ANUBHAV KUSHWAHA, MICHAEL GRAHAM, Univ of Wisconsin, Madison — Direct numerical simulations (DNS) of plane Poiseuille flow of a Newtonian fluid are performed in a large domain at transitional Reynolds numbers. In this Reynolds number regime, turbulent trajectories in minimal channels move chaotically between lower and upper branch invariant solutions known as exact coherent states (ECS). It is found that while they spend most of the time in a core region of the state space, fluctuating about the upper branch ECS, they occasionally escape the core region and pass through the vicinity of lower branch solutions. One particular set of the lower branch solutions form the lower bound of the turbulent trajectory with regard to flow properties like wall shear stress, energy dissipation rate and turbulent kinetic energy. We compare the evolution of wall shear stress in minimal channels with those in patches the size of minimal channels in a large domain and find that they are not only indistinguishable but also bounded on the lower end by the same set of lower branch ECS. This suggests that localised regions in a large box approach the travelling wave solutions in a way similar to minimal channels. We also show that low and high drag regions occur spatiotemporally when the turbulence trajectory approaches the lower and upper branch ECS, respectively.

¹The work was supported by AFOSR grant FA9550-15-1-0062.

Monday, November 21, 2016 8:00AM - 10:10AM –
Session G9 General Fluid Dynamics: Theory B117 - James Hanna, Virginia Polytechnic Institute and State University

8:00AM G9.00001 Bending and stretching of two-dimensional fluids and solids, JAMES HANNA, Virginia Polytechnic Institute and State University — Soap films, lipid membranes, and elastic sheets are often analyzed with similar (idealized) models that emphasize the geometric features of these surfaces. If deformations of these surfaces are area-preserving, simple and elegant expressions may be used to describe surface and bending energies and the corresponding equations of equilibrium. However, in general, one should make a distinction between geometric energies, as measured per unit area, and elastic energies, as measured per unit mass. I will discuss some of the differences between these types of energies, and the resulting potential difficulties and inelegancies in their mathematical descriptions.

8:13AM G9.00002 Peeling, sliding, pulling and bending, JOHN LISTER, GUNNAR PENG, University of Cambridge — The peeling of an elastic sheet away from thin layer of viscous fluid is a simply-stated and generic problem, that involves complex interactions between the flow and elastic deformation on a range of length scales. Consider an analogue of capillary spreading, where a blister of injected viscous fluid spreads due to tension in the overlying elastic sheet. Here the tension is coupled to the deformation of the sheet, and thus varies in time and space. A key question is whether or not viscous shear stresses ahead of the blister are sufficient to prevent the sheet sliding inwards and relieving the tension. Our asymptotic analysis reveals a dichotomy between fast and slow spreading, and between two-dimensional and axisymmetric spreading. In combination with bending stresses and gravity, which may dominate parts of the flow but not others, there is a plethora of dynamical regimes.

8:26AM G9.00003 Fluid-driven fracturing of adhered elastica: evolution of the vapour tip, THOMASINA V. BALL, BP Institute, Department of Earth Sciences, University of Cambridge, JEROME A. NEUFELD, BP Institute, Department of Earth Sciences, Department of Applied Mathematics and Theoretical Physics, University of Cambridge — The transient spreading of a viscous fluid beneath an elastic sheet is controlled by the dynamics at the tip. The large negative pressures needed to drive the viscous fluid into the narrowing gap necessitates a vapour tip separating the fluid front and the crack tip. Adhesion of the elastic sheet imposes a curvature at the tip giving rise to an elasto-capillary length scale and the possibility of a balance between elastic deformation and the strength of adhesion. Two dynamical regimes are therefore possible; viscosity dominant spreading controlled by the pressure in the vapour tip and adhesion dominant spreading controlled by interfacial adhesion. A series of constant flux experiments using clear PDMS elastic sheets allow for direct measurement of the vapour tip in the bending (thick sheet) limit. For small fluid fluxes, the experimental results can be explained by a constant interior pressure and a viscous boundary layer near the fluid front and result in an asymptotic model for the advance of adhesion and viscosity dominated fracture fronts resolving the vapour tip. Understanding the fluid-driven fracturing of adhered elastica provides insight into the spreading of shallow magmatic intrusions in the Earth's crust, and the fluid-driven fracturing of elastic media more generally.

8:39AM G9.00004 Finite time singularities in the quasigeostrophic model, RICHARD SCOTT, northwest research associates — A finite-time singularity in the evolution of a patch of surface temperature in the quasi-geostrophic equations via two distinct evolution routes is investigated with a grid-free, adaptive numerical scheme. In one case, the singularity proceeds through the formation of a corner, developing infinite curvature in the patch boundary in finite time. The corner is self-similar and the growth of curvature appears to be independent of initial patch shape. In the other case, the singularity proceeds through a self-similar cascade of filament instabilities with geometrically shrinking spatial and temporal scales, and the filament width approaches zero in a finite time. The spatially and temporally adaptive numerical scheme permits the accurate simulation of both corner and cascade singularities over a range of spatial scales spanning ten orders of magnitude. Some aspects of both singularity types exhibit universality, being independent of the initial patch shape and large-scale evolution; however, a simple extension of the initial temperature structure provides evidence that only the instability cascade persists in the case of a continuous initial temperature distribution.

8:52AM G9.00005 The Lorentz gas in Kaluza's MHD: Transport equations¹, ALFREDO SANDOVAL-VILLALBAZO, ALMA ROCIO SAGACETA-MEJIA, JOSE HUMBERTO MONDRAGON-SUAREZ, Department of Physics and Mathematics, U. Iberoamericana — Relativistic kinetic theory is applied to the study of the transport processes present in a Lorentz gas, using a geometric five-dimensional space-time. While the conventional transport equations are recovered in the Newtonian limit, it is shown that relativistic corrections to the conduction and diffusion fluxes arise within this formalism. A brief review of the conceptual advantages of the Kaluza-type approach to magnetohydrodynamics is also given.

¹The authors acknowledge support from CONACyT through grant CB2011/167563

9:05AM G9.00006 Density Stoke's Law for Particles having the Density Lower than the Surrounding Medium, ARJUN KRISHNAPPA, None — It has been observed from our experiments that Stoke's Law can be used only when the particle density is greater than the surrounding medium. When a microbubble is horizontally steered in a liquid, then the Stoke's Law can't be used to calculate the velocity or drag force. The reason underlying is that the density of the microbubble is lower than the density of the liquid. To overcome the problem, a modified Stoke's Law called "Density Stoke's Law (DSL)" is proposed. DSL works not only for the particles having the density lower than the surrounding medium, but also for the particles having the density greater than the surrounding medium. Therefore DSL can be considered as a general Stoke's Law.

9:18AM G9.00007 Investigation of Dalton and Amagats laws for gas mixtures with shock propagation¹, PATRICK WAYNE, IGNACIO TRUEBA MONJE, JASON H. YOO, C. RANDALL TRUMAN, PETER VOROBIEFF, University of New Mexico — Two common models describing gas mixtures are Daltons Law and Amagats Law (also known as the laws of partial pressures and partial volumes, respectively). Our work is focused on determining the suitability of these models to prediction of effects of shock propagation through gas mixtures. Experiments are conducted at the Shock Tube Facility at the University of New Mexico (UNM). To validate experimental data, possible sources of uncertainty associated with experimental setup are identified and analyzed. The gaseous mixture of interest consists of a prescribed combination of disparate gases – helium and sulfur hexafluoride (SF_6). The equations of state (EOS) considered are the ideal gas EOS for helium, and a virial EOS for SF_6 . The values for the properties provided by these EOS are then used used to model shock propagation through the mixture in accordance with Dalton's and Amagat's laws. Results of the modeling are compared with experiment to determine which law produces better agreement for the mixture.

¹This work is funded by NNSA grant DE-NA0002913

9:31AM G9.00008 Center of mass velocity during diffusion: Comparisons of fluid and kinetic models¹, ERIK VOLD, LIN YIN, WILLIAM TAITANO, KIM MOLVIG, B. J. ALBRIGHT, Los Alamos Natl Lab — We examine the diffusion process between two ideal gases mixing across an initial discontinuity by comparing fluid and kinetic model results and find several similarities between ideal gases and plasma transport. Binary diffusion requires a net zero species mass flux in the Lagrange frame to assure momentum conservation in collisions. Diffusion between ideal gases is often assumed to be isobaric and isothermal which requires constant molar density. We show this condition exists only in the lab frame at late times (many collision times) after a pressure transient relaxes. The sum of molar flux across an initial discontinuity is non-zero for species of differing atomic masses resulting in a pressure perturbation. The results show three phases of mixing: a pressure discontinuity forms across the initial interface (times of a few collisions), pressure perturbations propagate away from the mix region (time scales of an acoustic transit) and at late times characteristic of the diffusion process, the pressure relaxes leaving a non-zero center of mass flow velocity. The center of mass velocity associated with the outward propagating pressure waves is required to conserve momentum in the rest frame. Implications are considered in multi-species diffusion numerics and in applications.

¹Work performed under the auspices of the U.S. DOE by the LANS, LLC, Los Alamos National Laboratory under Contract No. DE-AC52-06NA25396. Funding provided by the Advanced Simulation and Computing (ASC) Program.

9:44AM G9.00009 Preserving the Helmholtz dispersion relation: One-way acoustic wave propagation using matrix square roots, LAURENCE KEEFE, Desondes LLC, West Linn, OR — Parabolized acoustic propagation in transversely inhomogeneous media is described by the operator update equation $U(x, y, z + \Delta z) = e^{ik_0(-1+\sqrt{1+\tilde{Z}})} U(x, y, z)$ for evolution of the envelope of a wavetrain solution to the original Helmholtz equation. Here the operator, $\tilde{Z} = \nabla_T^2 + (n^2 - 1)$, involves the transverse Laplacian and the refractive index distribution. Standard expansion techniques (on the assumption $\tilde{Z} \ll 1$) produce pdes that approximate, to greater or lesser extent, the full dispersion relation of the original Helmholtz equation, except that none of them describe evanescent/damped waves without special modifications to the expansion coefficients. Alternatively, a discretization of both the envelope and the operator converts the operator update equation into a matrix multiply, and existing theorems on matrix functions demonstrate that the complete (discrete) Helmholtz dispersion relation, including evanescent/damped waves, is preserved by this discretization. Propagation-constant/damping-rates contour comparisons for the operator equation and various approximations demonstrate this point, and how poorly the lowest-order, textbook, parabolized equation describes propagation in lined ducts.

9:57AM G9.00010 A Second Order Continuum Theory of Fluids – Beyond the Navier-Stokes Equations, SAMUEL PAOLUCCI, Univ of Notre Dame — The Navier-Stokes equations have proved very valuable in modeling fluid flows over the last two centuries. However, there are some cases where it has been demonstrated that they do not provide accurate results. In such cases, very large variations in velocity and/or thermal fields occur in the flows. It is recalled that the Navier-Stokes equations result from linear approximations of constitutive quantities. Using continuum mechanics principles, we derive a second order constitutive theory that application of which should provide more accurate results in such cases. One important case is the structure of gas-dynamic shock waves. It has been demonstrated experimentally that the Navier-Stokes formulation yields incorrect shock profiles even at moderate Mach numbers. Current continuum theories, and indeed most statistical mechanics theories, that have been advanced to reconcile such discrepancies have not been fully successful. Thus, application of the second order theory based solely on a continuum formulation provides an excellent test problem. Results of the second-order equations applied to the shock structure are obtained for monatomic and diatomic gases over a large range of Mach numbers and are compared to experimental results.

Monday, November 21, 2016 8:00AM - 10:10AM –
Session G10 DFD GPC: Convection and Buoyancy Driven Flows: Turbulent Convection B118-119 - Joerg Schumacher, Technische Universitat Ilmenau

8:00AM G10.00001 High-amplitude dissipation event in a turbulent convection cell, JANET SCHEEL, Occidental College, Los Angeles (USA), JOERG SCHUMACHER, TU Ilmenau (Germany) — A high-amplitude dissipation event in the bulk of a closed three-dimensional turbulent convection cell is found to be correlated with a strong reduction of the large-scale circulation flow in the system that happens at the same time as a plume emission event from the bottom plate. The reduction in the large-scale circulation allows for a nearly frontal collision of down- and upwelling plumes and the generation of a high-amplitude thermal dissipation layer in the bulk. This collision is locally connected to a subsequent high-amplitude energy dissipation event in the form of a strong shear layer. Our analysis illustrates the impact of transitions in the large-scale structures on extreme events at the smallest scales of the turbulence.

8:13AM G10.00002 Local patches of turbulent boundary layer behaviour in classical-state vertical natural convection, CHONG SHEN NG, ANDREW OOI, The University of Melbourne, DETLEF LOHSE, Physics of Fluids Group, Faculty of Science and Technology, J. M. Burgers Center for Fluid Dynamics and MESA+ Institute, University of Twente, DANIEL CHUNG, The University of Melbourne — We present evidence of local patches in vertical natural convection that are reminiscent of Prandtl–von Kármán turbulent boundary layers, for Rayleigh numbers 10^5 – 10^9 and Prandtl number 0.709. These local patches exist in the classical state, where boundary layers exhibit a laminar-like Prandtl–Blasius–Polhausen scaling at the global level, and are distinguished by regions dominated by high shear and low buoyancy flux. Within these patches, the locally averaged mean temperature profiles appear to obey a log-law with the universal constants of Yaglom (1979). We find that the local Nusselt number versus Rayleigh number scaling relation agrees with the logarithmically corrected power-law scaling predicted in the ultimate state of thermal convection, with an exponent consistent with Rayleigh–Bénard convection and Taylor–Couette flows. The local patches grow in size with increasing Rayleigh number, suggesting that the transition from the classical state to the ultimate state is characterised by increasingly larger patches of the turbulent boundary layers.

8:26AM G10.00003 Turbulent convection driven by internal radiative heating of melt ponds on sea ice, ANDREW WELLS, TOM LANGTON, DAVID REES JONES, University of Oxford, WOOSOK MOON, University of Cambridge — The melting of Arctic sea ice is strongly influenced by heat transfer through melt ponds which form on the ice surface. Melt ponds are internally heated by the absorption of incoming radiation and cooled by surface heat fluxes, resulting in vigorous buoyancy-driven convection in the pond interior. Motivated by this setting, we conduct two-dimensional direct-numerical simulations of the turbulent convective flow of a Boussinesq fluid between two horizontal boundaries, with internal heating predicted from a two-stream radiation model. A linearised thermal boundary condition describes heat exchange with the overlying atmosphere, whilst the lower boundary is isothermal. Vertically asymmetric convective flow modifies the upper surface temperature, and hence controls the partitioning of the incoming heat flux between emission at the upper and lower boundaries. We determine how the downward heat flux into the ice varies with a Rayleigh number based on the internal heating rate, the flux ratio of background surface cooling compared to internal heating, and a Biot number characterising the sensitivity of surface fluxes to surface temperature. Thus we elucidate the physical controls on heat transfer through Arctic melt ponds which determine the fate of sea ice in the summer.

8:39AM G10.00004 Effects of Exit Variability on Near-Field Statistics for Turbulent Buoyant Jets, NICHOLAS WIMER, CAELAN LAPOINTE, TORREY HAYDEN, JASON CHRISTOPHER, GREGORY RIEKER, PETER HAMILTON, Univ of Colorado - Boulder — Many engineering systems involve the use of high-temperature jets to heat nearby objects or surfaces. In such instances, proximity to the jet exit means that specific properties of the exit velocity and temperature can be of substantial importance in determining conditions at the heated object or surface. Moreover, compared to non-heated jets, the flow field complexity of high-temperature jets is substantially increased due to the presence of buoyant forcing. In this talk, we examine the effects of variability in exit velocity and temperature on near-field flow statistics of a high-temperature turbulent buoyant jet. The analysis is based on large eddy simulations (LES) of turbulent buoyant jets for a variety of velocity and temperature exit distributions, including uniform, pseudo-random, and Gaussian distributions with different means and standard deviations. The resulting near-field turbulent statistics are compared to properties of the exit distributions, with a specific focus on predicting spatial and temporal spectral exponents for velocity and temperature in the near-field. The importance of these results for the prediction and understanding of engineering applications involving high-temperature jets is outlined.

8:52AM G10.00005 The Turbulent Diffusivity of Convective Overshoot, DANIEL LECOANET, Princeton, JOSIAH SCHWAB, UC-Santa Cruz, ELIOT QUATAERT, UC-Berkeley, LARS BILDSTEN, UC-Santa Barbara, FRANK TIMMES, Arizona State University, KEATON BURNS, MIT, GEOFFREY VASIL, University of Sydney, JEFFREY OISHI, Bates College, BEN-JAMIN BROWN, University of Colorado - Boulder — There are many natural systems with convectively unstable fluid adjacent to stably stratified fluid; including the Earth's atmosphere, most stars, and perhaps even the Earth's liquid core. The convective motions penetrating into the stable region can enhance mixing, leading to changes in transport within the stable region. This work describes convective overshoot simulations. To study the extra mixing due to overshoot, we evolve a passive tracer field. The horizontal average of the passive tracer quickly approaches a self-similar state. The self-similar state is the solution to a diffusion equation with a spatially dependent turbulent diffusivity. We find the extra mixing due to convection can be accurately modeled as a turbulent diffusivity, and discuss implications of this turbulent diffusivity for the astrophysical problem of mixing in convectively bounded carbon flames.

9:05AM G10.00006 Sustained shear flows in Rayleigh-Bnard convection , TAYLER QUIST,

EVAN ANDERS, BENJAMIN BROWN, University of Colorado, JEFFREY OISHI, Bates College — Zonal shear flows play important roles in both the solar and geo dynamos. In two dimensional simulations, and at relatively narrow aspect ratios, Rayleigh-Bnard convection naturally achieves zonal shear flows. These zonal flows are driven by the convection and modify it, significantly altering the heat transport and convective structures. Here we study shear flows in two and three-dimensional simulations of Rayleigh-Bnard convection using the Dedalus pseudospectral framework. At small aspect ratios and at Prandtl number 1, a large horizontal shear naturally occurs. At larger aspect ratios, we find that shearing is naturally prevented unless manually induced; there is a bistability between states dominated by flywheel modes and states dominated by large scale shear. We explore these states and the possibilities of sustained large scale shear in 3-D simulations.

9:18AM G10.00007 Non-adiabatic Rayleigh-Taylor instability¹ , JESSE CANFIELD, NICHOLAS

DENISSEN, JON REISNER, Los Alamos National Laboratory — Onset of Rayleigh-Taylor instability (RTI) in a non-adiabatic environment is investigated with the multi-physics numerical model, FLAG. This work was inspired by laboratory experiments of non-adiabatic RTI, where a glass vessel with a layer of tetrahydofuran (THF) below a layer of toluene was placed inside a microwave. THF, a polar solvent, readily absorbs electromagnetic energy from microwaves. Toluene, a non-polar solvent, is nearly transparent to microwave heating. The presence of a heat source in the THF layer produced convection and a time-dependent Atwood number (A_t). The system, initially in stable hydrostatic equilibrium $A_t < 0$, was set into motion by microwave induced, volumetric heating of the THF. The point when $A_t > 0$, indicates that the system is RTI unstable. The observed dominant mode at the onset of RTI was the horizontal length scale of the vessel. This scale is contrary to classical RTI, where the modes start small and increases in scale with time. It is shown that the dominant RTI mode observed in the experiments was determined by the THF length scale prior to RTI. The dominant length scale transitions from the THF to the toluene via the updrafts and downdrafts in the convective cells. This happens when A_t passes from negative to positive.

¹This work was funded by the Advanced Simulation and Computing Program

9:31AM G10.00008 Scale-to-scale energy and enstrophy transport in two-dimensional Rayleigh-Taylor turbulence¹ , QUAN ZHOU, Shanghai Institute of Applied Mathematics and Mechanics, Shanghai University — We apply a recently developed filtering approach, i.e. filter-space technique (FST), to study the scale-to-scale transport of kinetic energy, thermal energy, and enstrophy in two-dimensional (2D) Rayleigh-Taylor (RT) turbulence. Although the scaling laws of the energy cascades in 2D RT system follow the Bolgiano-Obukhov (BO59) scenario due to buoyancy forces, the kinetic energy is still found to be, on average, dynamically transferred to large scales by an inverse cascade, while both the mean thermal energy and the mean enstrophy move toward small scales by forward cascades. In particular, there is a reasonably extended range over which the transfer rate of thermal energy is scale-independent and equals the corresponding thermal dissipation rate at different times. This range functions similar to the inertial range for the kinetic energy in the homogeneous and isotropic turbulence. Our results further show that at small scales the fluctuations of the three instantaneous local fluxes are highly asymmetrically distributed and there is a strong correlation between any two fluxes. These small-scale features are signatures of the mixing and dissipation of fluids with steep temperature gradients at the fluid interfaces.

¹This work was supported by NSFC under Grant No. 11572185.

9:44AM G10.00009 Investigation of turbulent Prandtl number subject to local acceleration and deceleration , EUNBEO JUNG, WOOK LEE, SEONGWON KANG¹, Dept. of Mechanical Engineering, Sogang University, Korea, GIANLUCA IACCARINO, Dept. of Mechanical Engineering, Stanford University, USA — The main objective of the present study is to analyze the turbulent Prandtl number (Pr_t) varying over space in a wall-bounded turbulent flow under local acceleration and deceleration. The Pr_t shows the opposite trends for the conditions of acceleration and deceleration. In order to explain these phenomena, the convection velocity from the space-time correlation is investigated. It is shown that small-scale motions experience larger acceleration and deceleration compared to large-scale ones. Also, a discrepancy between the momentum and heat transfer at small scales results in the spatially varying Pr_t . The budgets of the turbulent kinetic energy and temperature variance show a hint for the variation of Pr_t . The results from DNS and RANS with a constant Pr_t are compared and show that RANS prediction can be improved by using a modeled Pr_t . From the turbulent statistics, a few flow variables showing higher correlations with Pr_t are identified. Based on this, simple phenomenological models are devised and the corresponding simulations show a more accurate prediction of the heat transfer rate.

¹Corresponding author

9:57AM G10.00010 Heat and momentum transfer for magnetoconvection in a vertical external magnetic field¹ , TILL ZÜRNER, WENJUN LIU, DMITRY KRASNOV, JÖRG SCHUMACHER, Technische Universität Ilmenau — The scaling theory of Grossmann and Lohse (J. Fluid Mech. 407, 27 (2000)) for the turbulent heat and momentum transfer is extended to the magnetoconvection case in the presence of a (strong) vertical magnetic field. The comparison with existing laboratory experiments and direct numerical simulations in the quasistatic limit allows to restrict the parameter space to very low Prandtl and magnetic Prandtl numbers and thus to reduce the number of unknown parameters in the model. Also included is the Chandrasekhar limit for which the outer magnetic induction field \mathbf{B} is large enough such that convective motion is suppressed and heat is transported by diffusion. Our theory identifies four distinct regimes of magnetoconvection which are distinguished by the strength of the outer magnetic field and the level of turbulence in the flow, respectively.

¹LIMTECH Research Alliance and Research Training Group GK 1567 on Lorentz Force Velocimetry, funded by the Deutsche Forschungsgemeinschaft

Monday, November 21, 2016 8:00AM - 10:10AM –
Session G11 Jets: General and Supersonic C120-121-122 - Antonio Sanchez, University of California
Sand Diego

8:00AM G11.00001 Counterflow laminar gas jets issuing from aligned planar nozzles

, ADAM WEISS, WILFRIED COENEN, ANTONIO L SANCHEZ, University of California San Diego — The laminar flow resulting from the impingement of two gaseous jets issuing from aligned planar nozzles of semi-width R separated a distance $2H$ is investigated by numerical and analytical methods, with specific consideration given to the high-Reynolds and low-Mach number conditions typically present in counterflow-flame experiments. The resulting flow, nearly inviscid and effectively incompressible, can be described by using a density-weighted streamfunction/vorticity formulation that removes the need to consider specifically the boundary separating the two jet streams. Besides the geometric parameter H/R , the solution depends only on the shape of the velocity profiles in the feed streams and on the jet momentum-flux ratio Λ . While conformal mapping can be used to determine the potential solution corresponding to uniform feed-stream velocity profiles, numerical integration is required in general to compute vortical flows, including those arising with Poiseuille velocity profiles, with simplified solutions found in the limits $H/R \ll 1$ and $H/R \gg 1$.

8:13AM G11.00002 On plane submerged laminar jets, WILFRIED COENEN, ANTONIO L SANCHEZ,

University of California San Diego — We address the laminar flow generated when a developed stream of liquid of kinematic viscosity ν flowing along channel of width $2h$ discharges into an open space bounded by two symmetric plane walls departing from the channel rim with an angle $\alpha \sim 1$. Attention is focused on values of the jet volume flux $2Q$ such that the associated Reynolds number $Re = Qh/\nu$ is of order unity. The formulation requires specification of the boundary conditions far from the channel exit. If the flow is driven by the volume flux, then the far-field solution corresponds to Jeffery-Hamel self-similar flow. However, as noted by Fraenkel (1962), such solutions exist only for $\alpha < 129^\circ$ in a limited range of Reynolds numbers $0 \leq Re \leq Re_c(\alpha)$ (e.g. $Re_c \approx 1.43$ for $\alpha = \pi/2$). It is reasoned that an alternative solution, driven by a fraction of the momentum flux of the feed stream, may also exist for all values of Re and α , including a near-centerline Bickley jet, a surrounding Taylor potential flow driven by the jet entrainment, and a Falkner-Skan near-wall boundary layer. Numerical integrations of the Navier-Stokes equations are used to ascertain the existence of these different solutions.

8:26AM G11.00003 PIV measurements of in-cylinder, large-scale structures in a water-analogue Diesel engine¹, A. KALPAKLI VESTER, KTH Mechanics, Royal Institute of Technology, Y. NISHIO,

Tohoku University, Sendai, P.H. ALFREDSSON, KTH Mechanics, Royal Institute of Technology — Swirl and tumble are large-scale structures that develop in an engine cylinder during the intake stroke. Their structure and strength depend on the design of the inlet ports and valves, but also on the valve lift history. Engine manufacturers make their design to obtain a specific flow structure that is assumed to give the best engine performance. Despite many efforts, there are still open questions, such as how swirl and tumble depend on the dynamics of the valves/piston as well as how cycle-to-cycle variations should be minimized. In collaboration with Swedish vehicle industry we perform PIV measurements of the flow dynamics during the intake stroke inside a cylinder of a water-analogue engine model having the same geometrical characteristics as a typical truck Diesel engine. Water can be used since during the intake stroke the flow is nearly incompressible. The flow from the valves moves radially outwards, hits the vertical walls of the cylinder, entrains surrounding fluid, moves along the cylinder walls and creates a central backflow, i.e. a tumble motion. Depending on the port and valve design and orientation none, low, or high swirl can be established. For the first time, the effect of the dynamic motion of the piston/values on the large-scale structures is captured.

¹Supported by the Swedish Energy Agency, Scania CV AB and Volvo GTT, through the FFI program.

8:39AM G11.00004 Large eddy simulations of the flow field of a radially lobed nozzle

, NOUSHIN AMINI, Texas AM University, AARTHI SEKARAN, The University of Texas at Austin — Lobed nozzles have been studied over the past couple of decades due to their enhanced mixing capabilities. Despite experimental (Hu et al, 2000) and numerical studies (Cooper et al, 2005), the nature of the jet is yet to be fully understood. This numerical study intends to carry out a thorough analysis of the flow field within and downstream of a six lobed nozzle. The study aims to confirm vortical interaction mechanisms and establish the role of hydrodynamic instabilities in the mixing process. This was inspired by a prior study by the authors wherein the same flow was studied using hot-wire anemometry. Although this helped obtain a qualitative idea of the flow, the 2D data was incapable of visualizing streamwise structures and the flow within the nozzle. Previous numerical simulations have used RANS and to simulate a single lobe of the nozzle; these results show some deficiencies in predicting the potential core length. Previous simulations done by authors indicated that RANS models qualitatively capture the flow structures but do not accurately represent the values of key parameters in the flow field. The present study aims to perform a 3D LES study of the flow field within and downstream of the nozzle to follow the ensuing free jet and thus analyze various mechanisms.

8:52AM G11.00005 Mathematical design of fluidic jets, ADAM HAQUE, MARCUS ROPER, SARAH HAKIM, KAITLYN HOOD, UCLA, UCLA MYCO-FLUIDICS LAB TEAM — Recent experimental work has revealed the possibility of sculpting fluid jets by passing them through a sequence of fluid posts. Simplifying the transformation that occurs when a jet passes a post at finite Reynolds number, we ask: Are there rational ways to tailor the sequence of transformations to achieve a given jet shape? Studying these simplified transformations reveals fundamental constraints on what shape changes can be produced by single transformations, and suggests strategies for optimizing combinations of transformations to achieve desired jet shapes.**9:05AM G11.00006 Multi-Measurement Correlations in the Near-Field of a Complex Supersonic Jet Using Time-Resolved Schlieren Imaging**, ANDREW TENNEY, THOMAS COLEMAN, JACQUES LEWALLE, MARK GLAUSER, Syracuse Univ, SIVARAM GOGINENI, Spectral Energies, LLC — Supersonic flow from a three-stream non-axisymmetric jet is visualized using time resolved schlieren photography (up to 400,000 frames per second) while pressure on the aft deck plate of the nozzle is simultaneously sampled using kulites. Time series are constructed using the schlieren photographs and conditioned to reduce the effects of signal drift and clipping where necessary. The effect of this detrending and clipping reconstruction on signal statistics is examined. In addition, signals constructed from near field schlieren will be correlated with one another to visualize the propagation of information in the near field. The goal of utilizing space-time correlations is to assist in identifying and tracking the evolution of individual structures in the near field. The schlieren signals will also be correlated with the deck pressure traces to assist in unraveling the interaction of flow structures.

9:18AM G11.00007 Time-resolved schlieren POD and aft deck pressure correlations on a rectangular supersonic nozzle and sonic wall jet¹, MATTHEW BERRY, ANDREW MAGSTADT, Syracuse Univ, CORY STACK, DATTA GAITONDE, Ohio State Univ, MARK GLAUSER, Syracuse Univ — A multi-stream single expansion ramp nozzle (SERN) with aft deck, based on three-stream engine concepts, is currently undergoing experimental tests at Syracuse University's Skytop Turbulence Laboratory. In the context of this study, we view this as an idealized representation consisting of two canonical flows; a supersonic convergent-divergent (CD) nozzle and a sonic wall jet (representing the 3rd stream). The jet operates at a bulk flow of $M_{j,1} = 1.6$ and wall jet $M_{j,3} = 1.0$. Proper orthogonal decomposition (POD) is then performed on the schlieren images and the time-dependent coefficients are related to the near-field deck pressure. Structures within the flow field are correlated to particular flow events and help track the downstream evolution of the jet. A multitude of scales are seen within the flow corresponding to a wide range of coherent structures. High fidelity LES is also performed on the same nozzle geometry and relations are made back to the experiments.

¹AFOSR Turbulence and Transition Program (Grant No. FA9550-15-1-0435) with program managers Dr. I. Leyva and Dr. R. Ponnappan

9:31AM G11.00008 Dynamics of the Coherent Structures in a Supersonic Rectangular Jet of Aspect Ratio 2, KAMAL VISWANATH, ANDREW CORRIGAN, RYAN JOHNSON, KAZHIKATHRA KAILASANATH, Naval Research Lab, EPHRAIM GUTMARK, University of Cincinnati, UNIVERSITY OF CINCINNATI TEAM, LABORATORIES FOR COMPUTATIONAL PHYSICS AND FLUID DYNAMICS TEAM — Asymmetric exhaust nozzle configurations, in particular rectangular, are likely to become more important in the future for both civilian and military aircraft. Various nozzle geometry features including the presence of sharp corners impact the evolution of the cross-sectional shape of the jet and its mixing features. Asymmetric nozzles potentially offer a passive way of affecting mixing for low aspect ratio jets through both large-scale entrainment due to coherent structures and fine scale mixing at the corners. Data is presented that show the dynamic evolution of the coherent structures for an ideally expanded rectangular nozzle of aspect ratio 2. The sense of the vortex pairs setup through the self-induction at the corners and stretching of the azimuthal vortex ring into streamwise vortices results in diagonal elongation of the time-averaged jet cross-section and contraction at the sides. The phase averaged velocity contours further clearly show the effect of mixing at the sharp corners and the deformation of the rectangular exit cross-section as it propagates downstream. It is observed that the dominant vortex pairs in this case work against axis-switching.

9:44AM G11.00009 Flow and Noise Characteristics of Under- and Over-expanded Supersonic Rectangular Jets of Aspect Ratio 2, RYAN JOHNSON, KAMAL VISWANATH, ANDREW CORRIGAN, KAZHIKATHRA KAILASANATH, Naval Research Lab, EPHRAIM GUTMARK, University of Cincinnati — Simulations of under- and over-expanded flow for two operating temperatures were conducted with a low aspect ratio, AR = 2.0, rectangular nozzle. These cases are compared to the same nozzle at the design pressure ratio. The simulated acoustic data are validated against experimentally recorded sound pressure level (SPL) spectra. The axial flow structure is examined along parallel lines that originate at the center and nozzle walls in the direction of the core flow. The shock cell structure, jet core length, and the axial distributions of the flow are different in all observed planes for the over-, under-, and ideally-expanded jet flow cases. How these flow structures contribute to SPL and overall sound pressure levels (OASPL) is discussed.

9:57AM G11.00010 Coherent structures in a supersonic complex nozzle¹, ANDREW MAGSTADT, MATTHEW BERRY, MARK GLAUSER, Syracuse Univ — The jet flow from a complex supersonic nozzle is studied through experimental measurements. The nozzles geometry is motivated by future engine designs for high-performance civilian and military aircraft. This rectangular jet has a single plane of symmetry, an additional shear layer (referred to as a wall jet), and an aft deck representative of airframe integration. The core flow operates at a Mach number of $M_{j,c} = 1.6$, and the wall jet is choked ($M_{j,w} = 1.0$). This high Reynolds number jet flow is comprised of intense turbulence levels, an intricate shock structure, shear and boundary layers, and powerful corner vortices. In the present study, stereo PIV measurements are simultaneously sampled with high-speed pressure measurements, which are embedded in the aft deck, and far-field acoustics in the anechoic chamber at Syracuse University. Time-resolved schlieren measurements have indicated the existence of strong flow events at high frequencies, at a Strouhal number of $St = 3.4$. These appear to result from von Kármán vortex shedding within the nozzle and pervade the entire flow and acoustic domain. Proper orthogonal decomposition is applied on the current data to identify coherent structures in the jet and study the influence of this vortex street.

¹AFOSR Turbulence and Transition Program (Grant No. FA9550-15-1-0435) with program managers Dr. I. Leyva and Dr. R. Ponnappan

Monday, November 21, 2016 8:00AM - 10:10AM –
Session G12 Multiphase Flows: Diffused Interface, Finite Volume Methods and Applications C123 - Pedro Milani, Stanford University

8:00AM G12.00001 Parasitic Currents in Diffuse-Interface Two-Phase Flow Simulations¹, PEDRO MILANI, SEYEDSHAHABADDIN MIRJALILI, ALI MANI, Stanford University — Two phase flow phenomena are important in a wide range of applications, such as bubble generation in ocean waves and droplet dynamics in fuel injectors. Several methods can be used to simulate such phenomena. The focus of this study is the diffuse-interface method, in which the interface is described via a mixing energy and spans a few computational cells, while surface tension is modeled as a force density term on the right-hand side of the momentum equation. The advantages of this method include the ability to easily simulate complex geometries since it does not require special treatment around the interface, and to conserve mass exactly. However, this method suffers from parasitic currents, an unphysical velocity field generated close to the interface due to numerical imprecisions in the surface tension term. This can be a serious problem in low speed flows, where the parasitic currents are significant compared to the velocity scale of the problem. In this study, we consider a wide range of diffuse-interface schemes for two-phase flows, including different options for discrete representation of the surface tension force. By presenting an assessment of each method's performance in scenarios involving parasitic currents, we develop accuracy estimates and guidelines for selection among these models.

¹Supported by the ONR

8:13AM G12.00002 A compressible multiphase framework for simulating supersonic atomization, JONATHAN D. REGELE, DANIEL P. GARRICK, ZAHRA HOSSEINZADEH-NIK, MOHAMAD ASLANI, Iowa State University, MARK OWKES, Montana State University — The study of atomization in supersonic combustors is critical in designing efficient and high performance scramjets. Numerical methods incorporating surface tension effects have largely focused on the incompressible regime as most atomization applications occur at low Mach numbers. Simulating surface tension effects in high speed compressible flow requires robust numerical methods that can handle discontinuities caused by both material interfaces and shocks. A shock capturing/diffused interface method is developed to simulate high-speed compressible gas-liquid flows with surface tension effects using the five-equation model. This includes developments that account for the interfacial pressure jump that occurs in the presence of surface tension. A simple and efficient method for computing local interface curvature is developed and an acoustic non-dimensional scaling for the surface tension force is proposed. The method successfully captures a variety of droplet breakup modes over a range of Weber numbers and demonstrates the impact of surface tension in countering droplet deformation in both subsonic and supersonic cross flows.

8:26AM G12.00003 Multiphase flows in confinement with complex geometries, BENJAMIN AYMAR, Complex Multiscale Systems Group, Department of Chemical Engineering, Imperial College London, MARC PRADAS, Department of Mathematics and Statistics, Open University, Milton Keynes, URBAIN VAES, Department of Mathematics, Imperial College London, SERAFIM KALLIADASIS, Complex Multiscale Systems Group, Department of Chemical Engineering, Imperial College London — Understanding the dynamics of immiscible fluids in confinement is crucial in numerous applications such as oil recovery, fuel cells and the rapidly growing field of microfluidics. Complexities such as microstructures, chemical-topographical heterogeneities or porous membranes, can often induce non-trivial effects such as critical phenomena and phase transitions. The dynamics of confined multiphase flows may be efficiently described using diffuse-interface theory, leading to the Cahn-Hilliard-Navier-Stokes(CHNS) equations with Cahn wetting boundary conditions. Here we outline an efficient numerical method to solve the CHNS equations using advanced geometry-capturing mesh techniques both in two and three dimensional scenarios. The methodology is applied to two different systems: a droplet on a spatially chemical-topographical heterogeneous substrate and a microfluidic separator.

8:39AM G12.00004 Evaluating curvature for the volume of fluid method via interface reconstruction¹, FABIEN EVRARD, FABIAN DENNER, BEREND VAN WACHEM, Imperial College London — The volume of fluid method (VOF) is widely adopted for the simulation of interfacial flows. A critical step in VOF modelling is to evaluate the local mean curvature of the fluid interface for the computation of surface tension. Most existing curvature evaluation techniques exhibit errors due to the discrete nature of the field they are dealing with, and potentially to the smoothing of this field that the method might require. This leads to the production of inaccurate or unphysical results. We present a curvature evaluation method which aims at greatly reducing these errors. The interface is reconstructed from the volume fraction field and the curvature is evaluated by fitting local quadric patches onto the resulting triangulation. The patch that best fits the triangulated interface can be found by solving a local minimisation problem. Combined with a partition of unity strategy with compactly supported radial basis functions, the method provides a semi-global implicit expression for the interface from which curvature can be exactly derived. The local mean curvature is then integrated back on the Eulerian mesh. We show a detailed analysis of the associated errors and comparisons with existing methods. The method can be extended to unstructured meshes.

¹Financial support from Petrobras is gratefully acknowledged

8:52AM G12.00005 Computation of Two-Phase Flows with an Interface-Capturing Method on Arbitrarily-Shaped Polygonal Meshes, HIROSHI OTAKE, TAKESHI OMORI, TAKEO KAJISHIMA, Department of Mechanical Engineering, Osaka University — Arbitrarily-shaped polyhedral meshes are convenient for the computation of industrial flow systems which often have complex geometries. In two-phase flow problems, however, the employment of such meshes is rather challenging due to a poor accuracy of the existing methods based on the VOF (Volume of Fluid) method. In the previous work, we proposed an advection scheme for the interface indicator function on three-dimensional polyhedral meshes using the THINC (Tangent of Hyperbola Interface Capturing) method and a procedure to estimate interface curvatures on these meshes with the accuracy comparable to the conventional methods which employ structured meshes. To incorporate these schemes into the calculation of the Navier-Stokes equation, it is further required to eliminate the local numerical oscillation of the indicator function. In the present study, we discuss our recent improvement in the way to integrate the flux of the indicator function on cell faces and demonstrate its effectiveness in the calculation of two-phase flows performed on two-dimensional polygonal meshes.

9:05AM G12.00006 Numerical Investigation on Sensitivity of Liquid Jet Breakup to Physical Fuel Properties with Experimental Comparison¹, DOKYUN KIM, Cascade Technologies, LUIS BRAVO, Army Research Laboratory, KATARZYNA MATUSIK, DANIEL DUKE, ALAN KASTENGREN, ANDY SWANTEK, CHRISTOPHER POWELL, Argonne National Laboratory, FRANK HAM, Cascade Technologies — One of the major concerns in modern direct injection engines is the sensitivity of engine performance to fuel characteristics. Recent works have shown that even slight differences in fuel properties can cause significant changes in efficiency and emission of an engine. Since the combustion process is very sensitive to the fuel/air mixture formation resulting from disintegration of liquid jet, the precise assessment of fuel sensitivity on liquid jet atomization process is required first to study the impact of different fuels on the combustion. In the present study, the breaking process of a liquid jet from a diesel injector injecting into a quiescent gas chamber is investigated numerically and experimentally for different liquid fuels (n-dodecane, iso-octane, CAT A2 and C3). The unsplit geometric Volume-of-Fluid method is employed to capture the phase interface in Large-eddy simulations and results are compared against the radiography measurement from Argonne National Lab including jet penetration, liquid mass distribution and volume fraction. The breakup characteristics will be shown for different fuels as well as droplet PDF statistics to demonstrate the influences of the physical properties on the primary atomization of liquid jet.

¹supported by HPCMP FRONTIER award, US DOD, Office of the Army.

9:18AM G12.00007 A sharp interface in-cell-reconstruction method for volume tracking phase interfaces in compressible flows¹, DOMINIC KEDELTY, CARLOS BALLESTEROS, Arizona State University, RONALD CHAN, Stanford University, MARCUS HERRMANN, Arizona State University — To accurately predict the interaction of the interface with shocks and rarefaction waves, sharp interface methods maintaining the interface as a discontinuity are preferable to capturing methods that tend to smear the interface. We present a hybrid capturing/tracking method (Smiljanovski et al., 1997) that couples an unsplit geometric volume tracking method (Owkes & Desjardins, 2014) to a finite volume wave propagation scheme (LeVeque, 2010). In cells containing the phase interface, states on either side are reconstructed using the jump conditions across the interface, the geometric information of the volume tracking method, and the cell averages of the finite volume method. Cell face Riemann problems are then solved within each phase separately, resulting in area fraction weighted fluxes that update the cell averages directly. This, together with a linearization of the wave interaction across cell faces avoids the small cut-cell time step limitation of typical tracking methods. However, the interaction of waves with the phase interface cannot be linearized and is solved using either exact or approximate two-phase Riemann solvers with arbitrary jumps in the equation of state. Several test cases highlight the capabilities of the new method.

¹Support by the 2016 CTR Summer program at Stanford University and Taitech, Inc. under subcontract TS15-16-02-005 is gratefully acknowledged.

9:31AM G12.00008 Effect of Eccentricity in Compound Droplets Subject to a Simple Shear Flow., SANGKYU KIM, SADEGH DABIRI, Purdue University — A double emulsion, or a compound droplet, is a system where two liquids are separated by an immiscible third liquid, thereby forming an emulsion inside an emulsion. Compound drops benefit from this separation in applications such food sciences, microfluidics, pharmaceutical engineering, and polymer sciences. While the subjects of double emulsion preparations, deformations, and breakup mechanisms are well-explored, the time-evolution of non-concentric compound drops has received far less analytical or computational scrutiny. In this work, we present computational results using finite volume method with front-tracking approach for initially spherical and non-concentric compound drops in a shear flow. Our findings for low Reynolds number flows show that: 1. The surrounding shear flow to the outer drop induces a rotational velocity field inside it, causing the inner drop to tumble with the flow, 2. the tumbling motion persists in time, and acts to increase the eccentricity of the compound drop, and 3. the hemisection-plane to the outer drop that is aligned with the plane of the simple shear defines an unstable equilibrium for inner drop's center, and the inner drop continuously drifts away from that plane. This work suggests a means of favorably configuring compound drops suitable for breakups, and helps to understand their migration in channel flows.

9:44AM G12.00009 High-order positivity-satisfying scheme for multi-component flows, KHOSRO SHAHBAZI, South Dakota School of Mines and Technology — A high-order maximum-principle-satisfying scheme for the multi-component flow computations featuring jumps and discontinuities due to shock waves and phase interfaces is presented. The scheme is based on high-order weighted-essentially non-oscillatory (WENO) finite volume schemes and high-order limiters to ensure the maximum principle or positivity of the various field variables including the density, pressure, and order parameters identifying each phase. The two-component flow model considered besides the Euler equation of gas dynamics consists of advection of two parameters of the stiffened gas equation, characterizing each phase. The design of the high-order limiter is based on limiting the quadrature values of the density, pressure and order parameters reconstructed using a high-order WENO scheme. The convergence and the order of accuracy of the scheme is illustrated using the smooth isentropic vortex problem with very small density and pressure. The effectiveness and robustness of the scheme in computing the challenging problem of shock wave interaction with a cloud of bubbles tightly clustered and placed in a body of liquid is also demonstrated.

9:57AM G12.00010 Simulating shock-bubble interactions at water-gelatin interfaces¹, STEFAN ADAMI, JAKOB KAISER, Technical University of Munich, IVAN BERMEJO-MORENO, University of Southern California, NIKOLAUS ADAMS, Technical University of Munich — Biomedical problems are often driven by fluid dynamics, as in vivo organisms are usually composed of or filled with fluids that (strongly) affected their physics. Additionally, fluid dynamical effects can be used to enhance certain phenomena or destroy organisms. As examples, we highlight the benign potential of shockwave-driven kidney-stone lithotripsy or sonoporation (acoustic cavitation of microbubbles) to improve drug delivery into cells. During the CTR SummerProgram 2016 we have performed axisymmetric three-phase simulations of a shock hitting a gas bubble in water near a gelatin interface mimicking the fundamental process during sonoporation. We used our multi-resolution finite volume method with sharp interface representation (level-set), WENO-5 shock capturing and interface scale-separation and compared the results with a diffuse-interface method (Kobayashi et al., Phys. Med. Biol. 56(19), 2011). Qualitatively our simulation results agree well with the reference. Due to the interface treatment the pressure profiles are sharper in our simulations and bubble collapse dynamics are predicted at shorter time-scales. Validation with free-field collapse (Rayleigh collapse) shows very good agreement.

¹The project leading to this application has received funding from the European Research Council (ERC) under the European Unions Horizon 2020 research and innovation programme (grant agreement No 667483).

Monday, November 21, 2016 8:00AM - 10:10AM –
Session G13 DFD GPC: Geophysical Fluid Dynamics: Oceans I, Stratification, Waves & Tides C124 - Hussein Aluie, University of Rochester

8:00AM G13.00001 Restratification at oceanic fronts by baroclinic instabilities, VICKY VERMA, HIEU PHAM, VAMSI CHALAMALLA, SUTANU SARKAR, University of California, San Diego — Large eddy simulation with adaptive mesh refinement is used to investigate how stratification in the upper ocean surface layer evolves at frontal zones. The model includes a front with both lateral and vertical density gradients that is initially in geostrophic balance. The vertical density gradient consists of a mixed layer and a thermocline with constant stratification. Cases with different mixed layer depth are explored to contrast how the front equilibrates in different seasons. The evolution of the flow consists of the growth of baroclinic instability followed by nonlinear evolution into three-dimensional eddies that stir fluid across the front. These eddies create thin regions that have elevated shear, density gradient and turbulent mixing. The difference in the flow dynamics between the mixed layer and the thermocline is described using the organization of vortical structures and quantified through momentum and energy budgets.

8:13AM G13.00002 Energetics of baroclinic response to tidal forcing at steep topography

, MASOUD JALALI, Ph.D candidate, University of California, San Diego, SUTANU SARKAR, University of California, San Diego

— Topographic features with steep, supercritical slope are sites of large energy conversion from the oscillating barotropic tide to internal waves according to linear theory. However, large local energy loss is also reported in regions with steep supercritical (topographic slope larger than the slope of the wave propagation angle) topography, e.g. at Luzon strait. High-resolution, three-dimensional LES have been performed for a triangular obstacle and a more realistic obstacle taken as a scaled-down model of a Luzon Strait cross-section. These simulations resolve turbulence, compute a closed baroclinic energy budget and quantify the local baroclinic energy loss, q . The results are used to investigate the dependence of terms in the baroclinic energy budget on the tidal forcing amplitude, U_0 . Stronger barotropic forcing in the regime of low-to-moderate excursion number with $Ex \leq O(1)$ corresponding to broad, tall topography leads to stronger wave response and higher value of q . The rise in the energy loss to turbulence, P , is faster than U_0^2 , varying approximately as U_0^3 .

8:26AM G13.00003 Direct Measurements of the Baroclinic Instability in the Ocean¹

, MAHMOUD SADEK, HUSSEIN ALUIE, University of Rochester, MATTHEW HECHT, Los Alamos National Lab, GEOFFREY VALLIS, University of Exeter — The ocean is mechanically driven by wind and buoyancy at the surface which produce sloping isopycnals with a reservoir of available potential energy (APE). Large scale APE can be converted to kinetic energy via the baroclinic instability, which produces mesoscale eddies. Mesoscale eddies are ubiquitous in mid- and high-latitudes, and play a primary role in determining the strength and trajectories of currents and in generating intrinsic climate variability. The widespread belief that mesoscale eddies are generated through baroclinic instability is based on general accord between observations and linear stability analysis and the predicted behavior of nonlinear models. However, these models are unable to give us quantitative evidence of the extent to which the instability is responsible for eddy generation at various locations in the ocean. To this end, we implement a new coarse-graining framework, recently developed to study flow on a sphere, to directly analyze the baroclinic instability as a function of scale and geographic location, and implement it using strongly eddying high-resolution simulations in the North Atlantic and in the Southern Ocean. The results give us new information about location and intensity of the instability in both physical and spectral space.

¹Partial support was provided by National Science Foundation (NSF) Grant OCE-1259794, US Department of Energy (US DOE) Grant DE-SC0014318, and the LANL LDRD program through project number 20150568ER

8:39AM G13.00004 Optimal Transient Growth of Submesoscale Baroclinic Instabilities

, BRIAN WHITE, VARVARA ZEMSKOVA, PIERRE-YVES PASSAGGIA, UNC Chapel Hill — Submesoscale instabilities are analyzed using a transient growth approach to determine the optimal perturbation for a rotating Boussinesq fluid subject to baroclinic instabilities. We consider a base flow with uniform shear and stratification and consider the non-normal evolution over finite-time horizons of linear perturbations in an ageostrophic, non-hydrostatic regime. Stone (1966, 1971) showed that the stability of the base flow to normal modes depends on the Rossby and Richardson numbers, with instabilities ranging from geostrophic ($Ro \rightarrow 0$) and ageostrophic (finite Ro) baroclinic modes to symmetric ($Ri < 1$, $Ro > 1$) and Kelvin-Helmholtz ($Ri < 1/4$) modes. Non-normal transient growth, initiated by localized optimal wave packets, represents a faster mechanism for the growth of perturbations and may provide an energetic link between large-scale flows in geostrophic balance and dissipation scales via submesoscale instabilities. Here we consider two- and three-dimensional optimal perturbations by means of direct-adjoint iterations of the linearized Boussinesq Navier-Stokes equations to determine the form of the optimal perturbation, the optimal energy gain, and the characteristics of the most unstable perturbation.

8:52AM G13.00005 Numerical modeling of convective instabilities in internal solitary waves of depression shoaling over gentle slopes

, GUSTAVO RIVERA, PETER DIAMESSIS, Cornell University

— The shoaling of an internal solitary wave (ISW) of depression over gentle slopes is explored through fully nonlinear and non-hydrostatic simulations based on a high-accuracy deformed spectral multidomain penalty method. As recently observed in the South China Sea, in high-amplitude shoaling ISWs, the along-wave current can exceed the wave celerity resulting in convective instabilities. If the slope is less than 3%, the wave does not disintegrate as in the case of steeper slope shoaling but, instead, maintains its symmetric shape; the above convective instability may drive the formation of a turbulent recirculating core. The sensitivity of convective instabilities in an ISW is examined as a function of the bathymetric slope and wave steepness. ISWs are simulated propagating over both idealized and realistic bathymetry. Emphasis is placed on the structure of the above instabilities, the persistence of trapped cores and their potential for particle entrainment and transport. Additionally, the role of the baroclinic background current on the development of convective instabilities is explored. A preliminary understanding is obtained of the transition to turbulence within a high-amplitude ISW shoaling over progressively varying bathymetry.

9:05AM G13.00006 Effects of stratification on the equilibration of shallow fronts

, HIEU PHAM, SUTANU SARKAR, University of California, San Diego — Turbulence at frontal zones in the upper ocean is influenced by shear and density stratification that can vary in both lateral and vertical directions. The present study uses large eddy simulation to investigate the evolution of turbulence at a shallow front that consists of a geostrophic jet and stratification in both directions. The density gradient is represented by a hyperbolic-tangent profile in the lateral direction and a linear profile in the vertical direction. The vertical density gradient is varied among cases to explore the dynamics that evolve in different regimes of Richardson number, Rig . In the cases with $Rig < 0.25$, the turbulence rapidly develops throughout the surface layer and spreads laterally outward from the middle of the front. The front is quickly equilibrated within a fraction of an inertial period. In the cases with $0.25 < Rig < 1$, turbulence is initiated in a thin layer of elevated shear that forms near the surface. The turbulence that occurs in patches in the lateral direction spreads downward across the surface layer. The budgets of momentum, potential vorticity and energy are discussed to illustrate the different processes leading to the equilibration of the front.

9:18AM G13.00007 The response of an ocean front to small-scale turbulence

, MATTHEW CROWE, JOHN TAYLOR, DAMTP, University of Cambridge — Fronts, or regions with large horizontal density gradients, are common features of the upper ocean. Ocean fronts are hotspots for air/sea exchange and marine life. Observations indicate elevated levels of small scale turbulence at fronts, which nevertheless often have a stable density stratification. The dynamical processes that govern this stratification are not well understood. We consider the evolution of an initially balanced front to an imposed turbulent viscosity and diffusivity. Over long times the dominant balance is found to be the quasi-steady Turbulent Thermal Wind (TTW) balance with time-evolution due to an advection-diffusion balance in the buoyancy equation. We use the leading order balance to analytically determine similarity solutions for the spreading of a front and compare our results with numerical simulations.

9:31AM G13.00008 Instability properties under a model mode-1 internal tide. , JOHN

SEGRETO, PETER DIAMESSIS, Cornell University, Ithaca NY — The instability properties of the bottom boundary layer (BBL) under a model mode-1 internal tide in linearly stratified finite-depth water are studied, using 2-D direct numerical simulations (DNS) based on a spectral multidomain penalty method model. This model internal tide is a proxy for its lower-mode oceanic counterpart which is generated when stratified water is forced over topography by barotropic tidal currents. Such low-mode internal tidal waves tend to propagate long distances from the point of generation, carrying with them large amounts of energy. One mechanism through which this energy is dissipated is through wave-BBL interactions, where strong shear layers develop along the bed, leading to focused instabilities which are precursors for localized turbulent events. Such events in the BBL can cause sediment resuspension and drive benthic nutrient fluxes, playing a crucial role in ecosystem balances. In the model problem, the stability response of the time-dependent BBL is examined by introducing low-amplitude perturbations near the bed. The corresponding time-evolving BBL-integrated perturbation energy growth rates are then computed, by comparing both the perturbed and unperturbed cases. When an instability actually occurs, its vorticity structure and preferred location is identified. Ultimately, a stability boundary is constructed as a function of perturbation amplitude and internal wave steepness, aspect ratio and Reynolds number.

9:44AM G13.00009 Capturing remote mixing due to internal tides using multi-scale modeling tool: SOMAR-LES , EDWARD SANTILLI, Philadelphia University, VAMSI CHALAMALLA, ALBERTO SCOTTI,

Univ of NC - Chapel Hill, SUTANU SARKAR, University of California San Diego — Internal tides that are generated during the interaction of an oscillating barotropic tide with the bottom bathymetry dissipate only a fraction of their energy near the generation region. The rest is radiated away in the form of low- high-mode internal tides. These internal tides dissipate energy at remote locations when they interact with the upper ocean pycnocline, continental slope, and large scale eddies. Capturing the wide range of length and time scales involved during the life-cycle of internal tides is computationally very expensive. A recently developed multi-scale modeling tool called SOMAR-LES combines the adaptive grid refinement features of SOMAR with the turbulence modeling features of a Large Eddy Simulation (LES) to capture multi-scale processes at a reduced computational cost. Numerical simulations of internal tide generation at idealized bottom bathymetries are performed to demonstrate this multi-scale modeling technique. Although each of the remote mixing phenomena have been considered independently in previous studies, this work aims to capture remote mixing processes during the life cycle of an internal tide in more realistic settings, by allowing multi-level (coarse and fine) grids to co-exist and exchange information during the time stepping process.

9:57AM G13.00010 An experimental investigation of the Rossby two-slit problem¹ ,

ALEXIS KAMINSKI, DAMTP, University of Cambridge, JOSEPH PEDLOSKY, KARL HELFRICH, Woods Hole Oceanographic Institution — Rossby waves, which arise in response to buoyancy or winds at the sea surface, are a common feature of the oceans, and the problem of Rossby wave propagation in closed basins is a classical problem in geophysical fluid dynamics. Theoretical models of ocean circulation in basins with incomplete barriers such as ocean ridges or island chains (e.g. Pedlosky & Spall, JPO(29), 1999; Pedlosky, JPO(31), 2001) suggest that barriers extending through most of a basin are surprisingly inefficient at blocking the transmission of Rossby wave energy from one subbasin to the next. However, the existing theory neglects nonlinear effects and friction in the main basin interiors. To examine these effects, here we present the results of a series of experiments performed over a range of forcing frequencies and amplitudes, in which particle image velocimetry is used to measure the flow field. We find that while the linear theory appears to capture the large-scale structures of the flow, viscosity and nonlinearity significantly affect the flow along the boundaries and near the gaps in the barrier.

¹This project was funded by a 2014 WHOI GFD Fellowship.

Monday, November 21, 2016 8:00AM - 10:10AM –

Session G14 Free Surface Flows: Ships and Fluid-Solid Interactions C125-126 - Fotis Sotiropoulos, Stony Brook University

8:00AM G14.00001 The Impact of a Deepwater Wave on a Wall with Finite Vertical Extent¹ , AN WANG, JAMES H. DUNCAN, University of Maryland — The impact of a dispersively focused 2D plunging breaker (average wave frequency 1.15 Hz) on a 2D wall that is 45 cm high and 30 cm thick is studied experimentally. The temporal evolution of the water surface profile upstream of the wall is measured with a cinematic LIF technique using frame rates up to 4,500 Hz. Impact pressures on the wall are measured simultaneously at sample rates up to 900 kHz. The wall is located horizontally 6.41 m from the wave maker in all cases and the submergence of the bottom surface of the wall is varied. It is found that the impact behavior varies dramatically with the wall submergence. When the bottom is submerged by 13.3 cm, a flip-through impact occurs. In this case, the impact evolves without wave breaking and a vertical jet is formed. When the wall is submerged by less than 4.5 cm, small-amplitude components in the wave packet interact with the bottom of the wall before the main crest arrives. Ripples reflected during this interaction modify the behavior of the incoming breaker significantly. When the bottom of the wall is located sufficiently high above the mean water level, the first interaction occurs when the undisturbed wave crest collides with the wall. The highest pressures are observed in this case.

¹The support of the Office of Naval Research is gratefully acknowledged.

8:13AM G14.00002 Tsunami-induced force and surface pressure on multiple rectangular buildings in an unsteady free-surface channel flow¹ , ALIREZA BAHMANPOUR, IAN EAMES,

University College London — We study the flow around multiple rectangular obstacles in an unsteady free-surface channel flow using a combination of mathematical models, computations and experiments. The unsteady flow is triggered by a dam-break. The total drag force and surface pressure distribution on the obstacles are examined. The height and length of the building are fixed; the influence of initial water height and blocking ratio b/w is studied. The force scalings are confirmed from the computational analysis and found to be consistent with the experimental results. The effects of the additional buildings on the total drag force are noted and compared against the case of a single building. Increasing the number of buildings as well as the blocking ratio results in the water to inundate further onshore. The pressure distribution on the individual surfaces are analyzed and shown to vary linearly with height from the building base and dominated by the hydrostatic component. We summarize the results in terms of a new $Fr - b/w$ regime diagram and explain how the force on buildings subject to an unsteady flow can be estimated from the upstream velocity and water height.

¹We would like to thank HR Wallingford for their continued support in funding the project.

8:26AM G14.00003 Computational simulations of the interaction of water waves with pitching flap-type ocean wave energy converters¹, ASHISH PATHAK, MEHDI RAESSI, Univ of Mass - Dartmouth — Using an in-house computational framework, we have studied the interaction of water waves with pitching flap-type ocean wave energy converters (WECs). The computational framework solves the full 3D Navier-Stokes equations and captures important effects, including the fluid-solid interaction, the nonlinear and viscous effects. The results of the computational tool, is first compared against the experimental data on the response of a flap-type WEC in a wave tank, and excellent agreement is demonstrated. Further simulations at the model and prototype scales are presented to assess the validity of the Froude scaling. The simulations are used to address some important questions, such as the validity range of common WEC modeling approaches that rely heavily on the Froude scaling and the inviscid potential flow theory. Additionally, the simulations examine the role of the Keulegan-Carpenter (KC) number, which is often used as a measure of relative importance of viscous drag on bodies exposed to oscillating flows. The performance of the flap-type WECs is investigated at various KC numbers to establish the relationship between the viscous drag and KC number for such geometry. That is of significant importance because such relationship only exists for simple geometries, e.g., a cylinder.

¹Support from the National Science Foundation is gratefully acknowledged

8:39AM G14.00004 Computational modeling of pitching cylinder-type ocean wave energy converters using 3D MPI-parallel simulations¹, COLE FRENIERE, ASHISH PATHAK, MEHDI RAESSI, University of Massachusetts Dartmouth — Ocean Wave Energy Converters (WECs) are devices that convert energy from ocean waves into electricity. To aid in the design of WECs, an advanced computational framework has been developed which has advantages over conventional methods. The computational framework simulates the performance of WECs in a virtual wave tank by solving the full Navier-Stokes equations in 3D, capturing the fluid-structure interaction, nonlinear and viscous effects. In this work, we present simulations of the performance of pitching cylinder-type WECs and compare against experimental data. WECs are simulated at both model and full scales. The results are used to determine the role of the Keulegan-Carpenter (KC) number. The KC number is representative of viscous drag behavior on a bluff body in an oscillating flow, and is considered an important indicator of the dynamics of a WEC. Studying the effects of the KC number is important for determining the validity of the Froude scaling and the inviscid potential flow theory, which are heavily relied on in the conventional approaches to modeling WECs.

¹Support from the National Science Foundation is gratefully acknowledged.

8:52AM G14.00005 Fluid-structure interaction of complex bodies in two-phase flows on locally refined grids¹, DIONYSIOS ANGELIDIS, St. Anthony Falls Laboratory, University of Minnesota, LIAN SHEN, Department of Mechanical Engineering, University of Minnesota, FOTIS SOTIROPOULOS, Department of Civil Engineering, College of Engineering and Applied Sciences, Stony Brook University — Many real-life flow problems in engineering applications involve fluid-structure interaction (FSI) of arbitrarily complex geometries interacting with free surface flows. Despite the recent significant computational advances, conventional numerical methods are inefficient to resolve the prevailing complex dynamics due to the inherent large disparity of spatial and temporal scales that emerge in the air/water phases of the flow and around rigid bodies. To this end, the new generation 3D, unsteady, unstructured Cartesian incompressible flow solver, developed at the Saint Anthony Falls Laboratory (SAFL), is integrated with a FSI immersed boundary method and is coupled with the level-set formulation. The predictive capabilities of our method to simulate non-linear free surface phenomena, with low computational cost, are significantly improved by locally refining the computational grid in the vicinity of solid boundaries and around the free surface interface. We simulate three-dimensional complex flows involving complex rigid bodies interacting with a free surface both with prescribed body motion and coupled FSI and we investigate breaking wave events. In all the cases, very good agreement with benchmark data is found.

¹This material is based upon work supported by the National Science Foundation (CBET-1509071).

9:05AM G14.00006 Ship wave resistance on non-linear vertically sheared currents, BENJAMIN SMELTZER, YAN LI, SIMEN ELLINGSEN, Norwegian University of Science and Technology — Wave resistance is responsible for approximately one-third of the fuel consumption for large-size ships. We present calculations of this wave resistance force for a vessel traveling on a realistic background current of arbitrary depth-dependence. Previous theoretical work has considered currents that vary linearly with depth (constant shear), with results showing a dependence of the wave resistance forces on the shear strength as well as the direction of ship motion relative to the current profile. We extend these results to realistic measured profiles using a piecewise linear approximation to the current in the vertical direction. The background current is divided into layers each with a linear velocity profile. The method is applied to various measured current profiles, and wave resistance calculations are presented as a function of a number of system parameters such as ship Froude number, direction of motion relative to the surface current, and hull shape aspect ratio. For the profiles considered here, the wave resistance may vary up to a factor of two with direction of motion for low Froude numbers.

9:18AM G14.00007 Free-surface turbulent wake of a surface-piercing slender body at various Froude numbers¹, JEONGHWA SEO, Seoul Natl Univ, ABDUS SAMAD, Indian Institute of Technology Madras, SHIN HYUNG RHEE, Seoul Natl Univ — Free-surface effects on the near-wake around a surface-piercing slender body were investigated through flow field and wave elevation measurements. The near-wake flow field was measured by a towed underwater stereoscopic particle image velocimetry (SPIV) system. The measured flow field was analyzed to obtain coherent turbulence structures by using the Reynolds and proper orthogonal decomposition methods. Three different Froude numbers (Fr) - 0.126, 0.282, and 0.400 - were selected to represent mild, intermediate, and violent free-surface motions. At $Fr = 0.126$, the wave was hardly visible, although the turbulence strength and isotropy increased near the free-surface. At $Fr = 0.282$, though it was steady and smooth, wave-induced separation was clearly observed near the juncture of the free-surface and model trailing edge. At $Fr = 0.400$, wave breaking and the resulting bubbly free-surface were developed with an expanded wave-induced separation region. The wave-induced separation stimulated momentum transfer and turbulence dissipation, resulting in a significant change in the frequency of dominant free-surface motion in the downstream.

¹This research was supported by the IT RD program of MOTIE/KEIT (Grant No. 100660329) and the National Research Foundation of Korea, grant funded by the Korean government (Grant No. 2013R1A1A2012597)

9:31AM G14.00008 Modified Contact Line Dynamics about a Surface-Piercing Hydrofoil¹, MORGANE GRIVEL, DAVID JEON, MORTEZA GHARIB, California Institute of Technology — The contact line around a surface-piercing hydrofoil is modified by introducing alternating hydrophobic and hydrophilic bands along one side of the body. These bands are either aligned perpendicular or parallel to the flow direction. The other side of the hydrofoil is un-patterned and retains its original, uniformly hydrophilic properties. The hydrofoil is mounted onto air bearings, such that it can freely move side-to-side in the water tunnel. A force sensor is attached to the setup via a universal joint in order to measure the forces acting on the body for several Reynolds numbers (ranging from 10^4 to 10^5) and angles of attack (ranging from -10° to 10°). Cameras are also used to record the resulting flow structures and free surface elevation. The generation of wave trains and an altered free-surface elevation (also associated with the generation of surface waves) are observed over a wide range flow conditions. Force measurements elucidate how introducing these flow features impacts the forces acting on the hydrofoil, specifically with regards to the generation of lateral forces due to the asymmetric wetting conditions on either side of the hydrofoil.

¹Work is funded by ONR grant N00014-11-1-0031 and NSF GRFP grant DGE-1144469

9:44AM G14.00009 Hydrodynamic interaction between rigid surfaces planing on water., GHAZI BARI, KONSTANTIN MATVEEV, Washington State University — This study addresses hydrodynamic interaction of multi-surface planing hulls in the linearized, inviscid, steady flow approximation. A potential-flow-based hydrodynamic sources are distributed on the water surface to model water flow around three-dimensional hulls at finite Froude numbers. The pressure distribution on the hull surfaces are calculated as a part of the solution, and then the lift force and center of pressure are determined. For validation, numerical results are compared with an available analytical solution, experimental results, and empirical correlation equations. Parametric calculations are carried out for different hull designs in variable speed regimes, hull aspect ratios, deadrise angles and hull spacings. Results are presented for the lift coefficient, drag components, lift-drag ratio, center of pressure, and some illustrations are given for the water surface elevations. Obtained results can assist naval architects in improving design of high speed marine vehicles.

9:57AM G14.00010 Turbulent mass flux closure modeling for variable density turbulence in the wake of an air-entraining transom stern¹, KELLI HENDRICKSON, DICK YUE, Massachusetts Institute of Technology — This work presents the development and *a priori* testing of closure models for the incompressible highly-variable density turbulent (IHVDT) flow in the near wake region of a transom stern. This complex, three-dimensional flow includes three regions with distinctly different flow behavior: (i) the convergent corner waves that originate from the body and collide on the ship center plane; (ii) the "rooster tail" that forms from the collision; and (iii) the diverging wave train. The characteristics of these regions involve violent free-surface flows and breaking waves with significant turbulent mass flux (TMF) at Atwood number $At = (\rho_2 - \rho_1)/(\rho_2 + \rho_1) \approx 1$ for which there is little guidance in turbulence closure modeling for the momentum and scalar transport along the wake. Utilizing datasets from 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), we develop explicit algebraic turbulent mass flux closure models that incorporate the most relevant physical processes. Performance of these models in predicting the turbulent mass flux in all three regions of the wake will be presented.

¹Office of Naval Research

Monday, November 21, 2016 8:00AM - 10:10AM –
Session G15 Bio: Cardiovascular Flow I E143/144 - Yasser Aboelkassem, The Johns Hopkins University

8:00AM G15.00001 A Hybrid Windkessel Model of Blood Flow in Arterial Tree Using Velocity Profile Method, YASSER ABOELKASSEM¹, Johns Hopkins University, ZDRAVKO VIRAG², University of Zagreb — For the study of pulsatile blood flow in the arterial system, we derived a coupled Windkessel-Womersley mathematical model. Initially, a 6-elements Windkessel model is proposed to describe the hemodynamics transport in terms of constant resistance, inductance and capacitance. This model can be seen as a two compartment model, in which the compartments are connected by a rigid pipe, modeled by one inductor and resistor. The first viscoelastic compartment models proximal part of the aorta, the second elastic compartment represents the rest of the arterial tree and aorta can be seen as the connection pipe. Although the proposed 6-elements lumped model was able to accurately reconstruct the aortic pressure, it can't be used to predict the axial velocity distribution in the aorta and the wall shear stress and consequently, proper time varying pressure drop. We then modified this lumped model by replacing the connection pipe circuit elements with a vessel having a radius R and a length L. The pulsatile flow motions in the vessel are resolved instantaneously along with the Windkessel like model enable not only accurate prediction of the aortic pressure but also wall shear stress and frictional pressure drop. The proposed hybrid model has been validated using several in-vivo aortic pressure and flow rate data acquired from different species such as, humans, dogs and pigs. The method accurately predicts the time variation of wall shear stress and frictional pressure drop.

¹Institute for Computational Medicine, Dept. Biomedical Engineering

²Mechanical Engineering and Naval Architecture

8:13AM G15.00002 Finite element modeling of mass transport in high-Péclet cardiovascular flows¹, KIRK HANSEN, AMIRHOSSEIN ARZANI, SHAWN SHADDEN, Univ of California - Berkeley — Mass transport plays an important role in many important cardiovascular processes, including thrombus formation and atherosclerosis. These mass transport problems are characterized by Péclet numbers of up to 10^8 , leading to several numerical difficulties. The presence of thin near-wall concentration boundary layers requires very fine mesh resolution in these regions, while large concentration gradients within the flow cause numerical stabilization issues. In this work, we will discuss some guidelines for solving mass transport problems in cardiovascular flows using a stabilized Galerkin finite element method. First, we perform mesh convergence studies in a series of idealized and patient-specific geometries to determine the required near-wall mesh resolution for these types of problems, using both first- and second-order tetrahedral finite elements. Second, we investigate the use of several boundary condition types at outflow boundaries where backflow during some parts of the cardiac cycle can lead to convergence issues. Finally, we evaluate the effect of reducing Péclet number by increasing mass diffusivity as has been proposed by some researchers.

¹This work was supported by the NSF GRFP and NSF Career Award 1354541.

8:26AM G15.00003 Modelling Cerebral Blood Flow and Temperature Using a Vascular Porous Model, STEPHEN BLOWERS, Institute of Materials and Processes, University of Edinburgh, MICHAEL THRIPLLETON, IAN MARSHALL, Neuroimaging Sciences, Centre for Clinical Brain Sciences, University of Edinburgh, BRIDGET HARRIS, PETER ANDREWS, Critical Care Unit, NHS Lothian, Centre for Clinical Brain Sciences, University of Edinburgh, PRASHANT VALLURI, Institute of Materials and Processes, University of Edinburgh — Macro-modelling of cerebral blood flow can assist in determining the impact of temperature intervention to reduce permanent tissue damage during instances of brain trauma. Here we present a 3D two phase fluid-porous model for simulating blood flow through the capillary region linked to intersecting 1D arterial and venous vessel trees. This combined vasculature porous (VaPor) model simulates both flow and energy balances, including heat from metabolism, using a vasculature extracted from MRI data which are expanded upon using a tree generation algorithm. Validation of temperature balance has been achieved using rodent brain data. Direct flow validation is not as straight forward due to the method used in determining regional cerebral blood flow (rCBF). In-vivo measurements are achieved using a tracer, which disagree with direct measurements of simulated flow. However, by modelling a virtual tracer, rCBF values are obtained that agree with those found in literature. Temperature profiles generated with the VaPor model show a reduction in core brain temperature after cooling the scalp not seen previously in other models.

8:39AM G15.00004 Computational Modeling for Fluid-Porous Structure Interaction with Large Structural Deformation, RANA ZAKERZADEH, University of Texas at Austin, PAOLO ZUNINO, University of Pittsburgh — In this work, we utilize numerical models to investigate the importance of poroelasticity in the interaction of blood flow with a porohyperelastic vessel wall, and to establish a connection between the apparent viscoelastic behavior of the structure part and the intramural filtration flow. The main novelty is in the design of a Nitsche's splitting strategy, which separates the fluid from the structure sub-problems for the Fluid-Porous Structure Interaction system undergoing large deformations. The general idea is to use this model to study the influence of different parameters on energy dissipation in a poroelastic medium. We also study a new benchmark test specifically designed to investigate the effect of poroelasticity on large deformations.

8:52AM G15.00005 A 2D multiring model of blood flow in elastic arteries, ARTHUR GHIGO, PIERRE-YVES LAGRÉE, JOSE-MARIA FULLANA, Sorbonne Universités, CNRS and UPMC Université Paris 06, UMR 7190, Institut Jean Le Rond d'Alembert — Three-dimensional simulations of blood flow in elastic arteries are difficult and costly due to the complex fluid-structure interactions between the motion of the fluid and the displacement of the wall. We propose a two-dimensional multiring model to overcome those difficulties and obtain at a reasonable computational cost an asymptotically valid description of blood flow in large elastic arteries. The multiring equations are derived by integrating over concentric rings of fluid a simplified system of equations based on a long wave approximation of the axisymmetric Navier-Stokes equations and a thin-cylinder description of the arterial wall. Contrary to classical one-dimensional models, obtained by integrating the same system over a single ring, the multiring model computes the velocity profile as well as the wall shear stress and requires no a priori estimation of model coefficients. We show that by numerically solving the multiring system of equations, we are able to compute a large range of classical blood flow solutions, ranging from the elastic Womersley solution to the rigid tube Poiseuille solution.

9:05AM G15.00006 Secondary flow in a curved artery model with Newtonian and non-Newtonian blood-analog fluids¹, MOHAMMAD REZA NAJJARI, MICHAEL W PLESNIAK, George Washington University — Steady and pulsatile flows of Newtonian and non-Newtonian fluids through a 180°-curved pipe were investigated using particle image velocimetry (PIV). The experiment was inspired by physiological pulsatile flow through large curved arteries, with a carotid artery flow rate imposed. Sodium iodide (NaI) and sodium thiocyanate (NaSCN) were added to the working fluids to match the refractive index (RI) of the test section to eliminate optical distortion. Rheological measurements revealed that adding NaI or NaSCN changes the viscoelastic properties of non-Newtonian solutions and reduces their shear-thinning property. Measured centerline velocity profiles in the upstream straight pipe agreed well with an analytical solution. In the pulsatile case, secondary flow structures, i.e. deformed-Dean, Dean, Wall and Lyne vortices, were observed in various cross sections along the curved pipe. Vortical structures at each cross section were detected using the d_2 vortex identification method. Circulation analysis was performed on each vortex separately during the systolic deceleration phase, and showed that vortices split and rejoin. Secondary flow structures in steady flows were found to be morphologically similar to those in pulsatile flows for sufficiently high Dean number.

¹supported by the George Washington University Center for Biomimetics and Bioinspired Engineering

9:18AM G15.00007 High-order numerical simulations of pulsatile flow in a curved artery model¹, CHRISTOPHER COX, CHUNLEI LIANG, MICHAEL W PLESNIAK, George Washington University — Cardiovascular flows are pulsatile, incompressible and occur in complex geometries with compliant walls. Together, these factors can produce an environment that can affect the progression of cardiovascular disease by altering wall shear stresses. Unstructured high-order CFD methods are well suited for capturing unsteady vortex-dominated viscous flows, and these methods provide high accuracy for similar cost as low-order methods. We use an in-house three-dimensional flux reconstruction Navier-Stokes solver to simulate secondary flows and vortical structures within a rigid 180-degree curved artery model under pulsatile flow of a Newtonian blood-analog fluid. Our simulations use a physiological flowrate waveform taken from the carotid artery. We are particularly interested in the dynamics during the deceleration phase of the waveform, where we observe the deformed-Dean, Dean, Lyne and Wall vortices. Our numerical results reveal the complex nature of these vortices both in space and time and their effect on overall wall shear stress. Numerical results agree with and complement experimental results obtained in our laboratory using particle image velocimetry.

¹supported by the GW Center for Biomimetics and Bioinspired Engineering

9:31AM G15.00008 Color Doppler Ultrasound Velocimetry Flow Reconstruction using Vorticity-Streamfunction Formulation, BRETT MEYERS, PAVLOS VLACHOS, CRAIG GOERGEN, CARLO SCALO, Purdue University — Clinicians commonly utilize Color Doppler imaging to qualitatively assess the velocity in patient cardiac or arterial flows. However Color Doppler velocity are restricted to two-dimensional one-component measurements. Recently new methods have been proposed to reconstruct a two-component velocity field from such data. Vector Flow Mapping (VFM), in particular, utilizes the conservation of mass to reconstruct the flow. However, this method over-simplifies the influence of wall and surrounding blood motion on local measurements, which produce large, non-physical velocity gradients, requiring excessive smoothing operations to remove. We propose a new approach based on the Vorticity-Stream Function ($\psi\omega$) formulation that yields more physiologically accurate velocity gradients and avoids any added smoothing operations. Zero-penetration conditions are specified at the walls, removing the need for measurement of wall velocity from additional scans, which introduce further uncertainties in the reconstruction. Inflow and outflow boundary conditions are incorporated by prescribing Dirichlet boundary conditions. The proposed solver is compared against the VFM using computational data to evaluate measurement improvement. Finally we demonstrate the method by evaluating murine left ventricle Color Doppler scans.

9:44AM G15.00009 Flow Characterization of Severe Carotid Artery Stenosis in Pre- and Post-operative Phantoms by Using Magnetic Resonance Velocimetry¹, SEUNGBIN KO, SIMON SONG, Hanyang University, DOOSANG KIM, Seoul Veterans Hospital — It is remained unknown that the flow characteristics changes between pre- and post-operative severe carotid artery stenosis could affect the long-term patency or failure. However, in-vivo clinical experiments to uncover the flow details are far from bed-side due to limited measurement resolutions, blurring artifact, etc. We studied detailed flow characteristics of more than 75% severe carotid artery stenosis before and after surgical treatments. Real-size flow phantoms for 10 patients, who underwent carotid endarterectomy with patch/no patch closure, were prepared by using a 3D rapid-prototype machine from CT scanned images. The working fluid is a glycerin aqueous solution, and patient-specific pulsatile flows were applied to the phantoms, based on ultrasonic flow rate measurements. The flows were visualized with magnetic resonance velocimetry (MRV). The detailed flow characteristics are presented for both pre- and post-operative carotid arteries along with visualization data of 3 dimensional, 3 component velocity fields.

¹This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP) (No. 2016R1A2B3009541).

9:57AM G15.00010 ABSTRACT WITHDRAWN —

Monday, November 21, 2016 8:00AM - 10:10AM —

Session G16 Drops: Coalescence D133/134 - Hua Tan, Washington State University - Vancouver

8:00AM G16.00001 Drop coalescence at any Reynolds number, JAMES MUNRO, JOHN LISTER, University of Cambridge — When two drops touch, a fluid bridge forms between them and surface tension pulls this bridge wider, fighting against the viscosity of the fluid and the inertia of the drops. We present a new theoretical solution for the early-time behaviour of coalescence which includes both inertia and viscosity; earlier models neglect one or the other and so their predictions do not agree with experimental observations over the full range of fluid parameters. Our new solution is valid at early times for any Reynolds number and offers fresh insight into the physical processes governing this initial stage. Inertia plays a role on a scale proportional to $t^{1/2}$, but viscosity dominates on the smaller scales of the fluid bridge. The $t^{1/2}$ scale sets a boundary condition for the smaller scales, affecting the geometry of the curved surface and changing the prefactor in the rate of coalescence.

8:13AM G16.00002 Mist collection on parallel fiber arrays, ROMAIN LABB, CAMILLE DUPRAT, LadHyX, Ecole polytechnique — Fog is an important source of fresh water in specific arid regions such as the Atacama Desert in Chile. The method used to collect water passively from fog, either for domestic consumption or research purposes, consists in erecting large porous fiber nets on which the mist droplets impact. The two main mechanisms involved with this process are the impact of the drops on the fibers and the drainage of the fluid from the net, while the main limiting factor is the clogging of the mesh by accumulated water. We consider a novel collection system, made of an array of parallel fibers, that we study experimentally with a wind mist tunnel. In addition, we develop theoretical models considering the coupling of wind flow, droplet trajectories and wetting of the fibers. We find that the collection efficiency strongly depends on the size and distribution of the drops formed on the fibers, and thus on the fibers diameter, inclination angle and wetting properties. In particular, we show that the collection efficiency is greater when large drops are formed on the fibers. By adjusting the fibers diameter and the inter-fiber spacing, we look for an optimal structure that maximizes the collection surface and the drainage, while avoiding flow deviations.

8:26AM G16.00003 Surface flows and bulk mixing by coalescence of dissimilar drops: experiments and numerical simulations¹, MARK SIMMONS, EMILIA NOWAK, University of Birmingham, ZHIHUA XIE, CHRIS PAIN, OMAR MATAR, Imperial College London — Merging of dissimilar drops, being of different size and/or composition is an essential part of multiple promising applications enabling release and mixing of various species in bespoken way. However, till now there is still a lack of understanding of the effect of the various factor involved on the kinetics of coalescence and the rate of mixing of the contents of the drops. This study is aimed at providing a thorough understanding of the merging process immediately after the rupture of the thin liquid film separating the drops initially. The effect of such parameters as the difference in size and surface tension of the merging drops, as well as the viscosity of the surrounding liquid phase, is investigated. Numerical simulations provide a deeper insight into the liquid redistribution during the merging. Their results are in good agreement with the experimental data and will be discussed during the talk.

¹EPSRC UK Programme Grant MEMPHIS (EP/K003976/1)

8:39AM G16.00004 A multiphase ion-transport analysis of the electrostatic disjoining pressure: implications for binary droplet coalescence, LACHLAN MASON, Department of Chemical and Biomolecular Engineering, The University of Melbourne, FELIX GEBAUER, Chair of Separation Science and Technology, Technische Universitt Kaiserslautern, HANS-JRG BART, Chair of Separation Science and Technology, Technische Universitt Kaiserslautern, GEOFFREY STEVENS, DALTON HARVIE, Department of Chemical and Biomolecular Engineering, The University of Melbourne — Understanding the physics of emulsion coalescence is critical for the robust simulation of industrial solvent extraction processes, in which loaded organic and raffinate phases are separated via the coalescence of dispersed droplets. At the droplet scale, predictive collision-outcome models require an accurate description of the repulsive surface forces arising from electrical-double-layer interactions. The conventional disjoining-pressure treatment of double-layer forces, however, relies on assumptions which do not hold generally for deformable droplet collisions: namely, low interfacial curvature and negligible advection of ion species. This study investigates the validity bounds of the disjoining pressure approximation for low-inertia droplet interactions. A multiphase ion-transport model, based on a coupling of droplet-scale Nernst-Planck and Navier-Stokes equations, predicts ion-concentration fields that are consistent with the equilibrium Boltzmann distribution; indicating that the disjoining-pressure approach is valid for both static and dynamic interactions in low-Reynolds-number settings. The present findings support the development of coalescence kernels for application in macro-scale population balance modelling.

8:52AM G16.00005 Viscous Coalescence of Two Drops in a Saturated Vapor Phase ,

LINA BAROUDI, The City College of New York, SIDNEY R. NAGEL, University of Chicago, JEFFREY F. MORRIS, TAEHUN LEE, The City College of New York — When two liquid drops come into contact, a microscopic liquid bridge forms between them and rapidly expands until the two drops merge into a single bigger drop. Numerous studies have been devoted to the investigation of the coalescence singularity in the case where the drops coalesce in a medium of negligible vapor pressure such as vacuum or air. However, coalescence of liquid drops may also take place in a medium of relatively high vapor pressure (condensable vapor phase), where the effect of the surrounding vapor phase should not be neglected, such as the merging of drops in clouds. In this study, we carry out Lattice Boltzmann numerical simulations to investigate the dynamics of viscous coalescence in a saturated vapor phase. Attention is paid to the effect of the vapor phase on the formation and growth dynamics of the liquid bridge in the viscous regime. We observe that the onset of the coalescence occurs earlier and the expansion of the bridge initially proceeds faster when the coalescence takes place in a saturated vapor compared to the coalescence in a non-condensable gas. The initially faster evolution of the coalescence process in the saturated vapor is caused by the vapor transport through condensation during the early stages of the coalescence.

9:05AM G16.00006 Thermocapillary delay of drop coalescence , MICHELA GERI, GARETH

MCKINLEY, MIT- Department of Mechanical Engineering, JOHN BUSH, MIT - Department of Mathematics — We present the results of a combined experimental and theoretical investigation of drop coalescence. Particular attention is given to elucidating how the time to coalescence, or residence time, is affected by a temperature difference between drop and bath. Experiments show that the residence time increases as the temperature difference to the $2/3$ power. This simple scaling is rationalized through consideration of the thermal Marangoni flows induced.

9:18AM G16.00007 Coalescence of a Drop inside another Drop , VIVEK MUGUNDHAN, ZHEN

JIAN, FAN YANG, ERQIANG LI, SIGURDUR THORODDSEN, King Abdullah Univ of Sci & Tech (KAUST) — Coalescence dynamics of a pendent drop sitting inside another drop, has been studied experimentally and in numerical simulations. Using an in-house fabricated composite micro-nozzle, a smaller salt-water drop is introduced inside a larger oil drop which is pendent in a tank containing the same liquid as the inner drop. On touching the surface of outer drop, the inner drop coalesces with the surrounding liquid forming a vortex ring, which grows in time to form a mushroom-like structure. The initial dynamics at the first bridge opening up is quantified using Particle Image Velocimetry (PIV), while matching the refractive index of the two liquids. The phenomenon is also numerically simulated using the open-source code Gerris. The problem is fully governed by two non-dimensional parameters: the Ohnesorge number and the diameter ratios of the two drops. The validated numerical model is used to better understand the dynamics of the phenomenon. In some cases a coalescence cascade is observed with liquid draining intermittently and the inner drop reducing in size.

**9:31AM G16.00008 Adaptive-mesh-refinement simulation of partial coalescence cas-
cade of a droplet at a liquid-liquid interface , ABBAS FAKHARI, Univ of Notre Dame, DIOGO BOLSTER,**

University of Notre Dame — A three-dimensional (3D) adaptive mesh refinement (AMR) algorithm on structured Cartesian grids is developed, and supplemented by a mesoscopic multiphase-flow solver based on state-of-the-art lattice Boltzmann methods (LBM). Using this in-house AMR-LBM routine, we present fully 3D simulations of partial coalescence of a liquid drop with an initially flat interface at small Ohnesorge and Bond numbers. Qualitatively, our numerical simulations are in excellent agreement with experimental observations. Partial coalescence cascades are successfully observed at very small Ohnesorge numbers ($Oh \approx 10^{-4}$). The fact that the partial coalescence is absent in similar 2D simulations suggests that the Rayleigh-Plateau instability may be the principle driving mechanism responsible for this phenomenon.

**9:44AM G16.00009 Effect of Surfactants on Drop Coalescence at Liquid/liquid
Interfaces¹ , WEHELIYE HASHII WEHELIYE, TENG DONG, PANAGIOTA ANGELI², Univ Coll London —**

In this paper the coalescence of a drop with a liquid-liquid interface was investigated experimentally using Particle Image Velocimetry (PIV). Initially the drop rest on the interface was studied. It was found that during drop rest the interface deformed before rupture, and the deformation increased with increasing surfactant concentration. The results from PIV showed that two counter-rotating vortices formed inside the droplet during the rupture process which moved from the bottom to the top of the drop. The evolutions of vortices for three surfactant concentrations will be presented. The vortices moved faster in lower surfactant concentrations compared to the higher ones. The intensities of the vortices in different concentrations were also calculated. After the rupture, for low surfactant concentrations, the intensities increased with time and reached a maximum while at later times they decreased. At high surfactant concentrations, the increase and subsequent decrease in intensity was not as pronounced.

¹The work is support by the MEMPHIS program and the University College London. The author Teng Dong would like to thank the Chinese Scholarship Council (CSC) for providing funds for his overseas research.

²Corresponding author. Department of Chemical Engineering, University College London, Torrington Place, WC1E 7JE, UK

**9:57AM G16.00010 The Effect of a Yield Stress on the Drainage of the Thin Film
Between Two Colliding Newtonian Drops , SACHIN GOEL, ARUN RAMACHANDRAN, Chemical Engineering and**

Applied Chemistry, University of Toronto — Coalescence of drops immersed in fluids possessing a yield stress has been of interest to many industries such as the oil extraction, cosmetics and food industries. Unfortunately, a theoretical understanding of the drainage of the thin film of Bingham fluid (a model yield stress fluid) that develops between two drops undergoing a collision is still lacking, with the exception of two prior studies (Can. J. Chem. Eng., vol. 65, pp. 384-390, 1987, and J. Phys. Chem., vol. 90, pp. 6054-6059, 1986.) that make ad-hoc assumptions about the film shape. In this work, we examine this problem via a combination of scaling analysis and numerical simulations based on the lubrication analysis. There are four key features of the film drainage process of Bingham fluids. First, the introduction of a yield stress in the suspending fluid retards the drainage process relative to Newtonian fluid of the same viscosity. Second, the drainage time shows a minimum with respect to the capillary number. Third, the effect of yield stress on the drainage process becomes more pronounced at higher capillary numbers and lower Hamaker constant. Lastly, below a critical height, drainage can be arrested completely due to the yield stress. This critical height scales as $\tau_0^2 R^3 / \gamma^2$, where τ_0 is the yield stress, R is the drop radius and γ is the interfacial tension, and is, surprisingly, independent of the force colliding the drops. This and other distinguishing characteristics of the drainage process will be elucidated in the presentation.

Monday, November 21, 2016 8:00AM - 9:57AM –
Session G17 Reacting Flows: Flame Dynamics D131 - Yiguang Ju, Princeton University

8:00AM G17.00001 Large-activation-energy analysis of gaseous reactive flow in pipes

, DANIEL MORENO-BOZA, University of California San Diego, IMMACULADA IGLESIAS, Universidad Carlos III de Madrid, ANTONIO L SANCHEZ, University of California San Diego — Frank-Kamenetskii's analysis of thermal explosions is applied, using also a single-reaction model with an Arrhenius rate having a large activation energy, to describe the evolution of an initially cold gaseous mixture flowing along a circular pipe with constant wall temperature for moderately large values of the relevant Reynolds number. The analysis shows two modes of combustion. There is a flameless slowly reacting mode for low wall temperatures or small pipe radii, when the temperature rise resulting from the heat released by the reaction is kept small by the heat-conduction losses to the wall, so as not to change significantly the order of magnitude of the reaction rate. In the other mode, the slow reaction rates occur only in an initial ignition stage, which ends abruptly when very large reaction rates cause a temperature runaway, or thermal explosion, at a well-defined ignition distance. The analysis determines the slow streamwise evolution for the flameless mode of combustion as well as the ignition distance for the explosive mode.

8:13AM G17.00002 Propagation Limits of High Pressure Cool Flames , YIGUANG JU,

Princeton University — The flame speeds and propagation limits of premixed cool flames at elevated pressures with radiative heat loss are numerically modelled using dimethyl ether mixtures. The primary focus is paid on the effects of pressure, mixture dilution, flame size, and heat loss on cool flame propagation. The results showed that cool flames exist on both fuel lean and fuel rich sides and thus dramatically extend the lean and rich flammability limits. There exist three different flame regimes, hot flame, cool flame, and double flame. A new flame flammability diagram including both cool flames and hot flames is obtained at elevated pressure. The results show that pressure significantly changes cool flame propagation. It is found that the increases of pressure affects the propagation speeds of lean and rich cool flames differently due to the negative temperature coefficient effect. On the lean side, the increase of pressure accelerates the cool flame chemistry and shifts the transition limit of cool flame to hot flame to lower equivalence ratio. At lower pressure, there is an extinction transition from hot flame to cool flame. However, there exists a critical pressure above which the cool flame to hot flame transition limit merges with the lean flammability limit of the hot flame, resulting in a direct transition from hot flame to cool flame. On the other hand, the increase of dilution reduces the heat release of hot flame and promotes cool flame formation. Moreover, it is shown that a smaller flame size and a higher heat loss also extend the cool flame transition limit and promote cool flame formation.

8:26AM G17.00003 Flame speeds and curvature of premixed, spherically expanding flames advecting in a turbulent channel flow¹ , DAN FRIES, BRADLEY OCHS, DEVESH RANJAN, SURESH MENON, Georgia Institute of Technology — A new facility has been developed at the Georgia Institute of Technology to study sub- and supersonic combustion, which is based on classical flame bomb studies but incorporates a mean flow, allowing for a wider variety of turbulent conditions and the inclusion of effects like compressibility, while supporting shear-free spherical flames. Homogeneous, isotropic turbulence is generated via an active vane grid. Methane-air flame kernels advecting with the mean flow are generated using Laser Induced Breakdown ignition. The facility is accessing the thin reaction zone regime with $u'_{RMS}/S_L^0 = 6.9 - 22$, $L_{11}/\delta_F = 44 - 68$ and $Re_\lambda = 190 - 550$. The flame kernels are probed with OH-Planar Laser Induced Fluorescence (PLIF). To validate the facility, results at $\bar{U} = 30$ m/s are compared to existing data using a scaling derived from a spectral closure of the G-equation. This indicates the reacting flow remains Galilean invariant under the given conditions. The differences between global and local turbulent consumption speeds derived from OH-PLIF results are discussed with a focus on modeling efforts. The curvature of flame wrinkles is evaluated to examine the impact of different turbulent scales on flame development.

¹This work was supported by the Air Force Office of Scientific Research under basic research grant FA9550-15-1-0512 (Project monitor: Dr. Chiping Li)

8:39AM G17.00004 Non linear dynamics of flame cusps: from experiments to modeling , CHRISTOPHE ALMARCHA¹, BASILE RADISSON, ELIAS AL-SARRAF, JOEL QUINARD, EMMANUEL VILLERMAUX, BRUNO DENET, Aix-Marseille Univ., IRPHE, UMR 7342 CNRS, Centrale Marseille, Technopole de Chtea Gombert, 49 rue F. Joliot Curie, 13384 Marseille Cedex 13, France, GUY JOULIN, Institut P-prime, UPR 3346 CNRS, ENSMA, Universit de Poitiers, 1 rue clment Ader, B.P. 40109, 86961 Futuroscope Cedex, Poitiers, France — The propagation of premixed flames in a medium initially at rest exhibits the appearance and competition of elementary local singularities called cusps. We investigate this problem both experimentally and numerically. An analytical solution of the two-dimensional Michelson Sivashinsky equation is obtained as a composition of pole solutions, which is compared with experimental flames fronts propagating between glass plates separated by a thin gap width. We demonstrate that the front dynamics can be reproduced numerically with a good accuracy, from the linear stages of destabilization to its late time evolution, using this model-equation. In particular, the model accounts for the experimentally observed steady distribution of distances between cusps, which is well-described by a one-parameter Gamma distribution, reflecting the aggregation type of interaction between the cusps. A modification of the Michelson Sivashinsky equation taking into account gravity allows to reproduce some other special features of these fronts.

¹Aix-Marseille Univ., IRPHE, UMR 7342 CNRS, Centrale Marseille, Technopole de Chtea Gombert, 49 rue F. Joliot Curie, 13384 Marseille Cedex 13, France

8:52AM G17.00005 Role of Molecular Diffusion in Turbulent Flames: Two Examples

, ABHISHEK SAHA, SHENG YANG, CHUNG K LAW, Princeton University — In modeling turbulent flames, especially for high turbulent Reynolds number, it is often believed that diffusion is primarily controlled by turbulent diffusivity; and as such the effects of molecular diffusion can be ignored by artificially assuming unity Lewis number (Le) defined as the ratio of thermal diffusivity to mass diffusivity. Based on our recent experiments with expanding turbulent flames, we will present two examples where Le significantly alters the flame dynamics even in strong turbulent environments. In the first example, we show that ignition of a combustible mixture by a high-energy kernel can be facilitated by turbulence for $Le > 1$ mixture while it is commonly believed to be more difficult in turbulence due to the increased dissipation rate of the deposited energy. In the second example, we show that nominally nonflammable mixtures with low adiabatic flame temperatures can burn strongly in turbulence for $Le < 1$ mixtures. In both cases, turbulence morphs the positively stretched spherical flame into a multitude of wrinkled flamelets subjected to both positive and negative stretches. Mechanistically, these effects are consequences of the coupling between differential molecular diffusion and these positively or negatively stretched flamelets.

9:05AM G17.00006 Representation of the Essential Flame-Turbulence Dynamics using Specific Flame-Vortex Interactions, PAULO L. K. PAES, The Pennsylvania State University, JAMES BRASSEUR, University of Colorado, Boulder, YUAN XUAN, The Pennsylvania State University — Many engineering applications involve turbulent reacting flows, where nonlinear, multi-scale turbulence-combustion couplings are important. Directly resolving the complex fluid dynamics involved in these applications is associated with prohibitive computational costs, which makes it necessary to employ turbulent closure models and turbulent combustion models to account for the effects of unresolved scales on resolved scales. Most of these existent closure models rely on some assumptions about the turbulence dynamics and the scale separation between turbulence and the different combustion processes. 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. Instead of developing an extreme-resolution, high Reynolds number turbulent flame simulation that is limited to a localized part of the regime diagram, in this work, we propose to develop a series of numerical experiments of simplified interactions between a laminar premixed flame and specified vortex distributions of varying strengths and scales to capture the essential flame-turbulence dynamics over distinct premixed turbulent combustion regimes. The response of the laminar flame to different vortex time and length scales is investigated and the physical relevance of each dataset to practical turbulent premixed flames is discussed.

9:18AM G17.00007 ABSTRACT WITHDRAWN –

9:31AM G17.00008 Turbulent/Non-Turbulent Interface in a Reacting Compressible Shear Layer, REZA JAHANBAKHSHI, CYRUS K. MADNIA, SUNY at Buffalo — Since entrainment occurs across the turbulent/non-turbulent interface (TNTI), DNS data is used to study the characteristics of this interface in shear layers. Several cases are considered ranging from a low compressible non-reacting to highly compressible reacting flows. As the compressibility level increases, the average size of the structures that form the TNTI increases, however, as the heat release level increases, the average size of the structures that form the local shape of TNTI decreases. The geometrical shape of the turbulent/non-turbulent interface looking from the turbulent region is examined. It is observed that in non-reacting cases the TNTI is dominated by the concave shaped surfaces. As the level of compressibility increases, the probability of finding highly curved concave shaped surfaces on the TNTI decreases, while the probability of finding flatter concave and convex shaped surfaces increases. In reacting flows with high heat release level, the TNTI is dominated by the convex shaped surfaces. As the heat release level increases the probability of finding highly curved convex shaped surfaces on the TNTI increases, whereas the probability of finding flatter concave and convex shaped surfaces decreases.

9:44AM G17.00009 Entrainment in a Reacting Compressible Shear Layer, CYRUS K. MADNIA, REZA JAHANBAKHSHI, SUNY at Buffalo — DNS of reacting turbulent shear layer is performed to study the entrainment of the irrotational flow into the turbulent region. The effects of heat release and compressibility on the flow are examined. Infinitely fast chemistry approximation is used to model the one-step global reaction of hydrogen in air. Entrainment is studied via two mechanisms; nibbling, considered as the vorticity diffusion across the turbulent/non-turbulent interface, and engulfment, the drawing of the pockets of the outside irrotational fluid into the turbulent region. As the level of compressibility or heat release increases, the total entrained mass flow rate into the shear layer decreases. It is observed that nibbling is a viscous dominated mechanism in non-reacting cases, whereas it is essentially inviscid in reacting flows with high heat release level. It is shown that the contribution of the engulfment to entrainment is small for the non-reacting flows, while mass flow rate due to engulfment can constitute up to forty percent of total entrainment in reacting cases. This increase is primarily related to a decrease of mass flow rate due to nibbling while the mass flow rate due to engulfment does not change significantly in reacting cases.

Monday, November 21, 2016 8:00AM - 10:10AM – Session G18 Flow Instability: General D135 - Ryan Keedy, Sandia National Laboratories

8:00AM G18.00001 Numerical and experimental investigation of flow instabilities in the presence of a viscosity gradient¹, RYAN KEEDY, Sandia National Laboratories, ALBERTO ALISEDA, University of Washington — Laboratory experiments were performed to understand the effect of viscosity ratio on the development of the round jet when a miscible liquid is injected into another stagnant ambient liquid. Altering the viscosity of the injected liquid jet resulted in noticeable changes in the turbulent/non-turbulent interface in the jet's developing region, including the instability wavelength. The change in the formation of structures at the interface is apparent even when several key non-dimensional numbers (Pe , Re) associated with the flow are kept constant. Large, coherent structures in the turbulent jet resulting from the shear instability of the interface may affect the downstream development of the self-similar profile. Hence, it is important to examine and understand the characteristics of the shear layer instability in order to better understand the role that a viscosity gradient plays in turbulent jet development. The spatial stability equations for a flow in which viscosity varies arbitrarily as a function of scalar concentration are presented. These equations are evaluated at various viscosity ratios and the predicted instability frequencies are compared to experimental results in the range of $\mu_{jet}/\mu_{amb} = 0.5 - 2$ and $Re \approx 10^4$.

¹Sandia 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 NNSA under contract DE-AC04-94AL85000.

8:13AM G18.00002 On long-time algebraic and exponential instabilities found in linear dispersive flows, NATHANIEL BARLOW, KRISTINA KING, PAULA ZARETZKY, MICHAEL CROMER, STEVEN WEINSTEIN, Rochester Institute of Technology — A physically-motivated class of partial differential equations that describes the response of a system to disturbances is examined. Morphological differences are identified between system responses that exhibit algebraic growth and the more typical case of exponential growth. Specifically, the propagation characteristics of the response are examined in the context of spatio-temporal hydrodynamic stability theory. One key attribute of predicted algebraically growing solutions is the prevalence of transient growth in almost all of the response, with the long-time growth occurring asymptotically at precisely one wave speed.

8:26AM G18.00003 Morphological instabilities during the rapid solidification of three component systems, ANTHONY ALTIERI, STEPHEN DAVIS, PETER VOORHEES, Northwestern University — Rapidly solidifying binary mixtures of a major component and a dilute solute are known to be subject to morphological instabilities. The stability of ternary mixtures is not well understood. A linear stability analysis of ternary mixtures of one major component and two dilute solutes is performed. The growing cellular and oscillatory instabilities, present in binary systems, are investigated for ternary systems. The effect of thermal and concentration gradients, surface energy, interface kinetics, and nonequilibrium thermodynamics on the morphological instabilities, and the use of the second solute to stabilize the system, are considered.

8:39AM G18.00004 Effect of Discrete Roughness on Transition on a Sharp Cone at an Angle of Attack at Mach 6¹, ERIC MATLIS², THOMAS CORKE³, University of Notre Dame, MICHAEL SEMPER, U.S. Air Force Academy — Experiments were performed to investigate passive discrete patterned roughness for transition control on a sharp right-circular cone at a 6° angle of attack at Mach 6.0. The angle of attack was set to produce a mean cross-flow velocity component in the boundary layer over the cone in which the cross-flow instability is the dominant mechanism of turbulent transition. The focus is transition control which is based on exciting less-amplified stationary cross-flow modes that suppress the growth of the more-amplified cross-flow modes, and thereby delay transition. The passive roughness consisted of an azimuthal array of micron-size indentations (dimples) at an axial location that was just upstream of the first linear stability neutral growth branch for cross-flow modes. Both critical and sub-critical azimuthal mode numbers of roughness were examined. The receptivity of the stationary cross-flow modes to the roughness was evaluated using Silicone-oil surface flow visualization. The visualization images were post-processed using a pixel-intensity based spectral analysis. Of particular interest was the effect that higher (conventional) tunnel acoustic levels had on the roughness receptivity.

¹Supported by AFOSR

²Member

³Fellow

8:52AM G18.00005 Subcritical Hopf bifurcations in low-density jets, YUANHANG ZHU, VIKRANT GUPTA, LARRY K. B. LI, The Hong Kong University of Science and Technology — Low-density jets are known to bifurcate from a steady state (a fixed point) to self-excited oscillations (a periodic limit cycle) when the Reynolds number increases above a critical value corresponding to the Hopf point, Re_H . In the literature, this Hopf bifurcation is often considered to be supercritical because the self-excited oscillations appear only when $Re > Re_H$. However, we find that under some conditions, there exists a hysteretic bistable region at $Re_{SN} < Re < Re_H$, where Re_{SN} denotes a saddle-node bifurcation point. This shows that the Hopf bifurcation can also be subcritical, which has three main implications. First, low-density jets could be triggered into self-excited oscillations even when $Re < Re_H$. Second, in the modeling of low-density jets, the subcritical or supercritical nature of the Hopf bifurcation should be taken into account because the former is caused by cubic nonlinearity whereas the latter is caused by square nonlinearity. Third, the response of the system to external forcing and noise depends on its proximity to the bistable region. Therefore, when investigating the forced response of low-density jets, it is important to consider whether the Hopf bifurcation is subcritical or supercritical.

9:05AM G18.00006 Pattern-Formation in Moist Turbulent Rayleigh-Benard Convection, PRASANTH PRABHAKARAN, STEPHAN WEISS, ALEXEI KREKHOV, HOLGER NOBACH, EBERHARD BODENSCHATZ, Max Planck Institute for Dynamics and Self Organization — We report experiments on droplet-condensation patterns in turbulent Rayleigh-Benard convection, where a horizontal fluid layer is heated from below and cooled from above. We use compressed Sulphur Hexafluoride (SF6) as the working fluid for pressures and temperatures in the liquid/vapor coexistence region. The vapor evaporating from the liquid pool above the heated bottom-plate undergoes film condensation on the cooled top-plate. We observe a finite wavelength instability of the condensed liquid film, which is in stark contrast to the well-known long-wavelength Rayleigh Taylor instability. In the non-linear stationary state, droplets periodically fall into the liquid pool below. Under appropriate conditions, we observe hexagonal patterns with a well-defined wavelength. By varying the pressure and temperature, and with it the evaporation/condensation rates we investigate the influence of these parameters on the observed patterns.

9:18AM G18.00007 Hydrodynamic instabilities and concentration polarization coupled by osmotic pressure in a Taylor-Couette cell, DENIS MARTINAND, Aix-Marseille Universit, NILS TILTON, Colorado School of Mines — This study addresses analytically and numerically the coupling between hydrodynamic instabilities and osmotic pressure driven by concentration polarization. The configuration consists of a Taylor-Couette cell filled with a Newtonian fluid carrying a passive scalar. Whereas the concentric inner and outer cylinders are membranes permeable to the solvent, they totally reject the scalar. As a radial in- or outflow of solvent is imposed through both cylinders, a concentration boundary layer develops on the cylinder where the solvent exits, until an equilibrium steady state is reached. In addition, the rotation of the inner cylinder is used to drive centrifugal instabilities in the form of toroidal vortices, which interact with the concentration boundary layer. By means of the osmotic pressure, concentration polarization is found to promote or hinder the hydrodynamic instabilities, depending on capacity of the vortices and diffusion to increase the concentration field at the membrane. The results obtained by analytical stability analysis agree with dedicated Direct Numerical Simulations.

9:31AM G18.00008 Dynamics of a single flexible filament in a flowing soap film, CHAONAN CHEN, SHUNSHAN FENG, TONG ZHOU¹, School of Mechatronical Engineering, Beijing Institute of Technology — The interactions between flexible plates and surrounding fluids like two-dimensional flag-in-wind problems are important physical phenomena. Here we use a spandex filament with one end fixed flapping in gravity-driven soap film device which can be regarded as a quasi-two-dimensional flow tunnel. A silk filament had been used previously to demonstrate three stable dynamical states: stretched-straight, flapping, and bistable states. The similar phenomena occurred for a spandex filament while the bifurcation conditions seem to be different compared with a silk filament, as the critical filament length is longer and critical inflow velocity is higher than that for a silk filament. In the experiment, we considered some representative parameters (filament length, inflow velocity, and bending stiffness of the filament) to study their effects on the stability of the filament and its bifurcation conditions. An interface-tracking ALE finite element method was then conducted to reproduce the experiment and investigate more details about effects of these parameters. which are significant to reveal the underlying mechanism of flag-in-wind problem.

¹Corresponding Author. Email:zhoutong@bit.edu.cn

9:44AM G18.00009 Mode competition and destabilization of microfluidic channel flows by the Coriolis force, SAUNAK SENGUPTA, SANDEEP SAHA, SUMAN CHAKRABORTY, IIT Kharagpur — Understanding flow stability in inertial microfluidics is very important due to its increased application in medical and chemical engineering. On a steady rotating platform centrifugal actuation drives fluid flow but Coriolis force can destabilize the flow and enhance mixing in a short span. We investigate the role of Coriolis force in micro-mixing and the structure of the roll-cells formed in rotating channel flows using linear stability theory. We conduct a parametric study at different rotation numbers, Reynolds number, axial and spanwise wavenumbers. Our results reveal existence of multiple competing unstable modes (Types I to IV) due to Coriolis force: Types I and II have been reported in literature and are responsible for the formation of evenly-spaced roll-cells. We find new instabilities (Types III and IV) which contribute to the formation of twisted roll cells. The existence of the instabilities is clearly demarcated on a regime map to assist future experiments to identify them. The kinetic energy budget has been analyzed to gain insight into the mechanism of energy transfer by Coriolis force from the mean flow to the perturbations. We make a qualitative comparison of roll-cells predicted by linear stability with previously reported experiments.

9:57AM G18.00010 Flow Simulations of The Dynamics of a Perturbed Solid-Body

Rotation Flow, SHIXIAO WANG, Auckland University, CHUNJUAN FENG, Northwestern Polytechnical University, FENG LIU, University of California, Irvine, ZVI RUSAK, Rensselaer Polytechnic Institute — DNS is conducted to study the 3-D flow dynamics of a base solid-body rotation flow with a uniform axial velocity in a finite-length pipe. The simulation results describe the neutral stability line in response to either axisymmetric or 3-dimensional perturbations in a diagram of Reynolds number (Re , based on inlet axial velocity and pipe radius) versus the incoming flow swirl ratio (ω). This line is in good agreement with the neutral stability line recently predicted by the linear stability theory of Wang *et al.* (2016). The Wang & Rusak (1996) axisymmetric instability mechanism and evolution to an axisymmetric breakdown state is recovered in the simulations at certain operational conditions in terms of Re and ω . However, at other operational conditions there exists a dominant, 3-dimensional spiral type of instability mode that agrees with the linear stability theory of Wang *et al.* (2016). The growth of this mode leads to a spiral type of flow roll-up that subsequently nonlinearly saturates on a rotating spiral type of vortex breakdown. The computed time history of the velocity components at a certain point in the flow is used to describe 3-dimensional phase portraits of the flow global dynamics and its long-term behavior.

Monday, November 21, 2016 8:00AM - 10:10AM –

Session G19 Bio: Flapping and Swimming I D136 - Tyler Van Buren, Princeton University

8:00AM G19.00001 Effect of trailing edge shape on the wake and propulsive performance of pitching panels¹, TYLER VAN BUREN, DANIEL FLORYAN, Princeton University, DANIEL BRUNNER, Zurich University, UTKU SENTURK, Ege University, ALEXANDER SMITS, Princeton University and Monash University — We present the effects of the trailing edge shape on the wake and propulsive performance of a pitching panel with an aspect ratio of 1. The trailing edges are symmetric chevron shapes with convex and concave orientations of varying degree. Concave trailing edges delay the natural vortex bending and compression of the wake, and the streamwise velocity field contains a single jet-like structure. Conversely, convex trailing edges promote wake compression and produce a wake split into four jets. Deviation from the square trailing edge mostly reduces the thrust and efficiency.

¹Supported by the Office of Naval Research under MURI grant number N00014-14-1-0533

8:13AM G19.00002 Experimental study of surface pattern effects on the propulsive performance and wake of a bio-inspired pitching panel¹, JUSTIN KING, RAJEEV KUMAR, MELISSA GREEN, Syracuse University — Force measurements and stereoscopic particle image velocimetry (PIV) were used to characterize the propulsive performance and wake structure of rigid, bio-inspired trapezoidal pitching panels. In the literature, it has been demonstrated that quantities such as thrust coefficient and propulsive efficiency are affected by changes in the surface characteristics of a pitching panel or foil. More specifically, the variation of surface pattern produces significant changes in wake structure and dynamics, especially in the distribution of vorticity in the wake. Force measurements and PIV data were collected for multiple surface patterns chosen to mimic fish surface morphology over a Strouhal number range of 0.17 to 0.56. Performance quantities are compared with the three-dimensional vortex wake structure for both the patterned and smooth panels to determine the nature and magnitude of surface pattern effects in terms of thrust produced, drag reduced, and wake vortices reshaped and reorganized.

¹This work was supported by the Office of Naval Research under ONR Award No. N00014-14-1-0418

8:26AM G19.00003 Momentum distribution in the wake of a bio-inspired trapezoidal pitching panel, RAJEEV KUMAR¹, JUSTIN KING², MELISSA GREEN³, Syracuse Univ — A trapezoidal pitching panel that models a fish caudal fin was used to study the distribution of streamwise momentum in its wake. The three-dimensional phase-averaged velocity fields were captured using stereoscopic PIV at Strouhal numbers (St) ranging from 0.17 to 0.56. The pitching trapezoidal panel wake consists of chains of interacting vortex rings that induce significant three-dimensional flows. With increasing Strouhal number, this wake structure induces flow with increasing non-dimensional downstream momentum, which is consistent with greater non-dimensional thrust production at higher St shown previously in the literature. Also at higher St , these chains of vortex rings split and diverge in the transverse direction, giving rise to a pair of downstream jets. At the highest St , a region of downstream momentum lower than the freestream is observed along the centerline between the jet pair. This loss of momentum surplus may be related to a previously described decline in propulsive efficiency at higher St . The momentum distribution is also studied in the time-averaged velocity fields to show how the average momentum is distributed over the same range of St .

¹Research Assistant Professor

²PhD Student

³Assistant Professor, MAE Department

8:39AM G19.00004 A computational investigation of heaving flexible panels in a fluid, ALEXANDER HOOVER, RICARDO CORTEZ, LISA FAUCI, Tulane University, ERIC TYTELL, Tufts University — We present a 3-dimensional computational model of a flexible panel with heave oscillations at the leading edge. Our approach uses direct numerical simulations of the fully coupled fluid-structure interaction system within an immersed boundary framework. The effective flexibility of the panel is varied over a range of heaving frequencies and bending rigidities, with the resulting force measurements recorded. We find good agreement with recent experimental results, confirming that resonant peaks of the trailing edge amplitude correspond to localized boosts in thrust. We then use the model to explore the relationship between the thrust recorded from a tethered, heaving panel and the forward swimming speed of an untethered, heaving panel. The deflections of the panels are further examined with beam mode analysis from the Euler-Bernoulli beam equation. Spanwise variations of the panel dimensions are also considered.

8:52AM G19.00005 ABSTRACT WITHDRAWN –

9:05AM G19.00006 Adjoint-based optimization of fish swimming gaits¹ , DANIEL FLORYAN,

CLARENCE W. ROWLEY, Princeton University, ALEXANDER J. SMITS, Princeton University and Monash University — We study a simplified model of fish swimming, namely a flat plate periodically pitching about its leading edge. Using gradient-based optimization, we seek periodic gaits that are optimal in regards to a particular objective (e.g. maximal thrust). The two-dimensional immersed boundary projection method is used to investigate the flow states, and its adjoint formulation is used to efficiently calculate the gradient of the objective function needed for optimization. The adjoint method also provides sensitivity information, which may be used to elucidate the physics responsible for optimality.

¹Supported under ONR MURI Grants N00014-14-1-0533, Program Manager Bob Brizzolara

9:18AM G19.00007 Nonstandard Gaits in Unsteady Hydrodynamics¹ , MICHAEL FAIRCHILD,

CLARENCE ROWLEY, Princeton University — Marine biology has long inspired the design and engineering of underwater vehicles. The literature examining the kinematics and dynamics of fishes, ranging from undulatory anguilliform swimmers to oscillatory ostraciiform ones, is vast. Past numerical studies of these organisms have principally focused on gaits characterized by sinusoidal pitching and heaving motions. It is conceivable that more sophisticated gaits could perform better in some respects, for example as measured by thrust generation or by cost of transport. This work uses an unsteady boundary-element method to numerically investigate the hydrodynamics and propulsive efficiency of high-Reynolds-number swimmers whose gaits are encoded by Fourier series or by Jacobi elliptic functions. Numerical results are presented with an emphasis on identifying particular wake structures and modes of motion that are associated with optimal swimming.

¹This work was supported by the Office of Naval Research through MURI grant N00014-14-1-0533.

9:31AM G19.00008 Phototactic guidance of a tissue-engineered soft-robotic ray. ,

SUNG-JIN PARK, Harvard University, MATTIA GAZZOLA, University of Illinois at Urbana-Champaign, KYUNG SOO PARK, University of Michigan, SHIRLEY PARK, Stanford University, VALENTINA DI SANTO, Harvard University, KARL DEISSEROTH, Stanford University, GEORGE V. LAUDER, L. MAHADEVAN, KEVIN KIT PARKER, Harvard University — Inspired by the relatively simple morphological blueprint provided by batoid fish such as stingrays and skates, we created a biohybrid system that enables an artificial animal, a tissue-engineered ray – to swim and phototactically follow a light cue. By patterning dissociated rat cardiomyocytes on an elastomeric body enclosing a microfabricated gold skeleton, we replicated fish morphology at 1/10 scale and captured basic fin deflection patterns of batoid fish. Optogenetics allows for phototactic guidance, steering, and turning maneuvers. Optical stimulation induced sequential muscle activation via serpentine-patterned muscle circuits, leading to coordinated undulatory swimming. The speed and direction of the ray was controlled by modulating light frequency and by independently eliciting right and left fins, allowing the biohybrid machine to maneuver through an obstacle course.

9:44AM G19.00009 Outsourcing neural active control to passive composite mechanics: a tissue engineered cyborg ray. , MATTIA GAZZOLA, University of Illinois at Urbana-Champaign, SUNG JIN PARK, Harvard University, KYUNG SOO PARK, University of Michigan, SHIRLEY PARK, Stanford University, VALENTINA DI SANTO, Harvard University, KARL DEISSEROTH, Stanford University, GEORGE V. LAUDER, L. MAHADEVAN, KEVIN KIT PARKER, Harvard University — Translating the blueprint that stingrays and skates provide, we create a cyborg swimming ray capable of orchestrating adaptive maneuvering and phototactic navigation. The impossibility of replicating the neural system of batoids fish is bypassed by outsourcing algorithmic functionalities to the body composite mechanics, hence casting the active control problem into a design, passive one. We present a first step in engineering multilevel “brain-body-flow” systems that couple sensory information to motor coordination and movement, leading to behavior. This work paves the way for the development of autonomous and adaptive artificial creatures able to process multiple sensory inputs and produce complex behaviors in distributed systems and may represent a path toward soft-robotic “embodied cognition”.

9:57AM G19.00010 Stiffness Modulation of Rayed Fins by Curvature¹ , KHOI NGUYEN, Yale

University, NING YU, Brown University, MADHUSUDHAN VENKADESAN, Yale University, MAHESH BANDI, Okinawa Institute of Science and Technology, SHREYAS MANDRE, Brown University — Fishes with rayed fins comprise over 99% of all extant fish species. Multifunctional use of fins, from propulsion to station holding, requires substantial modulation of stiffness. We propose that fishes stiffen the fin by curving it transverse to its length. This effect is similar to stiffening a dollar bill by curling it because of curvature-induced coupling of out-of-plane bending with in-plane stretching. Unlike a piece of paper, rayed fins are a composite of rays and membranes. We model this as parallel elastic beams (rays) with springy interconnections (membranes). Our analysis shows that the key parameters stiffening the fin are the ray anisotropy to bending, the misalignment of principal bending directions of adjacent rays, and the membrane elasticity. The composite fin stiffens when the principal bending directions of adjacent rays are misaligned due to fin curvature, which necessarily causes the membrane to stretch. Unlike a homogenous thin sheet, composite rayed structures are able to mimic curvature-induced stiffening by using misaligned rays even if the fin appears geometrically flat. Preliminary radiographic evidence from the rays of fish fins supports such a mechanism.

¹Funding by Human Frontier Science Program

Monday, November 21, 2016 8:00AM - 9:57AM –

Session G20 Bio: Wing effects on Insect flight D137-138 - Haibo Dong, University of Virginia

8:00AM G20.00001 New Insights on Insect’s Silent Flight. Part I: Vortex Dynamics and Wing Morphing¹ , YAN REN, GENG LIU, HAIBO DONG, University of Virginia, BIAO GENG, XUDONG ZHENG, QIAN XUE, University of Maine — Insects are capable of conducting silent flights. This is attributed to its specially designed wing material properties for the control of vibration and surface morphing during the flapping flight. In current work, we focus on the roles of dynamic wing morphing on the unsteady vortex dynamics of a cicada in steady flight. A 3D image-based surface reconstruction method is used to obtain kinematical and morphological data of cicada wings from high-quality high-speed videos. The observed morphing wing kinematics is highly complex and a singular value decomposition method is used to decompose the wing motion to several dominant modes with distinct motion features. A high-fidelity immersed-boundary-based flow solver is then used to study the vortex dynamics in details. The results show that vortical structures closely relate to the morphing mode, which plays key role in the development and attachment of leading-edge vortex (LEV), thus helps the silent flapping of the cicada wings.

¹This work is supported by AFOSR FA9550-12-1-0071 and NSF CBET-1313217.

8:13AM G20.00002 The effect of butterfly scales on flight efficiency and leading edge vortex formation¹, AMY LANG, JACOB WILROY, REDHA WAHIDI, The University of Alabama, NATHAN SLEGRERS, MICAHEL HEILMAN, George Fox University, JACOB CRANFORD, University of Alabama Huntsville — It is hypothesized that the scales on a butterfly wing lead to increased flight efficiency. Recent testing of live butterflies tracked their motion over 246 flights for 11 different specimens. Results show a 37.8 percent mean decrease in flight efficiency and a flapping amplitude reduction of 6.7 percent once the scales were removed. This change could be largely a result of how the leading edge vortex (LEV) interacts with the wing. 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, as well as the subsequent effect on the LEV's growth rate and peak circulation. For this experiment butterfly inspired grooves were created using additive manufacturing and were attached to a flat plate with a chordwise orientation, thus increasing plate surface area. The vortex generated by the grooved plate was then compared to a smooth case as the plate translated vertically through a tow tank at $Re = 1500, 3000$, and 6000 . Using DPIV, the vortex formation was documented and a maximum vortex formation time of 4.22 was found based on the flat plate travel distance and chord length. Results indicate that the patterned surface slows down the growth of the vortex which corroborates the flight test results.

¹Funding from NSF CBET Fluid Dynamics is gratefully acknowledged.

8:26AM G20.00003 Observations on Leading-Edge Vortex Development¹, MICHAEL GLENN, Alabama A&M University, AMY LANG, REDHA WAHIDI, JACOB WILROY, The University of Alabama — Most of an insect's lift comes from the leading edge vortex (LEV) that they produce when flapping their wings. There are many variables that make a LEV either stronger or weaker such as: roughness from the scales on their wings, angle of attack (AoA) of wing, size of the wing, and speed of the wing during flapping motion. Experiments were conducted to study LEV development to gain a better understanding of butterfly flight and the importance of LEV formation. The variables emphasized in this particular experiment were the chord length Reynolds numbers. Two smooth plates of 4 inches and 7 inches were compared in this experiment with Re of 1500 and 3000. Matlab was used to track the LEV location and calculate the vorticity and circulation magnitudes. Differences in LEV vortex strength as a function of chord length will be presented.

¹Funding was provided by NSF REU site grant EEC 1358991 and CBET grant 1628600

8:39AM G20.00004 Roughness Effects on the Formation of a Leading Edge Vortex¹, CASSIDY ELLIOTT, AMY LANG, REDHA WAHIDI, JACOB WILROY, The University of Alabama — Microscopic scales cover the wings of Monarch butterflies, creating a patterned surface. This patterning is an important natural flow control mechanism that is thought to delay the growth of the leading edge vortex (LEV) produced by the flapping motion of a wing. The increased skin friction caused by the scales leads to a weaker LEV being shed into the butterfly's wake, lessening drag and increasing flight efficiency. To test this theory, a plate of random roughness was designed in SolidWorks and printed on the Objet 30 Pro 3D printer. A 2x3x5 cubic foot tow tank was used to test the rough plate at Reynold's numbers of 1500, 3000, and 6000 (velocities of 8, 16, and 32 mm/s) at an angle of attack of 45 degrees. Particle Image Velocimetry (PIV) captured images of the LEV generated by the plate when towed upwards through the particle-seeded flow. Codes written in MatLab were used to automatically track and determine the strength of the LEV. Circulation values for the randomly-rough plate were then compared to the same values generated in a previous experiment that used a smooth plate and a grooved plate to determine the effect of the patterning on vortex development.

¹Funding provided by NSF REU site grant EEC 1358991 and CBET 1628600.

8:52AM G20.00005 New insights into insect's silent flight. Part II: sound source and noise control¹, QIAN XUE, BIAO GENG, XUDONG ZHENG, The University of Maine, GENG LIU, HAIBO DONG, University of Virginia — The flapping flight of aerial animals has excellent aerodynamic performance but meanwhile generates low noise. In this study, the unsteady flow and acoustic characteristics of the flapping wing are numerically investigated for three-dimensional (3D) models of Tibicen linnei cicada at free forward flight conditions. Single cicada wing is modelled as a membrane with prescribed motion reconstructed by Wan et al. (2015). The flow field and acoustic field around the flapping wing are solved with immersed-boundary-method based incompressible flow solver and linearized-perturbed-compressible-equations based acoustic solver. The 3D simulation allows examination of both directivity and frequency composition of the produced sound in a full space. The mechanism of sound generation of flapping wing is analyzed through correlations between acoustic signals and flow features. Along with a flexible wing model, a rigid wing model is also simulated. The results from these two cases will be compared to investigate the effects of wing flexibility on sound generation.

¹This study is supported by NSF CBET-1313217 and AFOSR FA9550-12-1-0071.

9:05AM G20.00006 Helical vortices generated by flapping wings of bumblebees¹, MARIE FARGE, LMD-CNRS, Ecole Normale Supérieure, Paris, France, THOMAS ENGELS, ISTA, Technische Universität Berlin, Berlin, Germany, DMITRY KOLOMENSKIY, Biomechanical Engineering Laboratory, Chiba University, Chiba, Japan, KAI SCHNEIDER, I2M-CNRS, Centre de Mathématiques et d'Informatique, Aix-Marseille Université, Marseille, France, FRITZ LEHMANN, Department of Animal Physiology, UniversiRostock, Rostock, Germany, JÖRN SESTERHENN, ISTA, Technische Universität Berlin, Berlin, Germany — We analyze high resolution numerical simulation data of a bumblebee with fixed body and prescribed wing motion, flying in a numerical wind tunnel, presented in [Engels et al., Phys. Rev. Lett., 116, 028103, 2016]. The inflow condition of the tunnel varies from unperturbed laminar to strongly turbulent. The flow generated by the flapping wings indicates the important role of the leading edge vortex (LEV), responsible for elevated lift production and which is not significantly altered by the inflow turbulence. The LEV has a conical structure due to the three-dimensional motion of the wings. This flow configuration produces strong vorticity on the sharp leading edge and the outwards velocity (from the root to the tip of the wing) in the spanwise direction. Flow visualizations show that the generated vortical structures are characterized by a strong helicity. We study the evolution of the mean helicity for each wing and analyze the impact of turbulent inflow.

¹We thankfully acknowledge financial support from the French-German AIFIT project funded by DFG and ANR (Grant 15-CE40-0019). DK gratefully acknowledges financial support from the JSPS postdoctoral fellowship.

9:18AM G20.00007 Effect of wing mass in free flight by a butterfly-like 3D flapping wing-body model¹, KOSUKE SUZUKI, Institute of Engineering, Academic Assembly, Shinshu University, IORI OKADA, Graduate School of Science and Technology, Shinshu University, MASATO YOSHINO, Institute of Engineering, Academic Assembly, Shinshu University — The effect of wing mass in free flight of a flapping wing is investigated by numerical simulations based on an immersed boundary-lattice Boltzmann method. We consider a butterfly-like 3D flapping wing-model consisting of two square wings with uniform mass density connected by a rod-shaped body. We simulate free flights of the wing-body model with various mass ratios of the wing to the whole of the model. As a result, it is found that the lift and thrust forces decrease as the mass ratio increases, since the body with a large mass ratio experiences large vertical and horizontal oscillations in one period and consequently the wing tip speed relatively decreases. In addition, we find the critical mass ratio between upward flight and downward flight for various Reynolds numbers.

¹This work was supported by JSPS KAKENHI Grant Number JP16K18012

9:31AM G20.00008 Aerodynamics of a comb-like plate mimicking a fairyfly wing, SEUNGHUN LEE, CHEOLGYUN JUNG, DAEGYOUN KIM¹, KAIST — There have been many studies on the aerodynamics of a wing with smooth surface in a wide range of the Reynolds number. Unlike smooth wings of common insects or birds, however, fairyfly has a distinctive wing geometry; a frame with several bristles. Motivated by the peculiar wing geometry of the fairyfly we experimentally investigated the fluid dynamics of a translating comb-like wing in a wide range of Reynolds number in $O(1) - O(10^3)$. We conducted the same experiment in several fluids of different viscosities in order to investigate the effects of the Reynolds number on the aerodynamic performance. Aerodynamic force of various wing shapes was measured, and it was correlated with the flow structure generated by the wing.

¹Corresponding author

9:44AM G20.00009 A Numerical Investigation of Two-Different *Drosophila* Forward Flight Modes, MEHMET SAHIN, EZGI DILEK, Istanbul Technical University, BELKIS ERZINCANLI, Kocaeli University — The parallel large-scale unstructured finite volume method based on an Arbitrary Lagrangian-Eulerian (ALE) formulation has been applied in order to investigate the near wake structure of *Drosophila* in forward flight. DISTENE MeshGems-Hexa algorithm based on the octree method is used to generate the all hexahedral mesh for the wing-body combination. The mesh deformation algorithm is based on the indirect radial basis function (RBF) method at each time level while avoiding remeshing in order to enhance numerical robustness. The large-scale numerical simulations are carried out for a flapping *Drosophila* in forward flight. In the first case, the wing tip-path plane is tilted forward to generate forward force. In the second case, paddling wing motion is used to generate the forward fore. The λ_2 -criterion proposed by Jeong and Hussain (1995) is used for investigating the time variation of the Eulerian coherent structures in the near wake. The present simulations reveal highly detailed near wake topology for a hovering *Drosophila*. This is very useful in terms of understanding physics in biological flights which can provide a very useful tool for designing bio-inspired MAVs.

Monday, November 21, 2016 8:00AM - 10:10AM –
Session G21 Bubbles: Cavitation, Acoustics and Biomedical D139-140 - Marc K. Smith,
Georgia Institute of technology

8:00AM G21.00001 Interfacial Dynamics of Condensing Vapor Bubbles in an Ultrasonic Acoustic Field, THOMAS BOZIUK, MARC SMITH, ARI GLEZER, Georgia Institute of Technology — Enhancement of vapor condensation in quiescent subcooled liquid using ultrasonic actuation is investigated experimentally. The vapor bubbles are formed by direct injection from a pressurized steam reservoir through nozzles of varying characteristic diameters, and are advected within an acoustic field of programmable intensity. While kHz-range acoustic actuation typically couples to capillary instability of the vapor-liquid interface, ultrasonic (MHz-range) actuation leads to the formation of a liquid spout that penetrates into the vapor bubble and significantly increases its surface area and therefore condensation rate. Focusing of the ultrasonic beam along the spout leads to ejection of small-scale droplets from that are propelled towards the vapor liquid interface and result in localized acceleration of the condensation. High-speed video of Schlieren images is used to investigate the effects of the ultrasonic actuation on the thermal boundary layer on the liquid side of the vapor-liquid interface and its effect on the condensation rate, and the liquid motion during condensation is investigated using high-magnification PIV measurements. High-speed image processing is used to assess the effect of the actuation on the dynamics and temporal variation in characteristic scale (and condensation rate) of the vapor bubbles.

8:13AM G21.00002 Reduced Order Modeling of Bubble Cloud Dynamics in a Focused Ultrasound Field, KAZUKI MAEDA, TIM COLONIUS, California Institute of Technology — In order to characterize the cloud cavitation in burst wave lithotripsy, reduced order modeling of the dynamics of a spherical bubble cloud of a radius $O(1)$ mm interacting with traveling ultrasound waves of an amplitude $O(1)$ MPa in water is presented. Bubbles are treated as spherical, radially oscillating cavities dispersed in continuous liquid phase. The volume of Lagrangian point bubbles is mapped with a regularization kernel as void fraction onto three-dimensional Cartesian grids that define the Eulerian liquid phase. The flow field is solved using a WENO-based compressible flow solver. The initial size and number density of the bubbles are critical for their coherent dynamics in the cloud, yet three-dimensional simulations of clouds with various parameters are computationally demanding. For further reduced-order modeling, a new kernel is introduced into the model to regularize bubbles onto two-dimensional, axisymmetric grids. The evolution of the void fraction and the maximum pressure in the cloud simulated using the model agree with results of three-dimensional simulations, while the reduction in computational cost is a factor of $O(100)$. Finally, the model is applied to a parametric study of the coherent dynamics of bubbles.

8:26AM G21.00003 Ultrasound induced bubble clusters and tunnels in tissue-mimicking agar phantoms¹, POOYA MOVAHED, Univ of Illinois - Urbana, WAYNE KREIDER, ADAM D. MAXWELL, MICHAEL R. BAILEY, Univ of Washington, JONATHAN B. FREUND, Univ of Illinois - Urbana — Soft tissue fractionation induced by acoustic cavitation is desired for non-invasive tissue removal in histotripsy, while being a potential injury mechanism in other therapeutic ultrasound treatments such as lithotripsy. In this work, we investigate the formation of bubble clusters and tunnels in tissue-mimicking agar phantoms by focused ultrasound bursts to inform a class of damage models. Agar phantoms of different stiffness were subjected to a series of multi-cycle ultrasound bursts, using a burst wave lithotripsy (BWL) protocol [Maxwell et al., J. Urol., 193, 338-344 (2015)], and simultaneously imaged at 200 frames per second (1 image per ultrasound burst). Some bubbles become visible in images (~200 microns) due to the negative pressure (~7.5 MPa) in the initial bursts, and the number of visible bubbles increases continuously during the subsequent bursts. A Rayleigh-Plesset-type bubble dynamics model, which accounts for viscoelastic confinement of agar gels, is developed. Material fatigue leading to eventual irreversible fracture-like failure in this model is proposed to explain the key observations. In addition to isolated, approximately spherical bubbles, long tunnel-like features are observed, which are seemingly lines of joined bubbles along a possible fracture or defect. The geometry of these tunnel-like features is quantified, and a physical explanation for tunnel formation is proposed in terms of bubble expansion and unstable collapse.

¹This work was supported by NIH NIDDK grant P01-DK043881.

8:39AM G21.00004 Temperature and stress fields produced by ultrasound-induced cavitation in a viscoelastic medium, LAUREN MANCIA, ERIC JOHNSEN, Univ of Michigan - Ann Arbor — Ultrasound contrast agents can act as cavitation nuclei that mechanically damage surrounding tissue when they oscillate in diagnostic ultrasound. Encapsulated microbubbles have also been proposed as a means to improve the efficiency and efficacy of therapeutic ultrasound by increasing the rate of tissue heating. However, the thermal and mechanical effects of cavitation are difficult to distinguish from each other and to quantify experimentally as they often occur simultaneously. To address this challenge, we study the cavitation-induced temperature and stress fields produced by a spherical bubble oscillating in a Kelvin-Voigt viscoelastic medium with nonlinear elasticity using a model that also accounts for energy transport inside and outside the bubble. We find that the primary contribution to heating is viscous dissipation, which itself depends on both the material (viscosity) and the bubble dynamics. We examine the rate of viscous heating and the magnitude of stresses over a relevant range of tissue properties and waveform parameters to determine regimes where heating is expected to be dominant. A means of estimating the expected significance of viscous dissipation from given tissue properties and waveform parameters is proposed.

8:52AM G21.00005 The hydrodynamic and ultrasound-induced forces on microbubbles under high Reynolds number flow representative of the human systemic circulation, ALICIA CLARK, ALBERTO ALISEDA, University of Washington Department of Mechanical Engineering — Ultrasound contrast agents (UCAs) are micron-sized bubbles that are used in conjunction with ultrasound (US) in medical applications such as thrombolysis and targeted intravenous drug delivery. Previous work has shown that the Bjerknes force, due to the phase difference between the incoming US pressure wave and the bubble volume oscillations, can be used to manipulate the trajectories of microbubbles. Our work explores the behavior of microbubbles in medium sized blood vessels under both uniform and pulsatile flows at a range of physiologically relevant Reynolds and Womersley numbers. High speed images were taken of the microbubbles in an in-vitro flow loop that replicates physiological flow conditions. During the imaging, the microbubbles were insonified at different diagnostic ultrasound settings (varying center frequency, PRF, etc.). An in-house Lagrangian particle tracking code was then used to determine the trajectories of the microbubbles and, thus, a dynamic model for the microbubbles including the Bjerknes forces acting on them, as well as drag, lift, and added mass. Preliminary work has also explored the behavior of the microbubbles in a patient-specific model of a carotid artery bifurcation to demonstrate the feasibility of preferential steering of microbubbles towards the intracranial circulation with US.

9:05AM G21.00006 Acoustic radiation force expansions in terms of partial wave phase shifts for scattering: Applications¹, PHILIP L. MARSTON, Washington State University, LIKUN ZHANG, University of Mississippi — When evaluating radiation forces on spheres in soundfields (with or without orbital-angular momentum) the interpretation of analytical results is greatly simplified by retaining the use of s-function notation for partial-wave coefficients imported into acoustics from quantum scattering theory in the 1970s. This facilitates easy interpretation of various efficiency factors [1]. For situations in which dissipation is negligible, each partial-wave s-function becomes characterized by a single parameter: a phase shift allowing for all possible situations. These phase shifts are associated with scattering by plane traveling waves and the incident wavefield of interest is separately parameterized. (When considering outcomes, the method of fabricating symmetric objects having a desirable set of phase shifts becomes a separate issue.) The existence of negative radiation force “islands” for beams reported in 2006 by Marston is manifested. This approach and consideration of conservation theorems [2] illustrate the unphysical nature of various claims made by other researchers. This approach is also directly relevant to objects in standing waves. [1] L. Zhang and P. L. Marston, J. Acoust. Soc. Am. 140, EL178-EL183 (2016). [2] P. L. Marston and L. Zhang, J. Acoust. Soc. Am. 139, 3139-3144 (2016).

¹Supported by ONR.

9:18AM G21.00007 Revisiting the potential for bursting bubbles to damage cells below the free surface¹, PETER WALLS, JAMES BIRD, Boston University — The rapid motion associated with bubbles bursting at the surface of a liquid is known to cause damage to cells in a suspension, which is particularly problematic in bioreactors that require continuous injection of oxygen to sustain the cells. It is generally accepted that cells directly attached to the bubble's interface will experience lethal levels of damage. To prevent cells from initially attaching to the bubble's surface, surfactants are widely used. However, the potential for bursting bubbles to damage nearby, but not directly attached, cells is less clear. Previous numerical studies have predicted maximum energy dissipation rates (EDR) as high as 10^{10} W/m³ for bubbles with radii less than 1 mm; lethal to the commonly used mammalian CHO cell. Here we show that these studies tend to underestimate the generated EDR levels by several orders of magnitude due to limited numerical mesh resolution. Furthermore, we demonstrate how a downward traveling jet can cause damage away from the interface. We validate our numerical model with high-speed bubble bursting experiments and relate the dynamics of this downward jet to the boundary layer equations. We anticipate our results will be an integral step towards developing more efficient aeration platforms.

¹We acknowledge support from Biogen Inc.

9:31AM G21.00008 Nonlinear oscillation and interfacial stability of an encapsulated microbubble under dual-frequency ultrasound, MICHAEL CALVISI, University of Colorado, Colorado Springs, YUNQIAO LIU, Shanghai Jiao Tong University, QIANXI WANG, University of Birmingham — Encapsulated microbubbles (EMBs) are widely used in medical ultrasound imaging as contrast-enhanced agents. However, the potential damaging effects of violent, collapsing EMBS to cells and tissues in clinical practice have remained a concern. Dual-frequency ultrasound is a promising technique for improving the efficacy and safety of sonography. The EMB system modeled consists of the external liquid, membrane, and internal gases. The microbubble dynamics are simulated using a simple nonlinear interactive theory, considering the compressibility of the internal gas, viscosity of the liquid flow, and elasticity of the membrane. The radial oscillation and interfacial stability of an EMB under single and dual-frequency excitations are compared. The simulation results show that the dual-frequency technique produces larger backscatter pressure at higher harmonics of the primary driving frequency. This enriched acoustic spectrum can enhance blood-tissue contrast and improve sonographic image quality. The results further show that the acoustic pressure threshold associated with the onset of shape instability is greater for dual-frequency driving. This suggests that the dual-frequency technique stabilizes the EMB, thereby improving the efficacy and safety of contrast-enhanced agents.

9:44AM G21.00009 Effects of non-condensable gas on the dynamic oscillations of cavitation bubbles¹, YUNING ZHANG, North China Electric Power University — Cavitation is an essential topic of multiphase flow with a broad range of applications. Generally, there exists non-condensable gas in the liquid and a complex vapor/gas mixture bubble will be formed. A rigorous prediction of the dynamic behavior of the aforementioned mixture bubble is essential for the development of a complete cavitation model. In the present paper, effects of non-condensable gas on the dynamic oscillations of the vapor/gas mixture bubble are numerically investigated in great detail. For the completeness, a large parameter zone (e.g. bubble radius, frequency and ratio between gas and vapor) is investigated with many demonstrating examples. The mechanisms of mass diffusion are categorized into different groups with their characteristics and dominated regions given. Influences of non-condensable gas on the wave propagation (e.g. wave speed and attenuation) in the bubbly liquids are also briefly discussed. Specifically, the minimum wave speed is quantitatively predicted in order to close the pressure-density coupling relationship usually employed for the cavitation modelling. Finally, the application of the present finding on the development of cavitation model is demonstrated with a brief discussion of its influence on the cavitation dynamics.

¹This work was financially supported by the National Natural Science Foundation of China (Project No.: 51506051).

9:57AM G21.00010 Bubble absorption by an air-filled helically-supported capillary channel, NEGAR BEHESHTIPOUR, DAVID THIESSEN, Washington State University — Gas-liquid phase separation under microgravity conditions where buoyancy is not active represents a challenge for two-phase liquid-continuous space systems. Similar challenges are present in micro-scale electrochemical systems on Earth that generate gas bubbles in geometries where surface tension prevails over gravity. A possible ground-based application would be the removal of carbon dioxide bubbles from large aspect ratio channels in a direct-methanol fuel cell that could otherwise occlude the channel. In this study we use a 3-mm diameter stretched stainless-steel spring coated with a superhydrophobic layer to create a helically-supported capillary channel. Such a channel that is submerged in water and filled with air while vented to the atmosphere was found to absorb a stream of 2.5-mm diameter air bubbles at a rate of at least 36 bubbles/s. An optical detector and high-speed imaging system have been used to study bubble absorption dynamics. A significant finding is that the initial attachment of the bubble to the channel that involves the rupture of a thin film of water happens in less than 1 ms. The rapid rupture of the water film separating the bubble from the channel might be attributed to the roughness of the hydrophobic coating.

Monday, November 21, 2016 8:00AM - 10:10AM –
Session G22 Microscale Flows: General E141/142 - Andres Tejada-Martinez, University of South Florida

8:00AM G22.00001 Fabrication of Converging and Diverging Polymeric Microlens Arrays By A Thermocapillary Replication Technique¹, SOON WEI DANIEL LIM, KEVIN R. FIEDLER, SANDRA M. TROIAN, California Institute of Technology, 1200 E. California Blvd. MC 128-95, Pasadena, CA — Thermocapillary forces offer a powerful method for sculpting air/liquid interfaces at microscale dimensions. Here we demonstrate how square arrays of slender chilled pins in close proximity to a molten nanofilm enforce periodic distributions of thermocapillary stresses suitable for fabricating microlens arrays with ultrasmooth surfaces and excellent focusing capability. We applied this technique to shape and then solidify polystyrene films on quartz to form converging and diverging microlens arrays. By adjusting the growth time, width of the chilled pins, and pin pitch, we created simple convex, simple concave, caldera-like and even hierarchical microarray components. The latter two tend to form when the pitch and pin width are comparable in size. The diverging arrays were incorporated into a Shack-Hartmann wavefront sensor for imaging spatial fluctuations in refractive index caused by bursts of cooled spray. The caldera-like arrays were used to collimate an incident beam into annuli. These demonstrations illustrate how spatiotemporal control over thermocapillary distributions can be used to fabricate a multiplicity of micro-optical components in a single, non-contact step.

¹This work was supported by the Kiyo and Eiko Tomiyasu SURF scholarship (SWDL) and an NSTRF fellowship (KRF).

8:13AM G22.00002 Control of smearing during wiping stage of gravure printing of electronics, UMUT CEYHAN, University of California, Berkeley; Izmir Katip Celebi University, S. J. S. MORRIS, University of California, Berkeley — During gravure printing, a blade wipes the excess liquid from the engraved gravure roll, the objective is leaving liquid filled cells defining the image to be printed. Capillarity, however, draws some liquid from cell into a meniscus connecting to the blade; and the continuing motion of gravure roll smears that meniscus over its surface. Smearing delivers features lacking in sharpness at the micron scale. Ceyhan and Morris (BAPS.2015.DFD.M8.10) analyze smear formation for the blade-liquid contact angles covering the range $0 \leq \theta < \pi/2$ and show that the problem can be treated in plane. Using numerical solutions of the corresponding free boundary problem for the Stokes equations of motion, we show¹ that hydrostatic theory provides an upper bound for the smear volume for finite Ca . The approach of trailing edge of the cell to the meniscus isolates a certain volume of liquid from the cell, the isolated liquid is deformed with the continuing motion of gravure roll. The problem of controlling smear formation now reduces to the simpler problem of reducing the quantity of liquid drawn into the meniscus. The theory explains why polishing to reduce the tip radius of the blade is an effective way to control smearing.

¹Ceyhan and Morris, *J. Fluid Mech.* submitted 2016

8:26AM G22.00003 The unified slip boundary condition: addressing the breakdown of the no-slip boundary condition¹, JOSEPH THALAKKOTTOR, KAMRAN MOHSENI, University of Florida — The no-slip boundary condition has been contested for over a century. Although it has been successful in reproducing most continuum and macroscopic results, the condition breaks down in situations such as contact line motion, corner flow and in many micro- and/or nano-scale applications. The widely used Maxwell and Navier slip boundary conditions make an implicit assumption that velocity varies only in the wall normal direction. This assumption is not applicable in the vicinity of a contact and a corner point, where velocity varies in wall-normal and wall-tangential directions. Here, we present a generalized velocity boundary condition that shows that slip velocity is a function of not only the shear rate but also the linear strain rate. In addition, we present a universal relation for slip length which shows that, for a general flow, slip length is a function of the principal strain rate. The universal relation for slip length along with the generalized velocity boundary condition provides a unified slip boundary condition to model a wide range of steady Newtonian fluid flows.

¹Office of Naval Research

8:39AM G22.00004 Enhanced oil recovery with polymer flooding., SHIMA PARSA, DAVID WEITZ, Harvard University — Polymer flooding is a method for enhanced oil recovery, however the mechanism responsible for the effectiveness of polymer flooding is not well understood. We use confocal microscopy and bulk transport measurements to probe the effectiveness of different molecular weight and concentrations of Polyacrylamide solution in imbibition of crude oil in 3D micromodel. We show that large molecular weight and moderate to high concentration of polymer is required for enhanced recovery. By directly measuring the pore level velocities in the medium, we show that polymer retention in the medium results in diversion of flow in some pores. The inhomogeneous changes in the flow velocities result in redistribution of viscous forces and enhanced recovery of oil.

8:52AM G22.00005 Nanoscale measurement of apparent slip velocity near a moving contact line¹, JOONSIK PARK, KENNETH BREUER, Brown University, BROWN UNIVERSITY TEAM — We report the nanoscale flow measurements within tens of microns from a moving contact line on hydrophobic substrates. A moving contact line was generated using a liquid bridge instability induced by retreating syringe. Contact line speeds ranging from 0.15 to 3 mm/s were recorded. The motions of tracer nanoparticles were measured using two independent experimental techniques: multi-layer flood illumination and Total Internal Reflection Fluorescence Microscopy. The flow field was derived using a novel probabilistic particle tracking velocimetry, which allows the accurate estimation of the rapidly changing flow field near a contact line without bias due to binning or fitting. The results confirm that for distances larger than a few microns from the contact line, the velocity field scales with the instantaneous contact line speed and agrees well with the corner flow solution predicted by the biharmonic equation. A significant slip velocity is shown to exist close to the contact line, decaying rapidly within a few microns.

¹The authors gratefully acknowledge the National Science Foundation, grants CBET 0854148 and CBET 106614, for the support of this research

9:05AM G22.00006 Fluid mechanical proximity effects in high-resolution gravure printing for printed electronics¹, GERD GRAU, York University, WILLIAM J. SCHEIDELER, VIVEK SUBRAMANIAN, University of California, Berkeley — Gravure printing is a very promising method for printed electronics because it combines high throughput with high resolution. Recently, printed lines with 2 micrometer resolution have been demonstrated at printing speeds on the order of 1m/s. In order to build realistic circuits, the fluid dynamics of complex pattern formation needs to be studied. Recently, we showed that highly-scaled lines printed in close succession exhibit proximity effects that can either improve or deteriorate print quality depending on a number of parameters. It was found that this effect occurs if cells are connected by a thin fluid film. Here, we present further experimental and modeling results explaining the mechanism by which this thin fluid film affects pattern formation. During the transfer of ink from the roll to the substrate, ink can flow in between connected cells. Asymmetry in the fluid distribution created by the preceding doctor blade wiping process results in net fluid flow from cells that transfer first to cells that transfer subsequently. The proximity of these cells thus affects the final ink distribution on the substrate, which is critically important to understand and design optimally when printing highly-scaled patterns of electronic materials.

¹This work is based upon work supported in part by the National Science Foundation under Cooperative Agreement No. EEC-1160494.

9:18AM G22.00007 Interfacial melting of ice under a high-speed slider: real-time visualization and friction modeling, HYUNG-SEOK KIM, CHANG-HO YUN, DONG-JO KIM, HO-YOUNG KIM, Seoul National University — When a solid plate slides on ice, frictional heat melts asperities on the ice surface causing the real contact area to increase. Previous studies indicate the significance of contact area growth for ice friction, yet its quantitative understanding is far from clear mainly because the direct observation of the melting process at the interface has been extremely difficult. Here we describe a novel experimental setup that visualizes the interface of a rapidly rotating ice disc (up to the linear velocity of 10 m/s) and a transparent quartz surface in real time using the total internal reflection. We find that the melted area of the ice surface is a sensitive function of both sliding speed and temperature. We rationalize such quantitative measurements numerically and analytically, which allows us to predict the friction coefficient of ice as a function of relative velocity and temperature. This work can be used to develop friction-controlling mechanisms on ice surface, which are important in traffic safety as well as winter sports.

9:31AM G22.00008 A unifying framework for mass transfer dynamics in the Taylor flow of a dissolving train of bubbles, GHATA NIRMAL, ARUN RAMCHANDRAN, University of Toronto — Operation in the Taylor flow regime in microfluidics for estimation of mass transfer coefficients in multiphase flows has gained popularity due to the presence of high interfacial areas and well-characterized flow profiles. Although there are multiple models available for data interpretation, these are accompanied by two major limitations. First, mass transfer from the lubricating liquid film to the bulk liquid segment between bubbles has been incorrectly estimated. Second, the liquid segment is assumed to be well mixed. Both assumptions fail in the normal operating limits for Taylor flow experiments of dissolving bubbles. In this work, we rectify the two limitations described above and present a unifying framework to comprehend experimental results in a dissolving train of bubbles in microchannels. Based on a scaling analysis, the experiments can be operated in four regimes controlled by L_B / R , L_L / R , Peclet number and capillary number where L_B , L_L and R are the bubble length, the liquid segment length and the tube radius, respectively. Finally, we present the differences in the results due to a rectangular cross-sectional shape instead of a circular one, and in particular, on the additional leakage flux through the lubricating film around the corners of the cross-section.

9:44AM G22.00009 Conformation and stretching of end-tethered polymers in pressure-driven flow under confinement.¹, TAMAL ROY, STEFFEN HARDT, Institute for Nano- and Microfluidics, Technische Universität Darmstadt, INSTITUTE FOR NANO- AND MICROFLUIDICS, TECHNISCHE UNIVERSITÄT DARMSTADT TEAM — Understanding of the conformation and dynamics of polymers under confinement is important for both fundamental studies and applications. We experimentally study the conformation and stretching of surface-tethered polymer chains confined between parallel surfaces and exposed to a pressure-driven flow. λ -DNA molecules are tethered to the wall of a microchannel of height smaller than the contour lengths of the molecules. The DNA molecules, stained with a fluorescent dye, are visualized by epifluorescence and laser-scanning confocal microscopy (LSCM). The effects of the channel height, flow rate and contour length on the extension of the molecules are determined from epifluorescence images. From LSCM images the complete conformation and orientation of the DNA molecules is inferred. We find that the fractional extension of the molecules is uniquely determined by the fluid shear stress at the tethering surface and the chain contour length. There is no explicit influence of the channel height in the range of contour lengths we consider. We also derive analytical scaling relationships (in the weak and strong extension limits) that explain the experimentally observed stretching characteristics.

¹This work is supported by Deutsche Forschungsgemeinschaft (grant No. HA 2696/33-1)

9:57AM G22.00010 Comparison of Simulated and Measured Fluid-Surface Oscillation Frequencies in a Channel¹, MATTHEW TRAPUZZANO, KIESHA PIERRE, EMRE TUFECIOGLU, RASIM GULDIKEN, ANDRES TEJADA-MARTINEZ, NATHAN CRANE, Univ of South Florida — Many important processes from agriculture to manufacturing depend on the wetting of fluids on rough or textured surfaces. This has traditionally been studied from a macro-perspective. The effects of these surface features can be dramatically altered by vibrations that overcome energy barriers to contact line motion caused by surface roughness. In order to study these effects in confined geometries and at different length scales, a validated model is required. This presentation will compare the measured and simulated frequencies of capillary vibrations in a cylindrical glass tube. Fluid surface vibrations are excited externally through deformation of the interface. The resulting surface oscillations are observed with a high speed video camera and the dominant oscillation frequencies are calculated. The measured oscillation frequencies are compared to predictions from transient CFD simulations across a range of interface diameters from 400 μm to 1.5 mm. These results may be used to inform studies of wetting under vibration.

¹NSF CMMI-1361919

Monday, November 21, 2016 8:00AM - 10:10AM –
Session G25 Microscale Flows: Drops and Bubbles-I E145 - Thomas Cubaud, Stony Brook University

8:00AM G25.00001 Microfluidic destabilization of viscous stratifications: Interfacial waves and droplets¹, XIAOYI HU, THOMAS CUBAUD, Stony Brook University — Microfluidic two-fluid flows with large differences in viscosity are experimentally investigated to examine the role of fluid properties on hydrodynamic destabilization processes at the small scale. Two- and three-layer flow configurations are systematically studied in straight square microchannels using miscible and immiscible fluid pairs. We focus our attention on symmetric three-layer stratifications with a fast central stream made of low-viscosity fluid and a slow sheath flow composed of high-viscosity fluid. We quantify the influence of the capillary and the Reynolds numbers on the formation and evolution of droplets and wavy stratifications. Several functional relationships are developed for the morphology and dynamics of droplets and interfacial waves including size, celerity and frequency. In the wavy stratification regime, the formation and entrainment of thin viscous ligaments from wave crests display a rich variety of dynamics either in the presence or in the absence of interfacial tension between liquids.

¹This work is supported by NSF (CBET-1150389)

8:13AM G25.00002 ABSTRACT WITHDRAWN –

8:26AM G25.00003 Polymer Droplet Dynamic Wetting Measurement at the Nanometer Scale on Smooth Surfaces Using Atomic Force Microscopy, MOHAMMADREZA SOLEYMANIHA, JONATHAN ROBERT FELTS¹, Texas A&M University, ANML TEAM² — Fluid spreading is a complex phenomenon driven strongly by intermolecular forces that requires nanometer scale microscopy to observe and understand. We present a technique for measuring molten polymer spreading dynamics with nanometer scale spatial resolution at elevated temperatures on sapphire, silicon oxide and mica using tapping-mode atomic force microscopy (AFM). The experimental setup is used to measure the spreading dynamics of polystyrene droplets with 2 μm diameters at 115–175 $^{\circ}\text{C}$. Custom image processing algorithms realize the droplet height, radius, volume and contact angle of the droplet over time. The contact angle evolution followed a power law with time with experimental exponent values of -0.26, -0.08, and -0.2 for sapphire, silicon oxide, and mica, respectively at 115 $^{\circ}\text{C}$. The non-zero steady state contact angles result in a slower evolution of contact angle with time compared to Tanner's Law, as expected. We observe local crystallinity on the molten droplet surface, where crystalline structures appear to nucleate at the contact line and migrate toward the top of the droplet. Increasing the temperature from 115 $^{\circ}\text{C}$ to 175 $^{\circ}\text{C}$ reduced surface crystallinity from 35% to 12%, consistent with increasingly energetically favorable amorphous phase as the temperature approaches the melting temperature. This platform provides a way to measure spreading dynamics of extremely small volumes of heterogeneously complex fluids not possible through other means.

¹Dr.Jonathan Felts is the principal investigator of the ANML research group in Mechanical Engineering Department of Texas A&M University

²<http://felts.tamu.edu/index.html>

8:39AM G25.00004 Bubble deformations in corrugated microchannels at large capillary numbers¹, THOMAS CUBAUD, MARTIN SAUZADE, Stony Brook University — Multiphase flows in confined microgeometries display a variety of intriguing dynamics. Here, we experimentally examine trains of monodisperse gas bubbles of different sizes and concentrations passing through a series of extensions and constrictions from low to large capillary numbers. Using highly viscous carrier fluids, we show in particular that bubbles strongly deform in velocity fields set with the channel geometry. We measure the instantaneous front and rear velocities of periodically distorted capillary surfaces and develop functional relationships for predicting the morphology of multiphase flow patterns at the pore scale.

¹This work is supported by NSF (CBET-1150389)

8:52AM G25.00005 A Numerical Analysis of Droplet Breakup in Asymmetric T-Junctions with Different Outlet Pressure Gradients.¹, WAY LEE CHENG, Texas AM University at Qatar, ARUM HAN, Texas AM University, REZA SADR, Texas AM University at Qatar — Droplet splitting is the breakup of a parent droplet into two or more daughter droplets of desired sizes. It is done to improve production efficiency and investigational capacity in microfluidic devices. Passive splitting is the breakup of droplets into precise volume ratios at predetermined locations without external power sources. In this study, a 3-D simulation was conducted using the Volume-of-Fluid method to analysis the breakup process of a droplet in asymmetric T-junctions with different outlet arm lengths. The arrangement allows a droplet to be split into two smaller droplets of different sizes, where the volumetric ratio of the daughter droplets depends on the length ratios of the outlet arms. The study identified different breakup regimes such as primary, transition, bubble and non-breakup under different flow conditions and channel configurations. Furthermore, a close analysis to the primary breakup regimes were done to determine the breakup mechanisms at various flow conditions. The analysis show that the breakup mechanisms in asymmetric T-junctions is different than a regular split. A pseudo-phenomenological model for the breakup criteria was presented at the end. The model was an expanded version to a theoretically derived model for the symmetric droplet breakup.

¹The Qatar National Research Fund (a member of the Qatar Foundation), under Grant NPRP 5-671-2-278, supported this work.

9:05AM G25.00006 Using Microfluidics for Droplet and Particle Characterization of Environmental Fluids, ANDREW METCALF, CHRIS HOGAN, CARI DUTCHER, Univ of Minn - Minneapolis — Two-phase flows in microfluidic platforms enable high-throughput experiments for measurement of rheological, thermodynamic, and kinetic interfacial properties. In this talk, I will highlight biphasic microfluidic studies with environmental applications, including atmospheric aerosol properties and water contamination. For example, the fate of atmospheric aerosol particles can be profoundly affected by the presence of surface-active species within the aerosol liquid. In this work, the presence of these species is detected with microfluidic interfacial tensiometry, in which the behavior of the droplet interface under extensional shear is measured. Both secondary organic aerosol chemical mimics and aerosol filter extracts are used in droplet generation. In addition, preliminary work for use of the platform for water treatment applications will be highlighted. Particulate contamination in water can be detected by freezing contaminated droplets at different temperatures to study the perturbed thermodynamic state.

9:18AM G25.00007 Movement of liquid droplets containing polymers on substrate, GUOHUI HU, HENG WANG, Shanghai Institute of Applied Mathematics and Mechanics, Shanghai University — It is of both fundamental and practical interests to study the flow physics in the manipulation of droplets. As a microreactor, the macromolecules or particles inside the droplets might have significant influences on their movement. In the present study, the many-body dissipative particle dynamics (MDPD) is utilized to investigate the translocation of droplets containing polymer on a substrate driven by the wettability gradient, where the polymer is modelled as worm-like chain (WLC). The internal flows of the droplets are analyzed, as well as the comparison to the polymer-free moving droplets. The effects of physical parameters, such as the interaction potential between liquid particle and polymer beads, the mass of the beads, on the translocation speed are also addressed in the present study. These results might be helpful to the optimization in design of the microfluidic systems.

9:31AM G25.00008 Simulation of the self-assembly of colloidal droplets in a micro-channel¹, ZHOUYANG GE, LUCA BRANDT, KTH Mechanics, Sweden — In colloidal sciences, much progress has been made on the synthesis of complex building blocks mimicking molecular structures to elaborate innovative materials. The basic elements of such colloidal molecules are particles or droplets less than one millimeter in size. Their self-assembly relies on either lengthy brownian motion or careful microfluidic designs, on top of typical colloidal interactions, e.g. depletion attraction. Regardless of the approach, however, questions remain why the colloids undergo certain path to organize themselves and how such process can be optimized. Here, we perform direct numerical simulations using a Navier-Stokes solver at low Reynolds number, combined with either the immersed boundary method (IBM) or a newly-proposed level set (LS) method for interface description. In the IBM simulations, the colloids are treated as rigid, spherical particles under a Lennard-Jones-like potential, reproducing attractive depletion force. Results show that, for four particles, a planar diamond is formed under a weak potential while a 3D tetrahedron is formed under a strong potential, which agree qualitatively with experiments. In the next step, LS simulation of colloidal droplets will be performed to investigate the roles of surface tension in the self-assembly.

¹This project has received funding from the European Unions Horizon 2020 research and innovation programme under grant agreement No 664823.

9:44AM G25.00009 Inertia effects on bubble generation in thin T-junction microchannel, KAZUYASU SUGIYAMA, HIDEHIKO OKUBO, SEIGO NABESHIMA, TOMOAKI WATAMURA, Graduate School of Engineering Science, Osaka University — A numerical study on gas-liquid interface dynamics of bubble generation in a thin microchannel with a squeezed T-junction is performed. In consideration of liquid inertia, the basic equations consist of the Laplace law and the two-dimensional Euler-Darcy equation under the assumption of Hele-Shaw's flow owing to a large width-to-thickness aspect ratio of the channel cross-section. The velocity potential and the interface motion are numerically predicted by means of a boundary element method. The simulated results reasonably capture the experimentally observed behaviors that the interface pinches off at the channel junction and then a bubble forms. For a fixed liquid velocity, the generated bubble is likely to be smaller with decreasing the gas pressure, but the bubble is no longer generated at the gas pressure below a threshold. The bubble size minimized at the generation limit is arranged using the capillary, Reynolds and Weber numbers, and the results imply the significance of the liquid inertia in the bubble generation process in spite of the micrometer-scale phenomena.

9:57AM G25.00010 A close-up view of a pancake droplet in the microfluidic chips¹

, LAILAI ZHU, FRANCOIS GALLAIRE, Laboratory of Fluid Mechanics and Instabilities (LFMI), EPFL — We develop a boundary integral method to study the droplet dynamics in confined geometries in the low-capillary-number regime, where the lubrication film between the droplet and solid boundaries becomes important. We investigate a translating droplet tightly squeezed in a Hele-Shaw cell. The cell gap width is around $0.5 \sim 0.85$ the radius of a relaxed droplet and the capillary number is in the range [0.007, 0.16]. We highlight the three-dimensional feature of the droplet interface and flow field. The interface develops an arc-shaped ridge near the rear-half rim with a protrusion in the rear and a laterally symmetric pair of higher peaks; this pair of protrusions has been identified by recent experiments ² and predicted asymptotically ³. The mean film thickness is well predicted by the extended Bretherton model with fitting parameters. Flow fields with recirculation patterns are presented. On the horizontal plane, a dipolar disturbance flow field is identified and its $1/r^2$ spatial decay is confirmed numerically.

¹The European Research Council is acknowledged for funding the work through a starting grant (ERC SimCoMiCs 280117).

²Huerre et al., Phys. Rev. Lett., vol. 115 (6), 2015, 064501

³Burgess & Foster, Phys. Fluids A, vol. 2 (7), 1990, pp. 1105-1117

Monday, November 21, 2016 8:00AM - 10:10AM –

Session G26 Reacting Flows: Experiments I E146 - Mark Short, Los Alamos National Laboratory

8:00AM G26.00001 Dynamics of Detonation Propagation in Two-Dimensional

Curved Geometries , MARK SHORT, JAMES QUIRK, CARLOS CHIQUETTE, CHAD MEYER, Los Alamos National Laboratory — A detonation is a complex, compressible flow-reaction driven structure consisting of a lead shock wave and subsequent reaction zone in which reactants are converted into products. In condensed-phase explosives, the generated high pressures lead to yielding of confinement, and detonation reaction zone structure becomes multidimensional. The detonation structure and speed is then determined by a complex interaction between streamline divergence, compressibility and reaction. Curved geometries are important for understanding the effects of geometry on the compressible-reactive flow mechanisms of detonation propagation, as it includes elements of shock diffraction, flow divergence and boundary interactions. Here, we study this complex compressible flow evolution in two-dimensional curved geometries, highlighting the relation between the detonation motion and induced curvature of the detonation front.

8:13AM G26.00002 Body-Fitted Detonation Shock Dynamics and the Pseudo-Reaction-Zone Energy Release Model , CHAD MEYER, JAMES QUIRK, MARK SHORT, CARLOS CHIQUETTE,

Los Alamos National Laboratory — Programmed-burn methods are a class of models used to propagate a detonation wave, without the high resolution cost associated with a direct numerical simulation. They separate the detonation evolution calculation into two components: timing and energy release. The timing component is usually calculated with a Detonation Shock Dynamics model, a surface evolution representation that relates the normal velocity of the surface (D_n) to its local curvature. The energy release component must appropriately capture the degree of energy change associated with chemical reaction while simultaneously remaining synchronized with the timing component. The Pseudo-Reaction-Zone (PRZ) model is a reactive burn like energy release model, converting reactants into products, but with a conversion rate that is a function of the DSD surface D_n field. As such, it requires the DSD calculation produce smooth D_n fields, a challenge in complex geometries. We describe a new body-fitted approach to the Detonation Shock Dynamics calculation which produces the required smooth D_n fields, and a method for calibrating the PRZ model such that the rate of energy release remains as synced as possible with the timing component. We show results for slab, rate-stick and arc geometries.

8:26AM G26.00003 Shock Polar Angles and Confinement Effect on Detonation Propagation , CARLOS CHIQUETTE, MARK SHORT, CHAD MEYER, JAMES QUIRK, JOHN BDZIL, Los Alamos National Laboratory — In

high explosive (HE) engineering applications, the shape of a detonation front is influenced by the density and impedance of the inert material that surrounds the explosive. Where the detonation shock intersects the material boundary, a number of gasdynamic reflection patterns are possible involving shocks, Prandtl-Meyer fans and material interfaces. To leading-order, these reflection patterns can be predicted through a shock polar analysis. For the commonly used Detonation Shock Dynamics (DSD) front surface propagation model, the shape and evolution of the detonation wave is determined by the specification of the surface wave angle at the HE charge-confiner interface. Typically, the shock polar analysis is employed to approximate this necessary “edge angle” using specified equations of state for the HE-inert pair and a given phase velocity. For engineering applications, we need to evaluate how accurately a shock polar analysis can predict the DSD model edge-angle. We extend previous on this issue examining reactive flow simulations of detonation propagation in a confined HE compared to the predictions of a shock polar analysis.

8:39AM G26.00004 Effect of fuel stratification on detonation wave propagation , DAMIEN MASSELOT¹, ROMAIN FIEVET², VENKAT RAMAN³, University of Michigan - Ann Arbor — Rotating detonation engines (RDEs)

form a class of pressure-gain combustion systems of higher efficiency compared to conventional gas turbine engines. One of the key features of the design is the injection system, as reactants need to be continuously provided to the detonation wave to sustain its propagation speed. As inhomogeneities in the reactant mixture can perturb the detonation wave front, premixed fuel jet injectors might seem like the most stable solution. However, this introduces the risk of the detonation wave propagating through the injector, causing catastrophic failure. On the other hand, non-premixed fuel injection will tend to quench the detonation wave near the injectors, reducing the likelihood of such failure. Still, the effects of such non-premixing and flow inhomogeneities ahead of a detonation wave have yet to be fully understood and are the object of this study. A 3D channel filled with O₂ diluted in an inert gas with circular H₂ injectors is simulated as a detonation wave propagates through the system. The impact of key parameters such as injector spacing, injector size, mixture composition and time variations will be discussed.

¹PhD Candidate

²PhD Candidate

³Associate Professor

8:52AM G26.00005 Fast Hydrogen-Air Flames for Turbulence Driven Deflagration to Detonation Transition¹, JESSICA CHAMBERS, Propulsion and Energy Research Laboratory, University of Central Florida, KAREEM AHMED, Propulsion and Energy Research Laboratory University of Central Florida — Flame acceleration to Detonation produces several combustion modes as the Deflagration-to-Detonation Transition (DDT) is initiated, including fast deflagration, auto-ignition, and quasi-detonation. Shock flame interactions and turbulence levels in the reactant mixture drive rapid flame expansion, formation of a leading shockwave and post-shock conditions. An experimental study to characterize the developing shock and flame front behavior of propagating premixed hydrogen-air flames in a square channel is presented. To produce each flame regime, turbulence levels and flame propagation velocity are controlled using perforated plates in several configurations within the experimental facility. High speed optical diagnostics including Schlieren and Particle Image Velocimetry are used to capture the flow field. In-flow pressure measurements acquired post-shock, detail the dynamic changes that occur in the compressed gas directly ahead of the propagating flame. Emphasis on characterizing the turbulent post-shock environment of the various flame regimes helps identify the optimum conditions to initiate the DDT process. The study aims to further the understanding of complex physical mechanisms that drive transient flame conditions for detonation initiation.

¹American Chemical Society

9:05AM G26.00006 Experimental Study on Dynamics of 2H₂/O₂/2Ar Detonations with a Constant Mass Divergence, QIANG XIAO, Univ of Ottawa, JIAJIN CHANG, Univ of Toronto, MAXIME LA FLECHE, MATEI I. RADULESCU, Univ of Ottawa — Very recently, Borzou and Radulescu (2016) formulated a novel solution allowing for an easy and precise quantification of loss effects during detonation propagation involving an exponentially shaped channel. They found that the detonation dynamics departed from the ZND model predictions, particularly for very unstable detonations. The question arises if the ZND model can predict the dynamics of much less unstable mixtures, in spite of the presence of a cellular structure. The present study focuses on a more stable mixture of 2H₂/O₂/2Ar with better known reaction kinetics. The results obtained experimentally for the velocity deficit in terms of the amount of mass divergence were found in excellent agreement with the predictions made with the ZND model, in spite of the detonation reaction zone being organized in strong cellular structures with reactive transverse waves.

9:18AM G26.00007 Acquisition of high-fidelity flyer characteristics using PDV and streak imaging¹, JOSEPH OLLES, RYAN WIXOM, J. PATRICK BALL, Sandia National Laboratories, GRAHAM KOSIBA , Rensselaer Polytechnic Institute — Acquisition of experimental flight characteristics of electrically driven flyers (EDFs) is important in understanding the flyer's role in initiating detonator explosives. The velocity throughout a plastic flyer's flight was measured, as well as the magnitude and duration of the impulse while impacting an acrylic window. Despite the small size, thickness, and large accelerations of the EDFs, diagnostic techniques now have the temporal and spatial fidelity to measure validation-quality flyer characteristics. Using multipoint photonic Doppler velocimetry (PDV) in conjunction with streak imaging through a fiber array the velocity profile, bow shock (air cushion), time of impact, flyer shape at impact, and shock duration were measured. Shock physics simulations were then compared to this high fidelity data as a means of validating equations of state. Through the combination of experiments and simulations we can achieve a greater fundamental understanding of the energy transfer from the EDF to the energetic material prior to initiation.

¹Sandia National Labs is a multi-program lab 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 G26.00008 Analysis of Porous Media as Inlet Concept for Rotating Detonation Engines¹, KEVIN GROGAN, MATTHIAS IHME, Stanford University, DEPARTMENT OF MECHANICAL ENGINEERING TEAM — Rotating detonation engines combust reactive gas mixtures with a high-speed, annularly-propagating detonation wave, which provides many advantages including a stagnation pressure gain and a compact, lightweight design. However, the optimal design of the inlet to the combustion chamber inlet is a moot topic since improper design can significantly reduce detonability and increase pressure losses. The highly diffusive properties of porous media could make it an ideal material to prevent the flashback of the detonation wave and therefore, allow the inlet gas to be premixed. Motivated by this potential, this work employs simulation to evaluate the application of porous media to the inlet of a rotating detonation engine as a novel means to stabilize a detonation wave while reducing the pressure losses incurred by non-ideal mixing strategies.

¹Department of the Air Force

9:44AM G26.00009 NOx 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 (PDE). 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. Results to date show that NOx emissions are not a problem for the RDE due to the short residence times and the nature of the flow field. Furthermore, simulations show that the amount of NOx can be further reduced by tailoring the fluid dynamics within the RDE.

9:57AM G26.00010 A stabilization mechanism for the low-velocity gaseous detonations with losses, ASLAN KASIMOV, ALIOU SOW, King Abdullah Univ of Sci & Tech (KAUST), ROMAN SEMENKO, Novosibirsk State University, Russia — Using the reactive Euler equations, we investigate numerically the nonlinear stability of steady-state one-dimensional gaseous detonations in the presence of both momentum and heat losses. Our results point to a possible stabilization mechanism for the low-velocity detonations in such systems. The mechanism stems from the existence of a one-parameter family of steady-state solutions found in Semenko et al. *Shock Waves*, 26(2), 141-160, 2016.

Monday, November 21, 2016 8:00AM - 10:10AM – Session G27 Non-Newtonian Flows: Theory and Analysis E147-148 - Yuriko Renardy, Virginia Tech

8:00AM G27.00001 A viscoelastic model with thixotropic yield stress behavior for filament stretching¹, YURIKO RENARDY, HOLLY GRANT, Virginia Tech — The transient behavior of filament stretching is studied for a viscoelastic constitutive model that combines a Partially Extending strand Convection model with a Newtonian solvent. The vertical filament is fixed at the bottom and the top is pulled up and held. Gravity and surface tension are also included in the model though they are not the primary mechanisms in this study. An axisymmetric circular slender jet approximation is applied. An asymptotic analysis for the initial stages of evolution is performed for large relaxation time, so that an interplay of fast and slow time scales emerges, and gives a criterion for whether the fluid yields immediately or whether slow dynamics ensues, depending on elastic stresses, gravity and capillary stress.

¹ NSF-DMS-1311707

8:13AM G27.00002 ABSTRACT WITHDRAWN —

8:26AM G27.00003 Advection of nematic liquid crystals by chaotic flow, LENNON O NARAIKH, Univ Coll Dublin — Consideration is given to the effects of inhomogeneous shear flow (both regular and chaotic) on nematic liquid crystals in a planar two-dimensional geometry. The Landau-de Gennes equation coupled to an externally-prescribed flow field is the basis for the study: this is solved numerically in a periodic spatial domain. The focus is on a limiting case where the advection is passive, such that variations in the liquid-crystal properties do not feed back into the equation of motion for the fluid velocity. The numerical simulations demonstrate that the coarsening of the liquid-crystal domains is arrested by the flow. The nature of the arrest is different depending on whether the flow is regular or chaotic. For the specific case where tumbling is important, the flow has a strong effect on the the liquid-crystal morphology: this provides a mechanism for controlling the shape of the liquid-crystal domains.

8:39AM G27.00004 MOVED TO KP1.137 —

8:52AM G27.00005 Inertial migration of elastic particles in a pressure-driven power-law fluid¹, SAMUEL BOWIE, ALEXANDER ALEXEEV, Georgia Institute of Technology — Using three-dimensional computer simulations, we study the cross-stream migration of deformable particles in a channel filled with a non-Newtonian fluid driven by a pressure gradient. Our numerical approach integrates lattice Boltzmann method and lattice spring method in order to model fluid structural interactions of the elastic particle and the surrounding power fluid in the channel. The particles are modeled as elastic shells filled with a viscous fluid that are initially spherical. We focus on the regimes where the inertial effects cannot be neglected and cause cross-stream drift of particles. We probe the flow with different power law indexes including both the shear thickening and thinning fluids. We also examine migration of particles of with different elasticity and relative size. To isolate the non-Newtonian effects on particle migration, we compare the results with the inertial migration results found in the case where the channel is filled with a simple Newtonian fluid. The results can be useful for applications requiring high throughput separation, sorting, and focusing of both synthetic particles and biological cells in microfluidic devices.

¹Financial support provided by National Science Foundation (NSF) Grant No. CMMI1538161

9:05AM G27.00006 Viscous-elastic dynamics of power-law fluids within an elastic cylinder, AMIR GAT, EVGENIY BOYKO, MORAN BERCOVICI, Technion - Israel Institute of Technology — We study the fluid-structure interaction dynamics of non-Newtonian flow through a slender linearly elastic cylinder at the creeping flow regime. Specifically, considering power-law fluids and applying the thin shell approximation for the elastic cylinder, we obtain a non-homogeneous p-Laplacian equation governing the viscous-elastic dynamics. We obtain exact solutions for the pressure and deformation fields for various initial and boundary conditions, for both shear thinning and shear thickening fluids. In particular, impulse or a step in inlet pressure yield self-similar solutions, which exhibit a compactly supported propagation front solely for shear thinning fluids. Applying asymptotic expansions, we provide approximations for weakly non-Newtonian behavior showing good agreement with the exact solutions sufficiently far from the front.

9:18AM G27.00007 MOVED TO KP1.138 —

9:31AM G27.00008 Viscoplastic boundary layers, DUNCAN HEWITT, University of Cambridge, NEIL BALMFORTH, University of British Columbia, RICHARD CRASTER, Imperial College London — Viscoplastic fluids are characterized by a yield stress, below which they do not deform. If the yield stress is large, viscoplastic flows can develop narrow boundary layers that provide surfaces of failure between rigid or almost rigid regions, or between such regions and rigid boundaries. Oldroyd (1947) presented a theoretical discussion of these viscoplastic boundary layers, but they have been largely ignored since then, in part because of the complexity of the nonlinear boundary-layer equations. We revisit Oldroyd's analysis, and consider various examples of flow, including a jet-like intrusion, flow past a thin plate, and flow down channels with topography. By comparison with detailed numerical solutions, we verify Oldroyd's original theory, and also reveal its shortcomings. Where these exist, we present an alternative theory more akin to classical lubrication solutions. We also relate these viscoplastic flow solutions to slipline constructions in classical plasticity theory.

9:44AM G27.00009 Dynamics of two disks settling in a two-dimensional narrow channel: From periodic motion to vertical chain in Oldroyd-B fluid¹, TSORNG-WHAY PAN, ROLAND GLOWINSKI, Department of Mathematics, University of Houston, Houston, TX — In this talk we present a numerical study of the dynamics of two disks settling in a narrow vertical channel filled with an Oldroyd-B fluid. Two kinds of particle dynamics are obtained: (i) periodic interaction between two disks and (ii) the formation of the chain of two disks. For the periodic interaction of two disks, two different motions are obtained: (a) two disks stay far apart and interact periodically, which is similar to one of the motions of two disks settling in a narrow channel filled with a Newtonian fluid discussed by Aidun & Ding (Phys. Fluids, 15 (2003), 1612) and (b) two disks draft, kiss and break away periodically and the chain is not formed due to not strong enough elastic force. For the formation of two disk chain occurred at higher values of the elasticity number, it is either a tilted chain or a vertical chain. The tilted chain can be obtained for either that the elasticity number is less than the critical value for having the vertical chain or that the Mach number is greater than the critical value for a long body to fall broadside-on, which is consistent with the results for the elliptic particles settling in Oldroyd-B fluids.

¹ NSF

9:57AM G27.00010 Effect of the Convected Terms in the Transient Viscoelastic Flow, NARIMAN ASHRAFI, MEYSAM MOHAMADALI, Young Researchers and Elites Club, Science and Research Branch, Islamic Azad University, Tehran, Iran — Influence of fluid elasticity is examined for the plane Couette flow (PCF) of an improved Johnson Segalman (J.S) fluid through introduction of coefficients in the convected terms. The flow field is obtained from the conservation and constitutive equations using the Galerkin projection method. Effect of several values of governing parameters such as introduced coefficients, Reynolds number and Weissenberg number on velocity and normal and shear stresses profiles are explored. The results show that the oscillating behavior of velocity profile tends to grow as the coefficients increase. For higher Weissenberg, the oscillations are more intensive, whereas the amplitude of oscillation tends to reduce. This reveals that, the deviation decreases by increasing the coefficients. The amplitude of normal stress differences tend to grow as the coefficients of the convected terms grow, revealing more elastic behavior in the fluid. On the other hand; the effect of the convected terms on the steady behavior of normal stress difference is strongly dependent on the value of Weissenberg number. The shear stress behavior is also dependent on the coefficients of the convected terms and the flow properties, that is, for higher Reynolds the shear stress reaches a maximum and then decreases to minimum. For lower Reynolds, the opposite occurs.

Monday, November 21, 2016 8:00AM - 9:05AM –

Session G28 Particle-laden Flows: Experimental Techniques F149 - Sarah Hormozi, Ohio University

8:00AM G28.00001 X-ray Mapping of Dynamic Suspensions¹, MOHAMMAD GHOLAMI, Mechanical Eng. Dept., Ohio University, 251 Stocker , Athens, OH 45701, USA., NICOLAS LENOIR, GUILLAUME OVARLEZ, PLACAMAT, UMS3626-CNRS/University of Bordeaux, Pessac, 33608, France, SARAH HORMOZI, Mechanical Eng. Dept., Ohio University, 251 Stocker, Athens, OH 45701, USA. — Dense non-colloidal suspensions are materials with broad application both in industrial processes and natural phenomena. In most of these applications, the suspensions are either far from equilibrium or strongly non-Newtonian (i.e., non-colloidal particles are suspended in non-Newtonian fluid) meaning that the flow kinetics are not only strain-dependent but also strain-rate dependent. Therefore, experimental techniques must be developed to analyze the flows of these complex suspensions over a wide range of steady and transient shear rates. Techniques such as Nuclear Magnetic Resonance/Imaging (NMR/I) are inapplicable due to low sampling frequency and low image resolution[1] (typically 10 minutes per averaged NMR image of 1x1cm). We introduce a new technique using an X-ray/CT-scan system to study dynamic suspensions. We show our recent results on the application of this technique for the study of shear induced migration of particles in a yield stress matrix fluid in a wide-gap cylindrical Couette cell. This work opens new avenues to study dynamic non-colloidal suspensions and the suspensions with other types of nonlinear suspending fluids such as viscoelastic and shear thickening fluids. [1] Ovarlez,G., et al.,Journal of Rheology, Vol 50, 2006, pp. 259-292.

¹NFS(CBET-1554044-CAREER)

8:13AM G28.00002 Size segregated ring pattern formation in particle impactors , J.

R. SAYLOR, S. A. FREDERICKS, Clemson University — Typical particle impactors consist of a nozzle that directs a particle laden flow onto a plate, and is designed to capture particles greater than a cutoff diameter. Connected in series as a cascade, with each impactor designed to have a progressively smaller cutoff diameter, the particle size distribution can be measured. Typical impactors utilize a nozzle-to-plate distance S that is on the order of one nozzle diameter W , $S/W \sim 1$, and give a nominally Gaussian particle deposition pattern on the plate. We explored conditions where $S/W << 1$ and observed deposition patterns consisting of very fine rings. Moreover, we found that the ring diameter increased with decreasing particle diameter and the ring thickness increased with particle diameter. These results suggest a potential method for sizing particles by using the mature technology of impactors in a different way. Potential mechanisms for how these ring patterns are formed will be discussed. We note that prior studies have observed conditions where particle deposition patterns exhibited "halos". These halos appear less distinct than the rings we have observed, and it is unclear whether they are related.

8:26AM G28.00003 Development of a Magnetic Resonance Imaging-Based Method for Particle Concentration Measurement, DANIEL D. BORUP, CHRISTOPHER J. ELKINS, JOHN K. EATON, Stanford Univ — Magnetic Resonance Imaging (MRI) is well suited for the study of fluid mechanics in complex flows where optical access is not possible. Current MRI-based techniques allow for the measurement of 3D, 3-component velocity and scalar concentration fields. The current work aims to develop and validate a technique for measuring the concentration of a dispersed phase of solid microspheres in a turbulent water flow. Such a diagnostic would allow for the study of the transport of small particles in arbitrarily complicated biological, engineering, or natural flows. In the presence of paramagnetic particles, MRI signal decays more rapidly than it does for pure water due to small disturbances in the magnetic field. We predicted the spatial extent and magnitude of this disturbance using a standard theoretical framework for MRI and obtained reasonable agreement with experimental results. Using the linear relationship between particle volume fraction and signal decay rate, we also obtained 3D concentration data for a particle streak injected into a ribbed serpentine channel flow. These data were used to validate the new method, and the transport of solid particles was compared to the transport of a passive scalar in the same flow. Daniel Borup is supported by NSF Grant No. DGE-114747.

8:39AM G28.00004 Multi-camera PIV of two-phase oscillating sheet flow , CHANG LIU,

KEN KIGER, University of Maryland — We present a multi-camera thin light sheet imaging method to accurately measure dispersed phase concentration and velocity up to optical densities of close to 0[1]. The work is an extension of prior single camera methods that utilize particle image characteristics to identify the dispersed phase and infer the effective measurement volume thickness. By introducing multiple camera perspectives, stereo photogrammetry can be combined with the redundancy of information available in the images to provide 1) increased accuracy in determining individual particle locations, and 2) increased reliability in identifying all of the dispersed phase objects. As a byproduct, the velocity of all three components is also available. As an example, this new method is directly applied to oscillating sheet flow conditions. From a single image pair, individual particles are identified and tracked, giving the instantaneous volume concentration and dispersed phase velocity. A median filter method is used to isolate an image composed only of the much smaller tracer particles, and processed to generate a 3-component continuous phase velocity field. Given the concentration and velocities of the two phases, two-phase flow properties such as the sedimentation rate and momentum coupling will be reported.

8:52AM G28.00005 A simultaneous charge and size measurement method for individual airborne particles using digital holographic particle imaging¹, ADAM HAMMOND, ZHONGWANG DOU, ZACH LIANG, HUI MENG, University at Buffalo SUNY — Recently, significant inquiry to understand the effects of particle charge on particle laden flow have been made, particularly in the study of Lagrangian particle-pair statistics. Quantification of individual particle charge allows relation of inter-particle electric forces and turbulence-induced forces. Here we offer a simultaneous, individual particle charge and size measurement technique utilizing in-line digital holographic Particle Tracking Velocimetry (hPTV). The method measures particle electric mobility through its velocity response within a uniform electric field using a sequence of holograms, next the particle diameter is measured with the same holograms using a matched-filter developed by Lu et al. (2012) as an input for calculation of charge. Consequently, a benefit of this method is that particle charge is calculated on the individual level, versus a mean charge calculated from a group of particles, offering improved estimations of charge distributions for studies of particle laden flow.

¹This work was supported by NSF CBET-0967407 and CBET-0967349.

Monday, November 21, 2016 8:00AM - 10:10AM – Session G29 CFD: Large-eddy Simulation F150 - Karthik Duraisamy, University of Michigan

8:00AM G29.00001 Statistical Mechanics-based Closures for Large Eddy Simulations¹, ERIC PARISH, KARTHIK DURAISAMY, AYOUB GOUASMI, University of Michigan, COMPUTATIONAL AEROSCIENCES LABORATORY TEAM — The simulation of high Reynolds-number fluid flows is made challenging by the presence of an enormous range of temporal and spatial scales. The Mori-Zwanzig (MZ) formalism originates from non-equilibrium statistical mechanics and provides a formal backdrop for the construction of coarse-grained models. In this work, a class of models inspired from the Mori-Zwanzig formalism are applied to turbulent flows. The MZ-models are derived directly from the governing equations and require minimal heuristics. The resulting closures are non-Markovian and are akin to modeling the divergence of the sub-grid stress. Non-local temporal effects are captured through a finite memory approximation of the MZ memory kernel. Numerical simulations of rotating homogeneous turbulence and turbulent channel flow are presented. The MZ-based models are shown to accurately characterize the behavior of the unresolved dynamics associated energy transfer mechanisms.

¹This work was supported in part by AFOSR under the project "LES Modeling of Non-local effects using Statistical Coarse-graining" with Dr. Jean-Luc Cambier as the technical monitor.

8:13AM G29.00002 Unphysical scalar excursions in large-eddy simulations¹, GEORGIOS MATHEOU, Jet Propulsion Laboratory, California Institute of Technology, PAUL DIMOTAKIS, California Institute of Technology — The range of physically realizable values of passive scalar fields in any flow is bounded by their boundary values. The current investigation focuses on the local conservation of passive scalar concentration fields in turbulent flows and the ability of the large-eddy simulation (LES) method to observe the boundedness of passive scalar concentrations. In practice, as a result of numerical artifacts, this fundamental constraint is often violated with scalars exhibiting unphysical excursions. The present study characterizes passive-scalar excursions in LES of a turbulent shear flow and examines methods for error diagnosis. Typically, scalar-excursion errors are diagnosed as violations of global boundedness, i.e., detecting scalar-concentration values outside boundary/initial condition bounds. To quantify errors in mixed-fluid regions, a local scalar excursion error metric is defined with respect to the local non-diffusive limit. Analysis of such errors shows that unphysical scalar excursions in LES result from dispersive errors of the convection-term discretization where the subgrid-scale model (SGS) provides insufficient dissipation to produce a sufficiently smooth scalar field. Local scalar excursion errors are found not to be correlated with the local scalar-gradient magnitude.

¹This work is supported by AFOSR, DOE, and Caltech

8:26AM G29.00003 Wall-resolved large-eddy simulation of flow past a circular cylinder¹, W. CHENG, King Abdullah University of Science and Technology, D.I. PULLIN, California Institute of Technology, R. SAMTANEY, King Abdullah University of Science and Technology — Wall-resolved large-eddy simulations (LES) about a smooth-walled circular cylinder are described over a range of Reynolds number from $Re_D = 3.9 \times 10^3$ (subcritical) to above the drag crisis, $Re_D = 8.5 \times 10^5$ (supercritical), where D is the cylinder diameter. The span-wise domain is $3D$ for $Re_D \leq 10^5$ and D otherwise. The numerical method is a fourth-order finite-difference discretization on a standard curvilinear O-grid. The stretched-vortex sub-grid scale model is used in the whole domain, including regions of large-scale separated flow. For $Re_D \leq 10^5$, calculations of the skin-friction coefficient versus polar angle θ along the cylinder surface and its dependence on Re_D are well captured in comparison with experimental data. Proper separation behavior is observed. For high Re_D , a fine mesh $8192 \times 1024 \times 256$ is used. It is found that a blowing/suction-type perturbation of the wall-normal velocity along a span-wise strip, with angular position at $\theta = 50 - 60^\circ$, is then required in order to produce flow separation in accordance with experiment at Reynolds numbers in the drag-crisis regime. Results presented will focus on the skin-friction behavior and details of flow separation.

¹Supported partially by KAUST OCRM Award No. URF/1/1394-01 and partially by NSF award CBET 1235605. The Cray XC40, Shaheen, at KAUST was utilized for all simulations.

8:39AM G29.00004 Entropy-viscosity based LES of turbulent flow in a flexible pipe, ZHICHENG WANG, Massachusetts Institute Technology, FANGFANG XIE, Zhejiang University, MICHAEL TRIANTAFYLLOU, Massachusetts Institute Technology, YIANNIS CONSTANTINIDES, Chevron Energy Technology Company, GEORGE KARNIADAKIS, Brown University — We present large-eddy simulations (LES) of turbulent flow in a flexible pipe conveying incompressible fluid. We are interested in quantifying the flow-structure interaction in terms of mean quantities and their variances. For the LES, we employ an Entropy Viscosity Method (EVM), implemented in a spectral element code. In previous work, we investigated laminar flow and studied the complex interaction between structural and internal flow dynamics and obtained a phase diagram of the transition between states as function of three non-dimensional quantities: the fluid-tension parameter, the dimensionless fluid velocity, and the Reynolds number. Here we extend our studies in the turbulence regime, Re from 5,000 to 50,000. The motion of the flexible pipe affects greatly the turbulence statistics of the pipe flow, with substantial differences for free (self-sustained) vibrations and prescribed (forced) vibrations.

8:52AM G29.00005 A Stabilized Scale-Similarity Model for Explicitly-Filtered LES¹

, AYABOE EDOH, ANN KARAGOZIAN, UCLA, VENKATESWARAN SANKARAN, Air Force Research Laboratory — Accurate simulation of the filtered-scales in LES is affected by the competing presence of modeling and discretization errors². In order to properly assess modeling techniques, it is imperative to minimize the influence of the numerical scheme. The current investigation considers the inclusion of resolved and un-resolved sub-filter stress ([U]RSFS) components in the governing equations³, which is suggestive of a mixed-model approach. Taylor-series expansions of discrete filter stencils are used to inform proper scaling of a Scale-Similarity model representation of the RSFS term⁴, and accompanying stabilization is provided by tunable and scale-discriminант filter-based artificial dissipation techniques⁵ that represent the URFS term implicitly. Effective removal of numerical error from the LES solution is studied with respect to the 1D Burgers equation with synthetic turbulence, and extension to 3D Navier-Stokes system computations is motivated.

¹Distribution A: Approved for public release, distribution unlimited. Supported by AFOSR (PMs: Drs. Chiping Li and Michael Kendra)

²Ghosal, *J. Comp. Phys.*, 125, 187-206 (1996)

³Carati *et al.*, *J. Fluid. Mech.*, 441, 119-138 (2001)

⁴Pruett *et al.*, *Phys. of Fluids*, 13, 2578-2589 (2001)

⁵Edoh *et al.*, AIAA 2016-3794

9:05AM G29.00006 LES of a submarine model in self-propulsion¹ , ANTONIO POSA, ELIAS

BALARAS, The George Washington University — Wall-resolved LES computations are presented on the flow over a notional submarine geometry in self-propelled conditions at $Re=1.2e+06$ (based on the free-stream velocity and the length of the body). The rotational speed of the propeller was dynamically adjusted during the simulation using a proportional-integral controller, in order for the propeller thrust to balance the overall drag on the system. An immersed-boundary methodology was adopted to enforce boundary conditions over the body. Comparisons with the same submarine geometry in the towed configuration allowed us to verify that the boundary layer over the hull surface is affected only in the stern region, while that over the cylindrical mid-body is roughly in equilibrium in both towed and self-propelled conditions. The quasi-streamwise structures of the turbulent boundary layer and their development along the stern dominate the flow ingested by the propeller. The wake and junction flows from the fins break the axial symmetry of such flow. Comparisons with the towed configuration verify that the propeller affects substantially its own inflow, defined by the overlapping effects of adverse pressure gradient, due to the tapered geometry of the stern, and suction generated by the same propeller.

¹This work is supported by the Office of Naval Research, under Grant No. N000141110455, monitored by Dr. Ki-Han Kim.

9:18AM G29.00007 An investigation of slip wall boundary condition for wall-modeled large-eddy simulation¹ , HYUNJI JANE BAE, ADRIAN LOZANO-DURAN, SANJEEB BOSE, PARVIZ MOIN, Stanford Univ

— Wall models for large-eddy simulation are necessary to overcome the resolution requirements near the wall for high Reynolds number turbulent flows. In the present study, the slip wall boundary condition is examined (Bose and Moin, *Phys. Fluids*, 2014). The optimal slip length and its dependence on Reynolds number, grid size, subgrid scale model, etc. is investigated in turbulent channel flows up to $Re_\tau = 4200$. Two families of slip wall models are introduced. The first is derived from the Navier Stokes equations, and the second is based on error minimization in two different filter levels. These new and existing models are tested and compared to the optimal slip length and the filtered direct numerical simulation results.

¹Supported by NASA.

9:31AM G29.00008 Further reduction of near-wall resolution for wall-modeled LES¹ , ALEXANDRE MARQUES, QIQI WANG, Massachusetts Inst of Tech-MIT, JOHAN LARSSON, University of Maryland, GREGORY LASKOWSKI, GE Aviation, SANJEEB BOSE, Cascade Technologies — One of the greatest challenges to the use of Large Eddy Simulations (LES) in engineering applications is the large number of grid points required near walls. To mitigate this issue, LES is often coupled with a model of the flow close to the wall, known as wall model. One feature common to most wall models is that the first few (about 3) grid points must be located below the inviscid log-layer ($y/\delta \leq 0.2$), and the grid must have near isotropic resolution near the wall. Hence, wall-modeled LES may still require a large number of grid points, both in the wall-normal and span-wise directions. Because of these requirements, wall-modeled LES still is unfeasible in many applications. We present a new formulation of wall-modeled LES that is being developed to address this issue. In this formulation, LES is used to solve only for the features of the velocity field that can be adequately represented on the LES grid. The effects of the unresolved features are captured by imposing a balance of momentum integrated in the wall-normal direction. This integral momentum balance translates into a dynamic PDE defined on the walls, which is coupled to the LES equations. We discuss details of the new formulation and present results obtained in laminar and turbulent channel flows.

¹This work was partially supported by the Center of Turbulence Research at Stanford University, and by the U.S. Department of Energy under Award Number DE-SC-0011089

9:44AM G29.00009 A dummy cell immersed boundary method for incompressible turbulence simulations over dirty geometries , KEIJI ONISHI, RIKEN AICS, MAKOTO TSUBOKURA, Kobe

University / RIKEN AICS — A methodology to eliminate the manual work required for correcting the surface imperfections of computer-aided-design (CAD) data, will be proposed. Such a technique is indispensable for CFD analysis of industrial applications involving complex geometries. The CAD geometry is degenerated into cell-oriented values based on Cartesian grid. This enables the parallel pre-processing as well as the ability to handle 'dirty' CAD data that has gaps, overlaps, or sharp edges without necessitating any fixes. An arbitrary boundary representation is used with a dummy-cell technique based on immersed boundary (IB) method. To model the IB, a forcing term is directly imposed at arbitrary ghost cells by linear interpolation of the momentum. The mass conservation is satisfied in the approximate domain that covers fluid region except the wall including cells. Attempts to satisfy mass conservation in the wall containing cells leads to pressure oscillations near the IB. The consequence of this approximation will be discussed through fundamental study of an LES based channel flow simulation, and high Reynolds number flow around a sphere. And, an analysis comparing our results with wind tunnel experiments of flow around a full-vehicle geometry will also be presented.

9:57AM G29.00010 Wall-layer model for LES with massive separation¹ , AHMAD FAKHARI,

VINCENZO ARMENIO, University of Trieste - Trieste, FEDERICO ROMAN, IEFLUIDS s. r. l. - Trieste — Currently, Wall Functions (WF) work well under specific conditions, mostly exhibit drawbacks specially in flows with separation beyond curvatures. In this work, we propose a more general WF which works well in attached and detached flows, in presence and absence of Immersed Boundaries (IB). First we modified an equilibrium stress WF for boundary-fitted geometry making dynamic the computation of the k (von Karman constant) of the log-law; the model was first applied to a periodic open channel flow, and then to the flow over a 2D single hill using uniform coarse grids; the model captured separation with reasonable accuracy. Thereafter IB Method by Roman et al. (Phys. Fluids, 2009) was improved to avoid momentum loss at the interface between the fluid-solid regions. This required calibration of interfacial eddy viscosity; also a random stochastic forcing was used in wall-normal direction to increase Reynolds stresses and improve mean velocity profile. Finally, to reproduce flow separation, a simplified boundary layer equation was applied to construct velocity at near wall computational nodes. The new scheme was tested on the 2D single hill and periodic hills applying Cartesian and curvilinear grids; good agreement with references was obtained with reduction in cost and complexity.

¹Financial support from project COSMO "CFD open source per opera morta" PAR FSC 2007-2013, Friuli Venezia Giulia.

Monday, November 21, 2016 8:00AM - 10:10AM –

Session G30 Granular Flows: General F151 - Marie-Julie Dalbe, Massachusetts Institute of Technology

8:00AM G30.00001 Deformation of a 3D granular media caused by fluid invasion

, MARIE-JULIE DALBE, RUBEN JUANES, Massachusetts Institute of Technology — 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. Several experimental and computational studies have shown that the competition between capillary and friction forces can lead to different regimes of deformation, from frictional fingering to hydro-capillary fracturing (Sandnes et al., Nat. Comm. 2011, Holtzman et al., PRL 2012). Most of these investigations have focused, however, on 2D or quasi-2D systems. Here, we develop an experimental set-up that allows us to observe two-phase flow in a fully 3D granular bed and measure the fluid pressure while controlling the level of confining stress. We use an index matching technique to directly visualize the injection of a liquid in a granular media saturated with another, immiscible liquid. We extract the deformation the whole granular bulk as well as at the particle level. Our results show the existence of different regimes of invasion patterns depending on key dimensionless groups that control the system.

8:13AM G30.00002 Electrification and Charge Distribution in Vertically Shaken

Granular Media.¹, RUBEN ROJAS, FREJA NORDSIEK, DANIEL LATHROP, Univ of Maryland-College Park — Granular charging of particle laden flows at large scales is a widespread phenomenon and has long been observed in nature: Volcanic ash clouds, desert sandstorms, dust devils, thunderstorms and snowstorms all undergo electrification at large scale. As a first approach to understand this phenomenon, we confined granular particles to a vertically oscillating cylindrical chamber with top and bottom conducting plates. Long term voltage transients between the plates and a high dependence on the total particle surface area suggested the preponderance of collective effects in the electrification processes. In order to further explore this hypothesis, we reduced the electrode area for the measurement with two 2-cm circular flat probes on the top plate. With this setup we detected differences in the charge distribution among the particles due to a more localized measurement of the voltage.

¹This research was supported by the Julian Schwinger Foundation.

8:26AM G30.00003 Avalanches and local force evolution in a granular stick-slip experiment¹ , AGHIL ABED ZADEH, Duke University, JONATHAN BARES, Montpellier University, ROBERT BEHRINGER, Duke University — We perform a stick-slip experiment to characterize avalanches 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 spring. We study the PDF of energy release and slip size, avalanche shape in time, and other seismicity laws during slip avalanches. We analyze the power spectrum of the force signal and probability distributions 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 local stress change and flow of particles during avalanches. By image processing we detect the avalanches, as connected components in space and time, and the energy dissipation inside the granular medium and their PDFs. The PDFs of avalanches obey power laws both at global and local scales, but with different exponents. We try to understand the distribution and correlation of local avalanches in space and the way they coarse grain to the global avalanches.

¹NSF-DMR-1206351, NASA Grant NNX15AD38G, William M. Keck Foundation

8:39AM G30.00004 Vibrational Collapse of Hexapod Packings¹ , YUCHEN ZHAO, Department of

Physics, Duke University., JINGQIU DING, Department of Physics, Nanjing University, Nanjing, P.R.C., JONATHAN BARÉS, Laboratoire de Mécanique et Génie Civil, Université de Montpellier, Montpellier, France, KAROLA DIERICH, Institute for Computational Design, University of Stuttgart, Stuttgart, Germany , ROBERT BEHRINGER, Department of Physics, Duke University. — Columns made of convex noncohesive grains like sand collapse after being released from a confining container. However, structures built from concave grains can be stable without external support. Previous research show that the stability of the columns depends on column diameter and height, by observing column stability after carefully lifting their confinement tubes. Thinner and taller columns collapse with higher probability. While the column stability weakly depends on packing density, it strongly depends on inter-particle friction. Experiments that cause the column to collapse also reveal similar trends, as more effort (such as heavier loading or shearing) is required to destabilize columns that are intrinsically more stable. In the current experiments, we investigate the effect of vibration on destructing a column. Short columns collapse following the relaxation dynamics of disorder systems, which coincides with similar experiments on staple packings. However, tall columns collapse faster at the beginning, in addition to the relaxation process coming after. Using high-speed imaging, we analyze column collapse data from different column geometries. Ongoing work is focusing on characterizing the stability of hexapod packings to vibration.

¹We thanks NSF-DMR-1206351 and the William M. Keck Foundation

8:52AM G30.00005 Dynamic intermittency in discrete erodible-bed avalanches, MATTHEW ARRAN, NATHALIE VRIEND, DAMTP, University of Cambridge — The coexistence of fluid-like and solid-like behaviour in granular matter allows avalanches of grains to flow on the surface of a static but erodible bed. For sufficiently slow inflow, these avalanches are discrete, with previous experimentalists reporting that avalanche fronts pass a given point quasi-periodically. We report instead observations of dynamic intermittency between two regimes, one in which avalanches occur quasi-periodically and another in which the intervals between them are irregular. Finding the first regime consistent with existing models, we introduce a model for the second regime within the framework of Self-Organised Criticality, and describe the transition between the regimes with reference to the state of the erodible bed.

9:05AM G30.00006 Plowing of granular surface by a vertical blade, VACHITAR SINGH JUDGE, EMILIE DRESSAIRE, NYU Tandon School of Engineering, ALBAN SAURET, SVI (CNRS/Saint-Gobain) — The interaction of a blade with a layer of fluid is an important industrial problem involved in coating of substrates, removing of snow, plowing the fields for agriculture. Most experimental and theoretical work has focused on the drag and lift forces on plowing blade as it is dragged on a granular surface or studying the flow of the fluid while plowing. However the study of deformation of a granular surface by a vertical plow blade has received less attention, despite significant practical and fundamental implications. In this study, we investigate experimentally the behavior of a granular substrate as a vertical plow blade of finite width is translated at constant speed. The vertical blade directs the granular material outward and sand piles form on either sides. We characterize the dynamics of plowing by measuring and rationalizing the influence of the width of the vertical plow blade, the height of the plow blade, and the depth granular substrate.

9:18AM G30.00007 Characteristic correlation study of UV disinfection performance for ballast water treatment, TE BA, HONGYING LI, Institute of High Performance Computing, ASTAR, HAFIZ OSMAN, Research & Development, Sembcorp Marine Ltd, CHANG-WEI KANG, Institute of High Performance Computing, ASTAR — Characteristic correlation between ultraviolet disinfection performance and operating parameters, including ultraviolet transmittance (UVT), lamp power and water flow rate, was studied by numerical and experimental methods. A three-stage model was developed to simulate the fluid flow, UV radiation and the trajectories of microorganisms. Navier-Stokes equation with k-epsilon turbulence was solved to model the fluid flow, while discrete ordinates (DO) radiation model and discrete phase model (DPM) were used to introduce UV radiation and microorganisms trajectories into the model, respectively. The UV dose statistical distribution for the microorganisms was found to move to higher value with the increase of UVT and lamp power, but moves to lower value when the water flow rate increases. Further investigation shows that the fluence rate increases exponentially with UVT but linearly with the lamp power. The average and minimum resident time decreases linearly with the water flow rate while the maximum resident time decrease rapidly in a certain range. The current study can be used as a digital design and performance evaluation tool of the UV reactor for ballast water treatment.

9:31AM G30.00008 An Experiment Study about Forces and Shapes of Liquid bridges for Micron-particles, HUANG ZHANG, SHUIQING LI, Department of Thermal Engineering, Tsinghua University, Beijing, China, DEPARTMENT OF THERMAL ENGINEERING, TSINGHUA UNIVERSITY, BEIJING, CHINA TEAM — Wet micro-particles are often encountered in the electrostatic precipitator and fiber filtration system. The liquid bridges formed between wet granules are leading to change the flow or packing behavior of micro-particles. Firstly, the magnitude of liquid bridge force between a micron-particle and a plate is measured by atomic force microscope (AFM). The AFM tip is adopted as the sphere tip instead of the sharp one. As the humidity increases, the shape of liquid bridge between this particle and the plate is observed by confocal laser scanning microscopy (CLSM). Secondly, a suspension containing micro-particles are painting on a plate to form particles layers. As humidity increases, AFM is used to measure the forces between the sphere tip and the micro-particles in the layers. At the same time, the shape of liquid bridges is observed to see whether the bridges are formed as pendular, funicular and capillary ones by CLSM. Finally, it is found out that liquid bridges are hardly existed below a critical humidity, and the force between particles also grows slowly. Beyond this critical humidity, a sudden increase of the liquid bridge force is exhibited. Liquid forces are also compared between different shapes of liquid bridges.

9:44AM G30.00009 Effects of hydrodynamic interaction on random adhesive loose packings of micron-sized particles, WENWEI LIU, RAN TAO, SHENG CHEN, HUANG ZHANG, SHUIQING LI¹, Tsinghua Univ, KEY LABORATORY FOR THERMAL SCIENCE AND POWER ENGINEERING OF MINISTRY OF EDUCATION TEAM — Random loose packings of uniform spherical micron-sized particles under a uniform flow field are investigated via an adhesive discrete-element method with the two-way coupling of both the particles and the fluid. Characterized by a dimensionless adhesion parameter Ad , the packing fraction follows the similar law to that without fluid, but results in larger values due to the hydrodynamic compression. The total pressure drop through the packed bed increases with the packing fraction and agrees well with the theoretical predictions of Ergun function. The effects of different parameters, such as flow velocity, particle size and surface energy, on packing fraction and pressure drop take place through different ways, which can be associated with the local mechanical equilibrium in the presence of fluid.

¹Corresponding author

9:57AM G30.00010 Phase Transitions and Metastability in Self-Propelled Particle systems, AJINKYA KULKARNI, SUMESH THAMPI, MAHESH PANCHAGNULA, Indian Inst of Tech-Madras — Ordered motion of self-propelling micro-organisms produce interesting patterns. The objective of this study is to investigate the nature of the transition from disorganized thermal-like motion to organized vortical motion, and the resulting metastability in systems of self-propelled particles. A modified version of the Standard Vicsek Model has been used, where the particles are modeled as soft disks with finite mass, confined in a circular domain. We observe multiple phases as the local co-ordination coefficient is varied. We analyze the nature of transitions by calculating Binder Cumulants of the order parameters. An occurrence of metastability is investigated in the hysteretic region. The switching between the steady states of the system in the hysteretic region has been triggered via artificial nucleation of randomly picked particles spanning the entire domain. In addition, the effect of domain size on the nature of the phase transitions has been studied. Finally the motivation for these phase transitions has been explained via thrust generation ability and the geometry of the confinement.

Monday, November 21, 2016 8:00AM - 9:44AM – Session G31 Experimental Techniques - PIV-based Pressure and Wind Tunnel F152 - Joseph Katz, Johns Hopkins University

8:00AM G31.00001 Error Propagation dynamics of PIV-based pressure calculation 2: from Poisson equations to Kirchhoff plates, ZHAO PAN, JARED WHITEHEAD, TADD TRUSCOTT, Utah State University — Little research has been done to investigate the dynamics of error propagation from PIV-based velocity measurements to the pressure calculation. Rather than measure experimental error, we analytically investigate error propagation by examining the properties of the Poisson equation directly. Our results provide two contributions to the PIV community. First, we quantify the error bound in the pressure field by illustrating the mathematical roots of why and how PIV-based pressure calculations propagate. Second, we design the “worst case error” for a pressure Poisson solver. In other words, we provide a systematic example where the relatively small errors in the experimental data can lead to maximum error in the corresponding pressure calculations. The 2D calculation of the worst case error surprisingly leads to the classic Kirchhoff plates problem, and connects the PIV-based pressure calculation, which is a typical fluid problem, to elastic dynamics. The results can be used for optimizing experimental error minimization by avoiding worst case scenarios. More importantly, they can be used to design synthetic velocity error for future PIV-pressure challenges, which can be the hardest test case in the examinations.

8:13AM G31.00002 Quantification and correction of the error due to limited PIV resolution on the accuracy of non-intrusive spatial pressure measurement using a DNS channel flow database, XIAOFENG LIU, SETH SIDDLE-MITCHELL, San Diego State University — The effect of the subgrid-scale (SGS) stress due to limited PIV resolution on pressure measurement accuracy is quantified using data from a direct numerical simulation database of turbulent channel flow (JHTDB). A series of 2000 consecutive realizations of sample block data with 512x512x49 grid nodal points were selected and spatially filtered with a coarse 17x17x17 and a fine 5x5x5 box averaging, respectively, giving rise to corresponding PIV resolutions of roughly 62.6 and 18.4 times of the viscous length scale. Comparison of the reconstructed pressure at different levels of pressure gradient approximation with the filtered pressure shows that the neglect of the viscous term leads to a small but noticeable change in the reconstructed pressure, especially in regions near the channel walls. As a contrast, the neglect of the SGS stress results in a more significant increase in both the bias and the random errors, indicating the SGS term must be accounted for in PIV pressure measurement. Correction using similarity SGS modeling reduces the random error due to the omission of SGS stress from 114.5% of the filtered pressure r.m.s. fluctuation to 89.1% for the coarse PIV resolution, and from 66.5% to 35.9% for the fine PIV resolution, respectively, confirming the benefit of the error compensation method and the positive influence of increasing PIV resolution on pressure measurement accuracy improvement.

8:26AM G31.00003 Instantaneous Pressure Field Calculation from PIV Data with Least-Square Reconstruction, JIACHENG ZHANG, CARLO SCALO, PAVLOS VLACHOS, Purdue Univ — A method using least-square reconstruction of instantaneous pressure fields from PIV velocity measurements is introduced and applied to both planar and volumetric flow data. Pressure gradients are computed on a staggered grid from flow acceleration. An overdetermined system of linear equations which relates the pressure and the computed pressure gradients is formulated. The pressure field is estimated as the least-square solution of the overdetermined system. The flow acceleration is approximated by the vortex-in-cell procedure, providing the pressure field from a single velocity snapshot. The least-square method is compared against the omni-directional pressure gradient integration and solving the pressure Poisson equation. The results demonstrate that the omni-directional integration and the least-square method are more robust to the noise in velocity measurements than the pressure Poisson solver. In addition, the computational cost of the least square method is much lower than the omni-directional integration, and very easily extendable to volumetric data retaining computational efficiency. The least-square method maintains higher accuracy than the pressure Poisson equation while retaining a similar computational burden.

8:39AM G31.00004 GPU-based, parallel-line, omni-directional integration of measured acceleration field to obtain the 3D pressure distribution.¹, JIN WANG, CAO ZHANG, JOSEPH KATZ, JHU — A PIV based method to reconstruct the volumetric pressure field by direct integration of the 3D material acceleration directions has been developed. Extending the 2D virtual-boundary omni-directional method (Omni2D, Liu & Katz, 2013), the new 3D parallel-line omni-directional method (Omni3D) integrates the material acceleration along parallel lines aligned in multiple directions. Their angles are set by a spherical virtual grid. The integration is parallelized on a Tesla K40c GPU, which reduced the computing time from three hours to one minute for a single realization. To validate its performance, this method is utilized to calculate the 3D pressure fields in isotropic turbulence and channel flow using the JHU DNS Databases (<http://turbulence.pha.jhu.edu>). Both integration of the DNS acceleration as well as acceleration from synthetic 3D particles are tested. Results are compared to other method, e.g. solution to the Pressure Poisson Equation (e.g. PPE, Ghaemi et al. 2012) with Bernoulli based Dirichlet boundary conditions, and the Omni2D method. The error in Omni3D prediction is uniformly low, and its sensitivity to acceleration errors is local. It agrees with the PPE/Bernoulli prediction away from the Dirichlet boundary. The Omni3D method is also applied to experimental data obtained using tomographic PIV, and results are correlated with deformation of a compliant wall.

¹ONR

8:52AM G31.00005 Weighted least-squares solver for determining pressure from particle image velocimetry data¹, ROELAND DE KAT, University of Southampton — Currently, most approaches to determine pressure from particle image velocimetry data are Poisson approaches (e.g. de Kat & van Oudheusden, 2012, *Exp. Fluids* 52:1089–1106) or multi-pass marching approaches (e.g. Liu & Katz, 2006, *Exp. Fluids* 41:227–240). However, these approaches deal with boundary conditions in their specific ways which cannot easily be changed—Poisson approaches enforce boundary conditions strongly, whereas multi-pass marching approaches enforce them weakly. Under certain conditions (depending on the certainty of the data or availability of reference data along the boundary) both types of boundary condition enforcement have to be used together to obtain the best result. In addition, neither of the approaches takes the certainty of the particle image velocimetry data (see e.g. Sciacchitano et al., 2015, *Meas. Sci. Technol.* 26:074004) within the domain into account. Therefore, to address these shortcomings and improve upon current approaches, a new approach is proposed using weighted least-squares. The performance of this new approach is tested on synthetic and experimental particle image velocimetry data. Preliminary results show that a significant improvement can be made in determining pressure fields using the new approach.

¹RdK is supported by a Leverhulme Trust Early Career Fellowship

9:05AM G31.00006 On Using Shaped Honeycombs for Experimental Generation of Arbitrary Velocity Profiles in Test Facilities¹, ALIREZA SAFARIPOUR, DAVID OLSON, AHMED NAGUIB, MANOOCHEHR KOOCHESFAHANI, Michigan State University — It is common to use a uniform approach flow in the study of most problems in aerodynamics. Motivated by situations where the approach flow is not uniform, the focus of the current work is on the experimental generation of arbitrary velocity profiles in a flow facility (water tunnel) using the shaped honeycomb technique originally proposed by Kotansky (1966). Employing further refinement of this approach, multiple honeycomb devices are designed and fabricated to produce prescribed velocity profiles. The performance of these devices is assessed in terms of their agreement with the desired velocity profiles and the level of turbulence they produce. Single-component molecular tagging velocimetry (1c-MTV) is used to characterize the resulting mean and fluctuating streamwise velocity profiles and their streamwise development. The shaped honeycomb technique is shown to be effective in producing the desired velocity profiles with high fidelity while maintaining velocity fluctuations level at or below that of the freestream prior to installation of the devices.

¹This work was supported by AFOSR award number FA9550-15-1-0224.

9:18AM G31.00007 Use of 3D Printing for Custom Wind Tunnel Fabrication, PAUL GAGORIK, ZACHARY BATES, EMIN ISSAKHANIAN, Loyola Marymount University — Small-scale wind tunnels for the most part are fairly simple to produce with standard building equipment. However, the intricate bell housing and inlet shape of an Eiffel type wind tunnel, as well as the transition from diffuser to fan in a rectangular tunnel can present design and construction obstacles. With the help of 3D printing, these shapes can be custom designed in CAD models and printed in the lab at very low cost. The undergraduate team at Loyola Marymount University has built a custom benchtop tunnel for gas turbine film cooling experiments. 3D printing is combined with conventional construction methods to build the tunnel. 3D printing is also used to build the custom tunnel floor and interchangeable experimental pieces for various experimental shapes. This simple and low-cost tunnel is a custom solution for specific engineering experiments for gas turbine technology research.

9:31AM G31.00008 ABSTRACT WITHDRAWN —

Monday, November 21, 2016 8:00AM - 10:10AM —
Session G32 Turbulent Boundary Layers: High Reynolds Numbers and Pressure Gradient Effects Oregon Ballroom 201 - Michel Stanislas, Ecole Centrale de Lille

8:00AM G32.00001 Large Scale Organization of a Near Wall Turbulent Boundary Layer¹, MICHEL STANISLAS, RAOUL FLORENT DEKOU TIOMAJOU, JEAN MARC FOUCAUT, Ecole Centrale de Lille — This study lies in the context of large scale coherent structures investigation in a near wall turbulent boundary layer. An experimental database at high Reynolds numbers ($Re_\theta = 9830$ and $Re_\theta = 19660$) was obtained in the LML wind tunnel with stereo-PIV at 4 Hz and hot wire anemometry at 30 kHz [1]. A Linear Stochastic Estimation procedure, is used to reconstruct a 3 component field resolved in space and time. Algorithms were developed to extract coherent structures from the reconstructed field. A sample of 3D view of the structures is depicted in Figure 1. Uniform momentum regions are characterized with their mean hydraulic diameter in the YZ plane, their life time and their contribution to Reynolds stresses. The vortical motions are characterized by their position, radius, circulation and vorticity in addition to their life time and their number computed at a fixed position from the wall. The spatial organization of the structures was investigated through a correlation of their respective indicative functions in the spanwise direction. The simplified large scale model that arises is compared to the ones available in the literature. Streamwise low (green) and high (yellow) uniform momentum regions with positive (red) and negative (blue) vortical motions. **REFERENCES** Joel Delville, Patrick Braud, Sébastien Coudert, Jean-Marc Foucaut, Carine Fourment, WK George, Peter BV Johansson, Jim Kostas, Fahrid Mehdi, and Royer. The wallturb joined experiment to assess the large scale structures in a high Reynolds number turbulent boundary layer. In Progress in Wall Turbulence: Understanding and Modeling, pages 65–73. Springer, 2011.

¹This work was supported by Campus International pour la Sécurité et l'Intermodalité des Transports

8:13AM G32.00002 Turbulent Poiseuille & Couette flows at high Re ¹, MYOUNGKYU LEE, ROBERT D. MOSER, The University of Texas at Austin — We present the results of direct numerical simulation (DNS) of high Re turbulent Poiseuille and Couette flows. Couette flow has been simulated with a streamwise (x) domain that is $100\pi\delta$ long at Reynolds number up to $Re_T \approx 500$. In addition Poiseuille flow simulations up to $Re_T \approx 5200$ were performed (Lee & Moser, *J. Fluid Mech.*, **774**, 2015). In Couette flow, extremely large scale motions, which are approximately $50\pi\delta$ long in the x -direction with very strong intensity, have been observed. In this presentation we will focus on a comparison between these two flows in terms of the vorticity-velocity co-spectra, which are interesting because of the relationship between the Reynolds stress and the velocity-vorticity correlation ($\partial_y \langle u'v' \rangle = \langle w'y'_z \rangle - \langle v'w'_z \rangle$). Also considered will be the spectra of the turbulent transport term in the evolution equation for the turbulent kinetic energy. In both (co)-spectra it is shown that the difference between the two flows at high Re are primarily at large scales.

¹This 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.

8:26AM G32.00003 Evidence of an asymptotic geometric structure to the Reynolds stress motions in turbulent boundary layers¹, JOSEPH KLEWICKI, University of New Hampshire/Melbourne, JIMMY PHILIP, CALEB MORRILL-WINTER, University of Melbourne — Recent results suggest that the uv motions in turbulent wall-flows asymptotically exhibit self-similar geometric properties. Herein we use time series from high resolution boundary layer experiments up to high Reynolds numbers to discern additional properties associated with the uv signals. Their space filling properties are shown to reinforce previous observations, while the uv skewness profile suggests that the size and magnitude of these motions are correlated on the inertial domain. The size and length scales of the negative uv -motions are shown to increase with distance from the wall, while their occurrences decreases. A joint analysis of the signal magnitudes and their corresponding lengths reveals that the length scales that contribute most to $\langle -uv \rangle$ are distinctly larger than their average size. The u and v cospectra, however, exhibit invariance across the inertial region when their wavelengths are normalized by the width distribution, $W(y)$, of the scaling layer hierarchy surmised from analysis of the mean momentum equation. This distribution is associated with scale dependent zero-crossings in the contributions to $\langle -uv \rangle$, and derivative cospectra of $\langle -uv \rangle$ support the existence of this structural detail.

¹This work is supported by the Australian Research Council and the National Science Foundation

8:39AM G32.00004 Time resolved, near wall PIV measurements in a high Reynolds number turbulent pipe flow, C. WILLERT, German Aerospace Center (DLR), Germany, J. SORIA, Monash University, Australia, M. STANISLAS, Ecole Centrale de Lille, France, O. AMILI, Monash University, G. BELLANI, University of Bologna, Italy, C. CUVIER, Ecole Centrale de Lille, M. EISFELDER, Monash University, T. FIORINI, University of Bologna, N. GRAF, InnoLas GmbH, Germany, J. KLINNER, German Aerospace Center (DLR) — We report on near wall measurements of a turbulent pipe flow at shear Reynolds numbers up to $Re_\tau = 40000$ acquired in the CICLOPE facility near Bologna, Italy. With 900 mm diameter and 110 m length the facility offers a well-established turbulent flow with viscous length scales ranging from $y^+ = 85\mu\text{m}$ at $Re_\tau = 5000$ to $y^+ = 11\mu\text{m}$ at $Re_\tau = 40000$. These length scales can be resolved with a high-speed PIV camera at image magnification near unity. For the measurement the light of a high-speed, double-pulse laser is focused into a $\approx 300\mu\text{m}$ thin light sheet that is introduced radially into the pipe. The light scattered by $1\mu\text{m}$ water-glycerol droplet seeding is observed from the side by the camera via a thin high-aspect ratio mirror with a field of view covering 20mm in wall-normal and 5mm in stream-wise direction. Statistically converged velocity profiles could be achieved using 70000 samples per sequence acquired at low laser repetition rates (100Hz). Higher sampling rates of 10 kHz provide temporally coherent data from which frequency spectra can be derived. Preliminary analysis of the data shows a well resolved inner peak that grows with increasing Reynolds number. (Project funding through EuHIT - www.euhit.org)

8:52AM G32.00005 Two-dimensional energy spectra in a high Reynolds number turbulent boundary layer¹, DILEEP CHANDRAN, RIO BAIDYA, JASON MONTY, IVAN MARUSIC, Univ of Melbourne —

The current study measures the two-dimensional (2D) spectra of streamwise velocity component (u) in a high Reynolds number turbulent boundary layer for the first time. A 2D spectra shows the contribution of streamwise (λ_x) and spanwise (λ_y) length scales to the streamwise variance at a given wall height (z). 2D spectra could be a better tool to analyse spectral scaling laws as it is devoid of energy aliasing errors that could be present in one-dimensional spectra. A novel method is used to calculate the 2D spectra from the 2D correlation of u which is obtained by measuring velocity time series at various spanwise locations using hot-wire anemometry. At low Reynolds number, the shape of the 2D spectra at a constant energy level shows $\lambda_y \sim \sqrt{z\lambda_x}$ behaviour at larger scales which is in agreement with the literature. However, at high Reynolds number, it is observed that the square-root relationship gradually transforms into a linear relationship ($\lambda_y \sim \lambda_x$) which could be caused by the large packets of eddies whose length grows proportionately to the growth of its width. Additionally, we will show that this linear relationship observed at high Reynolds number is consistent with attached eddy predictions.

¹The authors gratefully acknowledge the support from the Australian Research Council

9:05AM G32.00006 Structure Functions in Wall-bounded Flows at High Reynolds Number

XIANG YANG, Johns Hopkins University, IVAN MARUSIC, University of Melbourne, PERRY JOHNSON, CHARLES MEN-EVEAU, Johns Hopkins University — The scaling of the structure function $D_{ij} = \langle (u_i(x+r)-u_i(x))(u_j(x+r)-u_j(x)) \rangle$ (where $i=1,2,3$ and r is the two-point displacement, u_i is the velocity fluctuation in the x_i direction), is studied in wall-bounded flows at high Reynolds number within the framework of the Townsend attached eddy model. While the scaling of D_{ij} has been the subject of several studies, previous work focused on the scaling of D_{11} for $r=(\Delta x, 0, 0)$ (for streamwise velocity component and displacements only in the streamwise direction). Using the Hierarchical-Random-Additive formalism, a recently developed attached-eddy formalism, we propose closed-form formulae for the structure function D_{ij} with two-point displacements in arbitrary directions, focusing on the log region. The work highlights new scalings that have received little attention, e.g. the scaling of D_{ij} for $r=(0, \Delta y, \Delta z)$ and for $i \neq j$. As the knowledge on D_{ij} leads directly to that of the Reynolds stress, statistics of the filtered flow field, etc., an analytical formula of D_{ij} for arbitrary r can be quite useful for developing physics-based models for wall-bounded flows and validating existing LES and reduced order models.

9:18AM G32.00007 DNS of self-similar adverse pressure gradient turbulent boundary layer¹, JULIO SORIA, VASSILI KITSIOS, ATSUSHI SEKIMOTO, CALLUM ATKINSON, Monash University, JAVIER JIMÉNEZ, Universidad Politécnica de Madrid — A direct numerical simulation (DNS) of a self-similar adverse pressure gradient (APG) turbulent boundary layer (TBL) at the verge of separation has been set-up and carried out. The DNS APG TBL has a displacement thickness based Reynolds number that ranges up to 30,000. The conditions for self-similarity and appropriate scaling will be highlighted, with the first and second order velocity statistical profiles non-dimensionalised using this scaling. The details of the DNS and the required boundary conditions that are necessary to establish this self-similar APG-TBL will be presented. The statistical properties of the self-similar adverse pressure gradient (APG) turbulent boundary layer (TBL) DNS will be presented, as will the profiles of the terms in the momentum equation, spanwise/wall-normal kinetic energy spectrum and two-point correlations, which will be compared to those of a zero pressure gradient turbulent boundary layer.

¹NCI and Pawsey SCC funded by the Australian and Western Australian governments as well as the support of PRACE funded by the European Union are gratefully acknowledged.

9:31AM G32.00008 Experimental Measurements of a High Reynolds Number Adverse Pressure Gradient Turbulent Boundary Layer

CALLUM ATKINSON, Monash University, OMID AMILI, Monash University and University of Minnesota, MICHEL STANISLAS, CHRISTOPHE CUVIER, JEAN-MARC FOUCAUT, SRICHARAN SRINATH, JEAN-PHILIPPE LAVAL, Laboratoire de Mécanique de Lille, France, CHRISTIAN KAEHLER, RAINER HAIN, SVEN SCHARNOWSKI, Bundeswehr University Munich, Germany, ANDREAS SCHROEDER, REINHARD GEISLER, JANOS AGOCS, ANNI ROESE, German Aerospace Centre (DLR), Goettingen, Germany, CHRISTIAN WILLERT, JOACHIM KLINNER, German Aerospace Centre (DLR), Koeln, Germany, JULIO SORIA, Monash University, King Abdulaziz University, Saudi Arabia — The study of adverse pressure gradient turbulent boundary layers is complicated by the need to characterise both the local pressure gradient and its upstream flow history. It is therefore necessary to measure a significant streamwise domain at a resolution sufficient to resolve the small scales features. To achieve this collaborative particle image velocimetry (PIV) measurements were performed in the large boundary layer wind-tunnel at the Laboratoire de Mécanique de Lille, including: planar measurements spanning a streamwise domain of 3.5m using 16 cameras covering 15δ ; spanwise wall-normal stereo-PIV measurements, high-speed micro-PIV of the near wall region and wall shear stress; and streamwise wall-normal PIV in the viscous sub layer. Details of the measurements and preliminary results will be presented.

9:44AM G32.00009 DNS study of wall-pressure fluctuations in a turbulent boundary layer with large pressure gradients, HIROYUKI ABE, YASUHIRO MIZOBUCHI, YUICHI MATSUO, Japan Aerospace Exploration Agency — DNS data are used to examine the behavior of wall-pressure fluctuations p_w in a turbulent boundary layer with large adverse and favorable pressure gradients, thus involving separation and reattachment. The Reynolds number Re_θ based on momentum thickness is equal to 300, 600 and 900. Comparison is also made with recent experiment by Weiss et al. (2015) for $Re_\theta = 5000$. Particular attention is given to the scaling law focusing on maximum value of each Reynolds stress. It is shown that rms value of p_w normalized by dynamic pressure is about twice larger near separation and reattachment than for zero-pressure gradient. In the former regions, p_w is affected noticeably by outer-layer pressure fluctuations where low pressure regions are closely associated with large-scale motions of negative streamwise velocity fluctuations. Among the scalings, rms value of p_w normalized by maximum Reynolds shear stress (Simpson et al. 1987; Na & Moin 1998) leads to near plateau in adverse pressure gradient and separated regions, indicating that shear stress makes significant contribution to p_w . This scaling also holds reasonably near reattachment where the scaling with maximum wall-normal Reynolds stress, pointed out by Ji & Wang (2012) for steps, yields better collapse.

9:57AM G32.00010 The laminarescent region in the quasi-laminarization process: a vorticity dynamics perspective¹, GUILLERMO ARAYA, DANIEL RODRIGUEZ, CARLOS QUINONES, University of Puerto Rico Mayaguez — Incompressible turbulent boundary layers subject to severe acceleration or strong Favorable Pressure Gradient (FPG) might experience a quasi-laminarization or a reversion process characterized by a meaningful depression of Reynolds shear stresses and reduction of turbulent production attributed to the dominance of pressure forces. Direct Numerical Simulation (DNS) of highly accelerated turbulent boundary layers is performed in order to shed some light on the energy redistribution, transport phenomena and vorticity dynamics of the laminarescent stage during the quasi-laminarization process. This region is one of the constituents of The Island of Ignorance according to Sreenivasan [Acta Mech. 44, pp1-48, 1982]. In a recent article by Araya, Castillo and Hussain [Journal of Fluid Mechanics, 775, pp189 - 200, 2015], DNS of an initially fully turbulent flow subjected to a very strong FPG has shown reduction of the Reynolds shear stresses with a logarithmic behavior in the mesolayer region, associated with the trend of the wall-normal advection in that region, i.e. $V^+ \partial U^+ / \partial y^+ \sim 1/y^+$ ($\sim \partial \bar{u}' v' / \partial y^+$).

¹NSF-CBET 1512393

Monday, November 21, 2016 8:00AM - 9:57AM –
Session G33 Turbulent Boundary Layers: Walls and Modeling Oregon Ballroom 202 - Joseph Klewicki, University of New Hampshire

8:00AM G33.00001 Extending the restricted nonlinear model for wall-turbulence to high Reynolds numbers¹, JOEL BRETHEIM, CHARLES MENEVEAU, DENNICE GAYME, Johns Hopkins University — The restricted nonlinear (RNL) model for wall-turbulence is motivated by the long-observed streamwise-coherent structures that play an important role in these flows. The RNL equations, derived by restricting the convective term in the Navier-Stokes equations, provide a computationally efficient approach due to fewer degrees of freedom in the underlying dynamics. Recent simulations of the RNL system have been conducted for turbulent channel flows at low Reynolds numbers (Re), yielding insights into the dynamical mechanisms and statistics of wall-turbulence. Despite the computational advantages of the RNL system, simulations at high Re remain out-of-reach. We present a new Large Eddy Simulation (LES) framework for the RNL system, enabling its use in engineering applications at high Re such as turbulent flows through wind farms. Initial results demonstrate that, as observed at moderate Re, restricting the range of streamwise varying structures present in the simulation (i.e., limiting the band of x Fourier components or k_x modes) significantly affects the accuracy of the statistics. Our results show that only a few well-chosen k_x modes lead to RNL turbulence with accurate statistics, including the mean profile and the well-known inner and outer peaks in energy spectra.

¹This work is supported by NSF (WindInspire OISE-1243482)

8:13AM G33.00002 A Nonlinear System Model of Wall Turbulence Generation Under Active Suppression and Enhancement of Streak Transient Growth Instability¹, SAMARESH MIDYA, ALAN DUONG, FLINT THOMAS, THOMAS CORKE, University of Notre Dame — Schoppa and Hussain (1998, 2002) demonstrated streak transient growth (STG) as the dominant streamwise coherent structure generation mechanism required for wall turbulence production. A novel, flush surface-mounted pulsed-DC plasma actuator was recently developed at the University of Notre Dame to actively intervene in STG. In recent high Reynolds number, zero pressure gradient turbulent boundary layer experiments, drag reduction of up to 68% was achieved. This is due to a plasma-induced near-wall, spanwise mean flow sufficient in magnitude to prevent the lift-up of low-speed streaks. This limits their flanking wall-normal component vorticity—a critical parameter in STG. Experiments also show that sufficiently large plasma-induced spanwise flow can exacerbate STG and increase drag by 80%. The ability to significantly increase or decrease drag by near-wall actuation provides an unprecedented new tool for clarifying the open questions regarding the interaction between near-wall coherent structures and those in the logarithmic region. In the reported experiments this interaction is experimentally characterized by a second-order Volterra nonlinear system model under both active suppression and enhancement of STG.

¹Supported by NASA Langley under NNX16CL27C

8:26AM G33.00003 Comparison two different LES closure models of the transitional boundary layer flow, DIMITRY IVANOV, ANDREI CHORNY, A.V. Luikov Heat and Mass Transfer Institute, Minsk, Belarus — The goal of the present research is to measure the velocity profile in the thin boundary layer of a flat plate at zero angle of attack. We consider a flow over a flat plate with a uniform velocity profile. The uniform velocity fluid hits the leading edge of the flat plate, and a laminar boundary layer begins to develop. The near-wall, subgrid-scale (SGS) model is used to perform Large Eddy Simulation (LES) of the incompressible developing, smooth-wall, flat-plate turbulent boundary layer. In this model, the stretched-vortex, SGS closure is utilized in conjunction with a tailored, near-wall model designed to incorporate anisotropic vorticity scales in the presence of the wall. 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. LES solver using Smagorinsky and the One-equation LES turbulence models. Results show that the normalized mean velocity profile is in good agreement with the universal law-of-the-wall and previous published data. In order to ensure the quality of the numerical results a convergence study was performed.

8:39AM G33.00004 Application of Wall-modeled LES to Turbulent Separated Flows

, PRAHLADH S. IYER, National Institute of Aerospace, GEORGE I. PARK, Center for Turbulence Research, Stanford University, MUJEEB R. MALIK, NASA Langley Research Center — Resolved Large-Eddy Simulations (LES) and Direct Numerical Simulations (DNS) are unaffordable for very high Reynolds number (Re) wall-bounded flows. While the Reynolds Averaged Navier-Stokes (RANS) based methods predict high Re attached flows accurately with little cost, their fidelity is degraded significantly in flows involving separation. A popular compromise between cost and accuracy is to use a Wall-modeled LES (WMLES) approach. In WMLES, the outer portion of the boundary layer is resolved with LES while the inner portion is modeled. In order to assess the performance of the widely used wall-stress models in separated flows, we perform WMLES simulations using an unstructured, compressible finite volume LES solver. The equilibrium and non-equilibrium wall models that require the solution of the simplified/full RANS on a separate near-wall domain are employed. Two configurations are studied: the shock-induced separation in a transonic flow over an axisymmetric bump placed on a cylinder, and a low-Mach flow past a NACA 4412 airfoil at a near-stall condition. Detailed comparisons will be made with available experimental data to comment on the applicability of WMLES in predicting complex turbulent flows involving separation.

8:52AM G33.00005 Wall-modeled large-eddy simulation of transonic airfoil buffet at high Reynolds number¹

, YUMA FUKUSHIMA, SOSHI KAWAI, Department of Aerospace Engineering, Tohoku University — In this study, we conduct the wall-modeled large-eddy simulation (LES) of transonic buffet phenomena over the OAT15A supercritical airfoil at high Reynolds number. The transonic airfoil buffet involves shock-turbulent boundary layer interactions and shock vibration associated with the flow separation downstream of the shock wave. The wall-modeled LES developed by Kawai and Larsson PoF (2012) is tuned on the K supercomputer for high-fidelity simulation. We first show the capability of the present wall-modeled LES on the transonic airfoil buffet phenomena and then investigate the detailed flow physics of unsteadiness of shock waves and separated boundary layer interaction phenomena. We also focus on the sustaining mechanism of the buffet phenomena, including the source of the pressure waves propagated from the trailing edge and the interactions between the shock wave and the generated sound waves.

¹This work was supported in part by MEXT as a social and scientific priority issue to be tackled by using post-K computer. Computer resources of the K computer was provided by the RIKEN Advanced Institute for Computational Science (Project ID: hp150254).

9:05AM G33.00006 Prediction of wall shear-stress fluctuations in wall-modeled large-eddy simulation¹

, GEORGE PARK, MICHAEL HOWLAND, ADRIAN LOZANO-DURAN, PARVIZ MOIN, Center for Turbulence Research, Stanford University — Wall-modeled large-eddy simulation (WMLES) is emerging as a viable and affordable tool for predicting mean flow statistics in high Reynolds number turbulent boundary layers. Recently, we examined the performance of two RANS-based wall models in prediction of wall pressure and shear stress fluctuations which are important in flow/structure interaction problems. Whereas the pressure statistics were predicted with reasonable accuracy, the magnitude of wall shear stress fluctuations was severely underestimated (Park & Moin, *Phys. Rev. Fluids* 1, 024404 (2016)). The present study expands on this finding to characterize in more detail the capabilities of wall models for predicting τ'_w . Predictions of several wall models in high Reynolds number channel flows ($Re_\tau = 2000$) will be presented. Additionally, a recent empirical inner-outer model for τ'_w (Mathis *et al.*, *J. Fluid Mech.* 715:163–180 (2013)) is reconstructed using channel flow DNS database, and it is coupled to WMLES to assess its performance as a predictive model in LES.

¹The majority of this work was carried out during the 16th biannual Center for Turbulence Research (CTR) summer program, 2016. George Park was partially supported through NASA under the Subsonic Fixed-Wing Program (Grant No. NNX11AI60A)

9:18AM G33.00007 A unified description of spatial and spectral distribution of fluctuation intensities in wall turbulence

, YONG JI, ZHEN-SU SHE, SKLTCS, COE, Peking Univ. — The streamwise turbulent intensity in wall turbulence (pipe and boundary layer) presents non-uniform distribution in both physical and wave number space. The well-known Townsend-Perry attached eddy hypothesis divides the energy spectrum into three distinct ranges: a constant range at small wavenumbers $k < k_c$, a k^{-1} law in the "attached eddy" range $k < k_i$ and the Kolmogorov form $k^{-5/3}$ in the inertial range $k < k_d$. However, the latest boundary layer experiment (Vallikivi *et al.*, *J. Fluid Mech.*, vol. 771, 2015, pp. 303-326) indicates that a more precise spectral model is needed. We present here a unified analytical expression, based on a generalized dilation-invariant ansatz. It will be shown that analytic description of a stress length ell giving rise to accurate description of the mean velocity profile yields equally accurate prediction of the integral scale wavenumber k_i , and the predicted dissipation gives rise of good prediction of the Kolmogorov dissipation wavenumber k_d . Finally, the large-scale characteristic wavenumber k_c follows a simple scaling law in terms of the stress length ell . Furthermore, we find that the Princeton data reveals possible anomalous scaling in the k^{-1} and $k^{-5/3}$ range. The spectral curves based on our generalized dilation-invariant ansatz agree very well with the experimental spectrum, and the kinetic energy profile is also accurately reproduced. We have thus achieved, for the first time, a unified description of spatial and spectral distribution of fluctuation intensity from a recently developed symmetry approach.

9:31AM G33.00008 Low-complexity stochastic modeling of spatially evolving flows

, ARMIN ZARE, WEI RAN, University of Minnesota, M. J. PHILIPP HACK, Center for Turbulence Research, Stanford University, MIHAJLO JOVANOVIC, University of Minnesota — Low-complexity approximations of the Navier-Stokes (NS) equations are commonly used for analysis and control of turbulent flows. In particular, stochastically-forced linearized models have been successfully employed to capture structural and statistical features observed in experiments and direct simulations. In this work, we utilize stochastically-forced linearized NS equations and their parabolized equivalents to study the dynamics of flow fluctuations in transitional and turbulent boundary layers. We exploit the streamwise causality of the parabolized model to efficiently propagate statistics of stochastic disturbances into statistics of velocity fluctuations. Our study provides insight into interactions of slowly-varying base flow with streamwise streaks, oblique modes, and Tollmien-Schlichting waves. It also offers a systematic, computationally efficient framework for quantifying the influence of stochastic excitation sources (e.g., free-stream turbulence and surface roughness) on velocity fluctuations in weakly non-parallel flows.

9:44AM G33.00009 A simple model of inertial layer dynamics in turbulent boundary layers¹, JUAN CUEVAS, ALIREZA EBADI, CHRISTOPHER WHITE, GREGORY CHINI, JOSEPH KLEWICKI, University of New Hampshire — Observations (e.g., Meinhart & Adrian, *Phys. Fluids.*, **7**, 694 (1995)) indicate that the inertial region of turbulent wall-flows consists of uniform momentum zones segregated by narrow vortical fissures. Multiscale analysis similarly reveals that the mean momentum equation admits a scaling layer hierarchy across the inertial region. Here, each layer increases in width with wall-normal distance, but the inner-normalized velocity increment remains fixed. The talk reports on a simple model that captures the essential elements of these observations and the theoretical scalings. In this model, the number of fissures is specified to satisfy the average total velocity increment across the inertial layer, while the average wall-normal locations of the fissures and their widths are informed by the theory. Ensembles of statistically independent instantaneous velocity profiles are then created by simply allowing the fissures to randomly displace in the wall normal direction. Results indicate that the model identically recovers a logarithmic mean profile, produces a logarithmic decay in the streamwise velocity variance, and generates sub-Gaussian behaviours in its skewness and kurtosis profiles on the inertial domain. These findings along with possible refinements are also discussed.

¹This research was partially supported by the National Science Foundation and partially supported by the Australian Research Council

Monday, November 21, 2016 8:00AM - 9:57AM – Session G34 Turbulence: Modeling via Parameterization and Analysis Oregon Ballroom 203 - Robert Moser, University of Texas at Austin

8:00AM G34.00001 Hybrid LES/RANS simulation of a turbulent boundary layer over a rectangular cavity¹, QI ZHANG, SIGFRIED HAERING, TODD OLIVER, ROBERT MOSER, University of Texas at Austin — We report numerical investigations of a turbulent boundary layer over a rectangular cavity using a new hybrid RANS/LES model [1] and the traditional Detached Eddy Simulation (DES). Our new hybrid method aims to address many of the shortcomings from the traditional DES. In the new method, RANS/LES blending controlled by a parameter that measures the ratio of the modeled subgrid kinetic energy to an estimate of the subgrid energy based on the resolved scales. The result is a hybrid method automatically resolves as much turbulence as can be supported by the grid and transitions appropriately from RANS to LES without the need for ad hoc delaying functions that are often required for DES. Further, the new model is designed to improve upon DES by accounting for the effects of grid anisotropy and inhomogeneity in the LES region. We present comparisons of the flow features inside the cavity and the pressure time history and spectra as computed using the new hybrid model and DES.

¹[1] S. Haering. Anisotropic hybrid turbulence modeling with specific application to the simulation of pulse-actuated dynamic stall control. PhD thesis, University of Texas at Austin, 2015.

8:13AM G34.00002 The Use of the Reynolds Stress Transport Equation to Constrain Eigenvectors Perturbations on Model Form UQ of RANS Simulations, LUIZ SAMPAIO, Stanford Univ, RONEY THOMPSON, Universidade Federal do Rio de Janeiro, WOUTER EDELING, AASHWIN MISHRA, GIANLUCA IACCARINO, Stanford Univ — Despite the recent developments in LES and DNS approaches for turbulent flow simulations, RANS modeling is still vastly used by industry, due to its inherent low cost. Since accuracy is a concern in RANS modeling, model-form UQ is an essential tool for assessing the impacts of this uncertainty on quantities of interest. Bounding values for the eigenvalues of the dimensionless deviatoric part of the Reynolds Stress tensor (RST) can be obtained from realizability constraints, and therefore can be used as a first step towards a general perturbation approach. In this connection, decoupling the perturbation into an intensity (kinetic energy), a shape (eigenvalues), and an orientation (eigenvectors) parts constitutes a natural methodology to evaluate the model form UQ associated to the RST modeling. In this work, we show that ignoring eigenvectors perturbations can lead to significant impacts on the results from the UQ analysis. Besides that, we use the RST Equation as a constraint to impose some consistency between eigenvectors and eigenvalues perturbations, where the latter can be obtained from a more standard technique. We applied this methodology on the convex channel flow, and show the benefits of including the eigenvectors perturbations predicted by this methodology.

8:26AM G34.00003 A dynamic hybrid subgrid-scale modeling framework for large eddy simulations, ROMIT MAULIK, OMER SAN, Oklahoma State University - Stillwater — We put forth a dynamic modeling framework for sub-grid parameterization of large eddy simulation of turbulent flows based upon the use of the approximate deconvolution (AD) procedure to compute the eddy viscosity constant self-adaptively from the resolved flow quantities. In our proposed framework, the test filtering process of the standard dynamic model is replaced by the AD procedure and a posteriori error analysis is performed. The robustness of the model has been tested considering the Burgers, Kraichnan, Kolmogorov turbulence problems. Our numerical assessments for solving these canonical decaying turbulence problems show that the proposed approach could be used as a viable tool to address the turbulence closure problem due to its flexibility.

8:39AM G34.00004 Axisymmetric Afterbody Test Case for CFD Validation, KEVIN DISOTELL¹, Flow Physics and Control Branch, NASA Langley Research Center, CHRISTOPHER RUMSEY, Computational AeroSciences Branch, NASA Langley Research Center — As simulation complexity increases, the corresponding need for systematic, high-fidelity validation data sets continues to be important to advance physics-based CFD models. To this end, a parametric body of revolution is proposed as an experimental platform to support a wide validation domain for turbulent boundary layers outside the current bounds of DNS. Recognizing the challenges of detailed flow exploration on complex 3-D geometries, an analytically-defined body of revolution is pursued as a tractable, state-of-the-art measurement case for complex turbulent flows having extra rates of strain. The central feature of the concept based upon work by Presz Jr. & Pitkin [*J. Aircraft* **11**, 677 (1974)] is an interchangeable afterbody which can be tailored to distort a turbulent boundary layer in various ways, with incoming properties controlled by the forebody. An introduction to the test case design and overview of recent progress focused on smooth-body, turbulent separation physics are presented.

¹Supported by appointment to NASA Postdoctoral Program, administered by Universities Space Research Association.

8:52AM G34.00005 Nonlinear forcing in the resolvent analysis of wall-turbulence¹

, KEVIN ROSENBERG, California Institute of Technology, ADRIAN LOZANO DURAN, AARON TOWNE, Center for Turbulence Research, BEVERLEY MCKEON, California Institute of Technology — The resolvent analysis of McKeon and Sharma (JFM, 2010) formulates the Navier-Stokes equations as an input/output system in which the nonlinearity is treated as a forcing that acts upon the linear dynamics to yield a velocity response across wavenumber/frequency space. DNS data for a low Reynolds number turbulent channel ($Re_\tau = 180$) is used to investigate the structure of the nonlinear forcing directly. Specifically, we explore the spatio-temporal scales where the forcing is active and analyze its interplay with the linear amplification mechanisms present in the resolvent operator. This work could provide insight into self-sustaining processes in wall-turbulence and inform the modeling of scale interactions in large eddy simulations.

¹We gratefully acknowledge Stanford's Center for Turbulence Research for support of this work

9:05AM G34.00006 An anisotropic subgrid stress model for high aspect ratio grids

, ROBERT MOSER, SIGFRIED HAERING, Institute for Computational Engineering and Sciences, The University of Texas at Austin — Standard algebraic eddy viscosity subgrid stress models are formulated based on scalar measures of the local grid, and implicitly assume that the resolution is isotropic. However, complex simulation domains and computational costs associated with problems of engineering interest often necessitate grids with high aspect ratio cells. We present an anisotropic extension of Metias and Lesieur's structure function subgrid stress model [1]. Unlike existing algebraic SGS models, this model is constructed directly through the composition of resolution and resolved turbulence anisotropy. Comparisons with filtered DNS of forced isotropic homogeneous turbulence show the model to significantly outperform general isotropic SGS models with increasing resolution anisotropy.

[1] O. Metias and M. Lesieur, "Spectral large-eddy simulation of isotropic and stably stratified turbulence," Journal of Fluid Mechanics, vol. 239, 1992

9:18AM G34.00007 Scaling analysis on filtered near wall turbulence , PRAKASH MOHAN,

ROBERT MOSER, Univ of Texas, Austin — Large Eddy Simulations (LES) directly represent larger scale turbulent motions and model the effects of small scale motions. However in the near wall region the large dynamically important eddies scale in viscous wall units, which makes resolving them in a high Reynolds number LES very expensive. This motivates the use of wall-modeled LES, in which these near-wall eddies are modeled. To aid in the development of new wall models, we pursue an asymptotic analysis of the filtered Navier-Stokes equations, in the limit in which the horizontal filter scale is large compared to the thickness of the wall layer. It will be shown that in this limit the filtered velocities \bar{u} and subgrid stresses $\bar{\tau}$ in the near-wall layer are determined to zeroth order by filtered velocities at the boundary of the wall layer. Further the asymptotics suggest that there is a scaled universal velocity profile f and subgrid stress profile g in the near-wall region. The validity of this result will be tested and the profiles f and g will be evaluated through analysis of DNS data from channel flow at $Re_\tau = 5200$.

9:31AM G34.00008 Modeling and simulation of turbulence not at statistical equilibrium , ARASH NOURI GHEIMASSI, PEYMAN GIVI, WILLIAM LAYTON, Univ of Pittsburgh — Standard eddy viscosity models, while popular, cannot represent backscatter and have severe difficulties with complex turbulence not at statistical equilibrium. In this work, we give a derivation of eddy viscosity models from an equation for the evolution of variance in a turbulent flow showing how to correct eddy viscosity models. We show the corrected models preserve important features of the true Reynolds stresses and give algorithms for their discretization including a minimally invasive modular step to adapt an eddy viscosity computational methodology to the extended models. A numerical test is given with the usual and over diffusive Smagorinsky model. The consistency and the predictive capability of the model are established by conducting LES of a three-dimensional compressible mixing layer, and comparison with DNS data. The correction does exhibit intermittent backscatter.

9:44AM G34.00009 Analytical study on the SGS force around an elliptic Burgers vortex¹ , HIROMICHI KOBAYASHI, Keio Univ — The subgrid-scale (SGS) force around an elliptic Burgers vortex is analytically examined. In turbulence, there are a lot of vortex-tubes whose cross sections are known to be approximated as the ellipse. In this study, the biaxial elliptic Burgers vortex is produced by adding the compressive and extensional background straining flow to the conventional Burgers vortex. By using a filtering operation, we revealed that the energy transfer by the Reynolds stress term applying the Bardina model exhibits negative correlation to that by the true SGS stress term. However, it has been recently reported that a combination of the Bardina Reynolds term and the eddy viscosity model gives good performance even for the coarse LES of turbulent channel flows. In order to understand that, we discuss some SGS forces: by the true SGS stress tensor, by the eddy viscosity model, by the modified Leonard term and by the Bardina Reynolds term.

¹This work was supported by JSPS KAKENHI Grant Number 26420122.

Monday, November 21, 2016 8:00AM - 10:10AM –

Session G35 Turbulence: Wakes Oregon Ballroom 204 - Nicholas Hamilton, Portland State University

8:00AM G35.00001 Anisotropic character of low-order turbulent flow descriptions with the proper orthogonal decomposition , NICHOLAS HAMILTON, Portland State University, MURAT TUTKUN, Institute for Energy Technology, RAL BAYON CAL, Portland State University — Proper orthogonal decomposition (POD) is applied to data from distinct sets to characterize the error arising in the description of turbulence. Wind turbine wake experiment data and direct numerical simulation data from a fully developed channel flow are used to illustrate dependence of the anisotropy tensor invariants on the modes used in low-order descriptions. Reduced order flow descriptions via truncated POD bases greatly exaggerate turbulence anisotropy and can lead to a loss of three-dimensionality in extreme cases. Simple corrections to the low-order descriptions significantly reduces the errors. Similar gains are seen in the anisotropy tensor invariants. Corrections of this form reintroduces three-dimensionality to severely truncated POD bases. A threshold for truncating the POD basis based on the equivalent anisotropy factor for each measurement set requires many more modes than a threshold based on energy. The mode requirement to reach the anisotropy threshold after correction is reduced by an order of magnitude for all example data sets, ensuring that economical low-dimensional models in terms of modes included account for the isotropic quality of the turbulence field.

8:13AM G35.00002 A Predictive Model for Wind Farms Using Dynamic Mode Decomposition¹, VAUGHAN THOMAS, CHARLES MENEVEAU, DENNICE GAYME, Johns Hopkins University — In this work we extend traditional dynamic mode decomposition (DMD) to develop a linear predictive model for the time evolution of the velocity field for a multiple-turbine wind farm. Traditional DMD identifies a set of DMD modes which can be used to produce a linear system that approximates the dynamics of the original system. Typically, these DMD modes consist of those that both grow and decay, but in order to develop predictive model we need a system that evolves along a manifold that neither grows nor decays. Here we modify the DMD calculation to build such a model. We then apply this method to three dimensional large eddy simulations (LES) of a multi-turbine wind farm. Our predictive wind farm model is initialized with a small time series of data independent of the original data used to create the system. When initialized in this manner our DMD based model can reproduce the subsequent time evolution of the velocity field over ten inter-turbine convective timescales with a gradual falloff in performance.

¹This work is supported by the National Science Foundation (grants ECCS-1230788 and OISE-1243482, the WINDINSPIRE project).

8:26AM G35.00003 Low dimensional representations of side-by-side cylinders in cross-flow subject to varying freestream turbulence, ELIZABETH CAMP, RAÚL BAYOÁN CAL, Portland State University — Particle image velocity is employed to capture the near and intermediate wakes of pairs of side-by-side cylinders in cross-flow with varying levels of incoming freestream turbulence. Four sets of inflow conditions are each applied to three different transverse cylinder-to-cylinder spacing values. The center-to-center distance between the cylinders as well as the turbulence of the inflow heavily impact the mean velocity components as well as the Reynolds stresses. Proper orthogonal decomposition is used to further characterize the influence of freestream turbulence of the inflow on the wakes of each set cylinder transverse spacing.

8:39AM G35.00004 Experimental investigation of axially aligned flow past spinning cylinders.¹, PASQUALE CARLUCCI, LIAM BUCKLEY, IGBAL MEHMEDAGIC, DONALD CARLUCCI, U. S. Army ARDEC, Picatinny Arsenal, NJ, SIVA THANGAM, Stevens Institute of Technology, Castle Point, Hoboken, NJ — Experimental and numerical results of ongoing subsonic investigations of the flow field about axially aligned spinning cylinders with variable inter-cylinder spacing are presented. The experimental design is capable of investigating wake dynamics of the modeled system up to a Reynolds Number of 300,000 and rotation numbers up to 2. The experimental results are used to validate and confirm numerical simulations with and without the effects of swirl. The focus of the overall effort is an understanding of the dynamics of multi-body problems in a flow field, as such we relate the ongoing effort to previous studies by both the authors and the community at large and our ongoing work in developing accurate plant models for use in engineering analysis and design.

¹Funded in part by U. S. Army ARDEC, Picatinny Arsenal, NJ

8:52AM G35.00005 The Effect of Flow Curvature on the Axisymmetric Wake¹, MARLIN HOLMES, JONATHAN NAUGHTON, University of Wyoming — The swirling turbulent wake is a perturbation to the canonical axisymmetric turbulent wake. Past studies of the axisymmetric turbulent wake have increased understanding of wake Reynolds number influence on wake characteristics such as centerline wake velocity deficit and wake width. In comparison, the axisymmetric turbulent swirling wake has received little attention. Earlier work by our group has shown that the addition of swirl can change the characteristics of the wake. The goal of this current work is to examine how wake mean flow quantities are related to the wake Reynolds number and the swirl number, where the latter quantity is the ratio of the angular momentum flux to the axial momentum deficit flux. A custom designed swirling wake generator is used in a low turbulence intensity wind tunnel flow to study the turbulent swirling wake in isolation. Stereoscopic Particle Image Velocimetry is used to obtain three component velocity fields in the axial-radial plane. From this data, the wake Reynolds number, the swirl number, centerline velocity decay, wake width, and other relevant wake mean flow quantities are determined. Using these results, the impact of swirl on wake development is discussed.

¹This work was supported by the U.S. Department of Energy, Office of Science, Basic Energy Sciences, under Award DE-SC0012671.

9:05AM G35.00006 Hydrodynamic mechanism behind the suppression of vortex-induced vibration with permeable meshes¹, GUSTAVO R S ASSI, MURILO M CICOLIN, CESAR M FREIRE, University of Sao Paulo — Vortex-induced vibration (VIV) induces resonant vibrations on elastic bluff bodies when exposed to a flow. A VIV suppressor called "ventilated trousers" (VT) – consisting of a flexible net with tens of bobbins fitted every other node – has been developed as a commercial solution. Only a few experiments in the literature have evaluated the effectiveness of the VT, but very little is known about the underlying mechanism behind the suppression. Experiments have been carried out in a water channel with models of circular cylinders fitted with three different permeable meshes. VIV response and drag were obtained for models free to oscillate in the cross-flow direction with low mass and damping ($Re = 5,000$ to $25,000$). All meshes achieved an average 50% reduction of the peak amplitude and reduced the mean drag when compared to that of a bare cylinder. PIV visualization of the wake revealed that the VT produced a much longer vortex-formation length, thus explaining its enhanced efficiency in suppressing VIV and reducing drag. The geometry and distribution of the bobbins proved to be important parameters. PIV also revealed the rich three-dimensional flow structures created by the bobbins that disrupt the formation of a coherent vortex wake.

¹FAPESP 11/00205-6, 14/50279-4; CNPq 306917/2015-7

9:18AM G35.00007 Vortex dynamics in the near-wake of tabs with various geometries using 2D and 3D PIV., AXY PAGAN-VAZQUEZ, US Army Construction Engineering Research Laboratory, DOLAANA KHOVALYG, University of Illinois at Urbana-Champaign, CHARLES MARSH, US Army Construction Engineering Research Laboratory, ALI M. HAMED, LEONARDO P. CHAMORRO, University of Illinois at Urbana-Champaign — The vortex dynamics and turbulence statistics in the near-wake of rectangular, trapezoidal, triangular, and ellipsoidal tabs were studied in a refractive-index-matching channel at $Re = 2000$ and 13000 , based on the tab height. The tabs share the same bulk dimensions including a 17 mm height, a 28 mm base width, and a 24.5° angle. 3D PIV was used to study the mean flow and dominant large-scale vortices, while high-spatial resolution planar PIV was used to quantify high-order statistics. The results show the coexistence of counter-rotating vortex pair (CVP) and hairpin structures. These vortices exhibit distinctive topology and strength across Re and tab geometry. The CVP is a steady structure that grows in strength over a significantly longer distance at the low Re due to the lower turbulence levels and the delayed shedding of the hairpin vortices. These features at the low Re are associated with the presence of K-H instability that develops over three tab heights. The interaction between the hairpins and CVP is measured in 3D for the first time and shows complex coexistence. Although the CVP suffers deformation and splitting at times, it maintains its presence and leads to significant spanwise and wall-normal flows.

9:31AM G35.00008 Characterizing cycle-to-cycle variations of the shedding cycle in the turbulent wake of a normal flat plate using generalized phase averages.¹, ROBERT MARTINUZZI, University of Calgary — Quasi-periodic vortex shedding in the turbulent wake of a thin-flat plate placed normal to a uniform stream at Reynolds number of 6700 is investigated based on Particle Image Velocimetry experiments. The wake structure and vortex formation are characterized using a generalized phase average (GPA), a refinement of the triple decomposition of Reynolds and Hussain (1970) incorporating elements of mean-field theory (Stuart, 1958). The resulting analysis highlights the importance of cycle-to-cycle variations in characterizing vortex formation, wake topology and the residual turbulent Reynolds Stresses. For example, it is shown that during high-amplitude cycles vorticity is strongly concentrated within the well-organized shed vortices, whereas during low-amplitude cycles the shed vortices are highly distorted resulting in significant modulation of the shedding frequency. It is found that high-amplitude cycles contribute more to the coherent Reynolds stress field while the low-amplitude cycles contribute to the residual stress field. It is further shown that traditional phase-averaging techniques lead to an over-estimation of the residual stress field.

¹Natural Sciences and Engineering Research Council of Canada

9:44AM G35.00009 Coherent structures and enstrophy dynamics in highly stratified flow past a sphere at $Re = 3700$, KARU CHONGSIRIPINYO, ANIKESH PAL, SUTANU SARKAR, Univ of California - San Diego — Vortex dynamics of flow past a sphere in a linearly stratified environment is investigated. Simulations are carried out for a flow with Reynolds number of 3700 and for several Froude numbers (Fr) ranging as low as 0.025. Isosurface of Q criterion is used to identify vortical structures whose cross-section and orientation are found to be affected by buoyancy. At low $Fr = 0.025$, pancake eddies and surfboard-like inclined structures emerge in the near wake and have a regular streamwise spacing that is associated with the frequency of vortex shedding from the sphere. Similar to turbulent kinetic energy, the enstrophy in the near wake decreases with decreasing Fr (increasing stratification) until a minimum at $Fr = 0.5$ but the trend reverses in the low- Fr regime. Vortex stretching by fluctuating and mean strain are both responsible for enhancing vorticity with relatively small contribution from the baroclinic term. Decreasing Fr to $O(1)$ values tends to suppress vortex stretching. Upon further reduction of Fr below 0.25, the vortex stretching term takes large values near the sphere.

9:57AM G35.00010 Dynamics of flow over a sphere at moderate Re in a highly stratified fluid, ANIKESH PAL, SUTANU SARKAR, University Of California San Diego — Direct numerical simulations (DNS) are performed to investigate the flow past a sphere at $Re = 3700$ and $Fr \in [0.025, 1]$. Unlike previous experimental and numerical studies of flow over a sphere at low Re and low Fr , it is found that the fluctuations tend to regenerate at Fr lower than a critical value for moderate $Re = 3700$. High stratification suppress vertical motion and, for a three-dimensional body, the fluid flows horizontally around the sides leading to a new regime of unsteady vortex shedding. Vertically thin layers of shear interspersed between quasi-two dimensional motions undergo secondary Kelvin-Helmholtz (KH) instabilities if the buoyancy Reynolds number, $Re_b \geq O(1)$. The combined effect of unsteady vortex shedding, enhanced horizontal shear, and secondary KH instabilities results in the regeneration of turbulence at low Fr . There is an *increase* in the coefficient of drag C_d at high stratification (low Fr), for $Re = 3700$. This result is contrary to previous experiments on flow over sphere at low Re where C_d was found to *decrease* with increase in stratification in the low- Fr regime.

Monday, November 21, 2016 8:00AM - 10:10AM – Session G36 Drops: Surface Wetting Portland Ballroom 251 - Paul Steen, Cornell University

8:00AM G36.00001 Measuring contact-line mobility during inertial spreading¹, PAUL STEEN, SUSAN DANIEL, YI XIA, Cornell University — During "inertial spreading", when inertia drives a partially wetting liquid across a solid, the role of bulk viscosity may be neglected. For such inertial-capillary motions, behavior of the moving contact-line (CL) can be understood within the context of ideal (or nearly ideal) fluid motion, provided an alternate to the Voinov-Hocking-Cox model of mobility is adopted. The alternate we adopt is the so-called Hocking condition. In this talk, we report experiments with Resonantly-Driven Droplets (RDD) whereby the bulk resonance of the drop amplifies the small and fast CL motion sufficiently to be measurable. The RDD approach enables us to measure a CL mobility and to infer a CL dissipation for droplets on a number of hydrophobic surfaces, surfaces with varying contact-angle hysteresis. Our results are compared to prior results in the literature, measured with alternative approaches.

¹National Science Foundation grant no. CBET-1236582

8:13AM G36.00002 Effects of elasticity and surface tension on the spreading dynamics of a thin film under the influence of intermolecular forces., YUAN-NAN YOUNG, Department of Mathematical Sciences, NJIT, HOWARD STONE, Princeton University — The spreading dynamics of a thin layer of viscous Newtonian fluid between an elastic sheet and a wetting solid substrate is examined using the lubrication theory. On the wetting substrate an ultra thin film (precursor film) develops as a result of the intermolecular force between the fluid and the wetting solid substrate. Such a precursor film prevents the stress singularity associated with a moving contact line. Following the methodology by \citet{Glasner2003_PoF}, the effects of elasticity on the macroscopic contact line structure in the quasistatic limit are elucidated by an ordinary differential equation derived from an analysis of the energy and its dissipation. Similar to the case of a regular fluid interface with surface tension (capillary spreading), the elasto-capillary thin film profile also consists of a core at the center, an ultra thin film in the far field, and a contact line region where the core film profile connects smoothly to the precursor film. For capillary spreading, the precursor film transitions monotonically to the core film. Due to the interfacial elasticity, a spatial oscillation of film height in the contact line region is found. In addition, it is found that elasticity causes the sliding motion of the thin film: the contact angle close to zero as

8:26AM G36.00003 How a Nanodroplet Diffuses on Smooth Surfaces.¹, CHU LI, Department of Mechanical and Aerospace Engineering, The Hong Kong University of Science and Technology, Clear Water Bay, Kowloon, Hong Kong, JIZU HUANG, Institute of Computational Mathematics and Scientific/Engineering Computing, Academy of Mathematics and Systems Science, Chinese Academy of Sciences, ZHIGANG LI, Department of Mechanical and Aerospace Engineering, The Hong Kong University of Science and Technology, Clear Water Bay, Kowloon, Hong Kong — In this study, we investigate how nanodroplets diffuse on smooth surfaces through molecular dynamics (MD) simulations and theoretical analyses. The simulations results show that the surface diffusion of nanodroplet is different from that of single molecules and solid nanoparticles. The dependence of nanodroplet diffusion coefficient on temperature is surface wettability dependent, which undergoes a transition from linear to nonlinear as the surface wettability is weakened due to the coupling of temperature and surface energy. We also develop a simple relation for the diffusion coefficient by using the contact angle and contact radius of the droplet. It works well for different surface wettabilities and sized nanodroplets, as confirmed by MD simulations.

¹This work was supported by the Research Grants Council of the Hong Kong Special Administrative Region under Grant No. 615312.

8:39AM G36.00004 Liquid drop spreading on surfaces: Initial regimes revisited, SURJYASISH MITRA, SUSHANTA MITRA, York University — Liquid drop spreading on a given surface is fundamental towards technological processes like coating and paints, inkjet printing, surface characterization, etc. Though, the underlying dynamics is well understood, we have revisited this problem through experiments conducted on surfaces kept in air as well as immersed in water. It was found that the two key parameters that dictated the spreading process were drop-surrounding medium viscosity ratio and the characteristic viscous length scale. It was observed that irrespective of the drop liquid and surrounding liquid medium (air and water in this case), spreading always began in a regime dominated by drop viscosity, where the spreading radius scales as $r \sim t$. However, the prefactor of the scaling observed was different for air (of the order of unity) and under-water (much less than unity). Following this initial regime, a second intermediate regime dominated by drop inertia (typically found for water drops spreading in air) was observed only when the characteristic viscous length scale favored such a transition. In this regime as well, a non-universal prefactor was noted for the scaling law, i.e., $r \sim t^{1/2}$. In all cases considered, the spreading process terminated in the Tanner's regime where the spreading radius scaled as $r \sim t^{1/10}$.

8:52AM G36.00005 Thickness effect in the statics and dynamics of wetting on soft materials, MENGHUA ZHAO, CNRS UMR 7615 - ESPCI PARIS; CNRS UMR 7057 - Universit Paris Diderot, MATHIEU ROCHE, JULIEN DERVAUX, LAURENT ROYON, CNRS UMR 7057 - Universit Paris Diderot, TETSUHARU NARITA, FRANCOIS LEQUEUX, CNRS UMR 7615 - ESPCI PARIS, LAURENT LIMAT, CNRS UMR 7057 - Universit Paris Diderot, MATIRE ET SYSTEMES COMPLEXES - UMR 7057 TEAM, SCIENCES ET INGNIERIE DE LA MATIRE MOLLE - UMR 7615 TEAM — The wetting of liquids on soft materials such as elastomers has received a great deal of attention in the past decades. Many experiments were performed to gain insight into both the statics and dynamics of wetting in such systems, but most neglected the effect of finite thickness of the gel. Here we report results of a study of the thickness effect on both the statics and dynamics of wetting. We vary the thickness of soft silicone elastomers from 10^{-2} to a few mm. First, we develop a quantitative Schlieren optics enabling us to observe the surface deformation after the deposition of droplets. We measure the vertical deformation outside droplets as a function of droplet size, gel thickness and elasticity. We identify a submicrometer-deep dimple, that extends over mm away from the contact line. Second, we characterize the receding dynamics and we show that the dynamic contact angle, hence dissipation, depends on the thickness of the sample. We rationalize our experiments, with an analytical model accounting for the linear elastic response of the gel bulk and its surface tension. We find excellent agreement with experiments.

9:05AM G36.00006 Wetting on a deformable substrate with finite deformations and asymmetrical substrate surface energies, LAURENT LIMAT, RICCARDO DE PASCALIS, JULIEN DERVAUX, MSC lab., Matiere et Systemes Complexes, UMR7057 of CNRS and Univ Paris Diderot, IOAN IONESCU, LSPM, UPR3407 of CNRS, Institut Galile, Univ. Paris 13, BENOIT PERTHAME, J.-L. Lions lab., Univ. P. & M. Curie, Univ. Paris Diderot and CNRS, UMR7598 — Wetting on soft compounds is still imperfectly understood, especially when the dry and wetted parts of the substrate have two different values of surface energies (contact angle different than 90 degrees). The problem is made very complex by geometrical non-linearities arising from finite slope of the substrate and finite deformations, that must be absolutely considered, to distinguish at second order between Young law and Neuman equilibrium of surface tensions. We have developed a numerical, finite element, code that allows one to minimize surface and bulk energies, with finite deformations and asymmetry of the surface energies. The results are compared to a linear theory based on Green function theory [1,2] and Fredholm integrals, and with recent experiments using X-ray visualization [3]. The non-linear numerics reproduce very well the observed profiles, while the linear approach gives helpful analytical approximates. [1] L. Limat, EPJ-E Soft Matter, 35, 134 (2012). [2] J. Dervaux & L. Limat, Proc. Roy. Soc. A 471, 20140813 (2015). [3] S. J. Park, B. M. Weon, J. S. Lee, J. Lee, J. Kim & J. H. Je, Nature Com. 5, 4369 (2014).

9:18AM G36.00007 Low-order models of the motion of sessile droplets on highly hydrophobic surfaces¹, ALEX WRAY, LYES KAHOUADJI, OMAR MATAR, Imperial College London, STEPHEN DAVIS, Northwestern University — We consider the behaviour of a droplet deposited onto a hydrophobic substrate. This and associated problems have received attention due to their significance in a wide array of experimental and industrial contexts, such as the post-rupture wetting problem is of importance to coating flow applications. Such systems have typically defied low-order analysis due to the multi-valued nature of the interface, but we demonstrate how to resolve this issue in this instance. We begin by analysing the static case. We find that the system is governed by the Young-Laplace equation with the equilibrium shape depending on the Bond number, the contact angle and the volume of the droplet. We solve the system numerically, and use these results to validate a variety of low-order models. We then solve the dynamic problem using both direct numerical simulations and a low-order model based on conservation of energy.

¹EPSRC UK platform grant MACIPh (EP/L020564/1) and programme Grant MEMPHIS (EP/K003976/1)

9:31AM G36.00008 Drop Spreading with Random Viscosity¹, FENG XU, OLIVER JENSEN, The University of Manchester — Airway mucus acts as a barrier to protect the lung. However as a biological material, its physical properties are known imperfectly and can be spatially heterogeneous. In this study we assess the impact of these uncertainties on the rate of spreading of a drop (representing an inhaled aerosol) over a mucus film. We model the film as Newtonian, having a viscosity that depends linearly on the concentration of a passive solute (a crude proxy for mucin proteins). Given an initial random solute (and hence viscosity) distribution, described as a Gaussian random field with a given correlation structure, we seek to quantify the uncertainties in outcomes as the drop spreads. Using lubrication theory, we describe the spreading of the drop in terms of a system of coupled nonlinear PDEs governing the evolution of film height and the vertically-averaged solute concentration. We perform Monte Carlo simulations to predict the variability in the drop centre location and width (1D) or area (2D). We show how simulation results are well described (at much lower computational cost) by a low-order model using a weak disorder expansion. Our results show for example how variability in the drop location is a non-monotonic function of the solute correlation length increases.

¹Engineering and Physical Sciences Research Council

9:44AM G36.00009 Droplet wetting transitions on inclined substrates in the presence of external shear and substrate permeability, LEONARDO ESPIN, SATISH KUMAR, University of Minnesota — Understanding the gravity-driven motion of droplets on inclined substrates in the presence of external shear and substrate permeability is important for applications such as spray coating and filtration. In this work, we use a lubrication-theory-based model to study how external shear and substrate permeability affect droplet wetting transitions. A nonlinear evolution equation for the droplet height as a function of time and two spatial variables is derived and numerically solved. The contact-line region is described using a precursor film and disjoining pressure. Depending on its direction, external shear can either suppress or drive wetting transitions, but does not appear to significantly change the critical droplet speeds associated with these transitions. Substrate permeability generally suppresses wetting transitions due to liquid absorption and does appear to significantly affect these critical droplet speeds. The strong influence of substrate permeability and external shear on droplet wetting transitions indicates that it will be important to account for these effects when developing accurate models for industrial applications.

9:57AM G36.00010 Sinking, wedging, spreading – viscous spreading on a layer of fluid, NICO BERGEMANN, ANNE JUEL, MATTHIAS HEIL, The University of Manchester — We study the axisymmetric spreading of a sessile drop on a pre-existing layer of the same fluid in a regime where the drop is sufficiently large so that the spreading is driven by gravity while capillary and inertial effects are negligible. Experiments performed with 5 ml drops and layer thicknesses in the range $0.1 \text{ mm} \leq h \leq 1 \text{ mm}$ show that at long times the radius of the drop evolves as $R \propto t^n$, where the spreading exponent n increases with the layer thickness h . Numerical simulations, based on the axisymmetric free-surface Navier-Stokes equations, reveal three distinct spreading regimes depending on the layer thickness. For thick layers the drop sinks into the layer, accompanied by significant flow in the layer. By contrast, for thin layers the layer ahead of the propagating front is at rest and the spreading behaviour resembles that of a gravity-driven drop spreading on a dry substrate. In the intermediate regime the spreading is characterised by an advancing wedge, which is sustained by fluid flow from the drop into the layer.

Monday, November 21, 2016 8:00AM - 10:10AM – Session G37 Drops: Impacts including Irregular Surfaces Portland Ballroom 252 - Jeremy Marston, Texas Tech University

8:00AM G37.00001 Impact dynamics of liquid marbles, JEREMY MARSTON, TINKU SUPAKAR, Texas Tech University — The impact of particle coated droplets (a.k.a. liquid marbles or armored droplets) onto solid substrates is assessed experimentally with high-speed video. The impact is characterized by the maximum spread diameter, which conforms to scaling laws in terms of the impact Weber number, meaning that the marbles behave similar to water droplets during this stage. However, the motion of the particles across the surface allows us to observe both clustering and divergence of the particle shell and, in particular, we observe the formation of arrested shapes (i.e. jammed interfaces) after impact onto hydrophobic surfaces, from an initially spherical shape. In this case, we postulate that the speed of retraction and rate of change of surface coverage is a key ingredient leading to arrested shapes.

8:13AM G37.00002 Impact cratering on granular beds: from the impact of raindrops to the strike of hailstones¹, LEONARDO GORDILLO, JUNPING WANG, FRED JAPARDI, WARREN TEDDY, XIANG CHENG, Univ of Minnesota - Twin Cities — The craters generated by the impact of a spherical object onto a granular bed strongly depend on the material properties of impactors. As an example, impact cratering by liquid drops and by solid spheres exhibit qualitatively different power-law scalings for the size of resulting impact craters. While the basic energy conservation and dimensional analysis provide simple guiding rules, the detailed dynamics governing the relation between these power-law scalings are still far from clear. To analyze the transition between liquid-drop and solid-sphere impact cratering, we investigate impact cratering by liquid drops in a wide range of impact energies, viscosities, surface tensions and drop sizes. Using high-speed photography and laser profilometry to survey more than 8000 laboratory-controlled impact cratering events, we fully delineate the solid-to-liquid transition and unveil a rich set of regimes with different scaling laws and crater morphologies. Our research provides a unified framework for understanding the scaling relations in granular impact cratering a phenomenon ubiquitous in nature ranging from daily-life raindrop and hailstone impacts on sandy surfaces to catastrophic asteroid strikes on planetary bodies.

¹Research funded by the National Science Foundation. LG is supported by Conicyt/Becas Chile de Postdoctorado 74150032.

8:26AM G37.00003 ABSTRACT WITHDRAWN –

8:39AM G37.00004 Failure Mechanisms of Air Entrainment in Drop Impact on Lubricated Surfaces¹, MIN PACK, HAN HU, DONG-OOK KIM, Drexel University, ZHONG ZHENG, HOWARD STONE, Princeton University, YING SUN, Drexel University, DREXEL UNIVERSITY TEAM, PRINCETON UNIVERSITY TEAM — Lubricated surfaces have recently been introduced and studied due to their potential benefit in various applications. Combining the techniques of total internal reflection microscopy and reflection interference microscopy, we examine the dynamics of an underlying air film upon drop impact on a lubricated substrate. In contrast to drop impact on solid surfaces where asperities cause random breakup of the entraining air film, we report two air film failure mechanisms on lubricated surfaces. In particular, using thin liquid films of high viscosity, we show that air film rupture shifts from a randomly driven to a controlled event. At low Weber numbers (We) the droplet bounces. At intermediate We , the air film fails at the center as the drop top surface crashes downward owing to impact-induced capillary waves; the resulting liquid-liquid contact time is found to be independent of We . In contrast, at high We , the air film failure occurs much earlier in time at the first inflection point of the air film shape away from the drop center, where the liquid-liquid van der Waals interactions become important. The predictable failure modes of the air film upon drop impact sheds light on droplet deposition in applications such as lubricant-infused self-cleaning surfaces.

¹Support for this work was provided by the National Science Foundation under grant No. CMMI-1401438 to Y.S

8:52AM G37.00005 Asymmetry of Drop Impacts on Patterned Hydrophobic Microstructures, GEOFF WILLMOTT, The University of Auckland, SIMON ROBSON, The University of Melbourne, MATHEU BROOM, The University of Auckland — When a water drop falls on to a flat solid surface, asymmetries in the geometry of the spreading drop can be specifically determined by patterned surface microstructures. For hydrophobic (or superhydrophobic) micropillar arrays, the most important asymmetric mechanisms appear to be the surface energy of contact lines, and pathways for gas escaping from penetrated microstructure [1]. In this presentation, static wetting and drop impact experiments will be discussed in relation to drop asymmetries. In addition to micropillar arrays, natural superhydrophobic surfaces (leaves) have been studied [2], and may suggest possibilities for controlling drop impacts in applications. Some of the clearest large scale drop asymmetries on leaves, which are similar to those associated with low drop impact contact times on synthetic surfaces [3], appear to be caused by features which generate high contact angle hysteresis, and are therefore indicative of poor superhydrophobicity. [1] S. Robson and G. R. Willmott, Soft Matter 12, 4853 (2016). [2] A. Fritsch, G. R. Willmott and M. Taylor, J. R. Soc. N. Z. 43, 198 (2013). [3] J. C. Bird, R. Dhiman, H. M. Kwon and K. K. Varanasi, Nature 503, 385 (2013).

9:05AM G37.00006 Droplet impact on a needle, BEN LOVETT, ANDREW MERRITT, TADD TRUSCOTT, Utah State University — A droplet impinging a hydrophobic surface (low We) spreads to a maximum diameter before retracting to the center of impact and sometimes lifting off. If the impact surface is augmented by a small ridge, the droplet will often split at this surface feature upon spreading resulting in a shorter time to lift off [Bird et. al., Reducing the contact time of a bouncing drop. *Nature*, 503, 2013]. We investigate how a singular feature (needle point) generates a similar reduction in droplet contact time. Droplets of diameter ($< 2 \text{ mm}$) were controlled to impact varying needles at various impact velocities (We range: 16 - 256). While it was initially supposed that splitting the droplet into more pieces would further decrease contact time, this was not observed. Rather, the spreading event at the center of the droplet has a greater effect on contact time.

9:18AM G37.00007 Grating droplets with a mesh , DAN SOTO, ANTOINE LE HELLOCO, Massachusetts Institute of Technology - MechE, CRISTOPHE CLANET, Ecole Polytechnique - Ladhix, DAVID QUERE, Espci - Pmmh, KRIPA VARANASI,

Massachusetts Institute of Technology - MechE — A drop thrown against a mesh can pass through its holes if impacting with enough inertia. As a result, although part of the droplet may remain on one side of the sieve, the rest will end up grated through the other side. This inexpensive method to break up millimetric droplets into micrometric ones may be of particular interest in a wide variety of applications: enhancing evaporation of droplets launched from the top of an evaporative cooling tower or preventing drift of pesticides sprayed above crops by increasing their initial size and atomizing them at the very last moment with a mesh. In order to understand how much liquid will be grated we propose in this presentation to start first by studying a simpler situation: a drop impacting a plate pierced with a single off centered hole. The study of the role of natural parameters such as the radius drop and speed or the hole position, size and thickness allows us to discuss then the more general situation of a plate pierced with multiple holes: the mesh.

9:31AM G37.00008 Drop impact onto semi-infinite solid surface , HUANCHEN CHEN, ALIDAD AMIRFAZLI, York University

— The drop impact onto solid surfaces has been studied intensively due to its importance in different applications, e.g. spray coating, inkjet printing and agricultural sprays. The previous studies on this topic were typically focused either on the drop impact onto an infinite solid surface (i.e. a solid surface that is large, and the impact happens far away from the surface edges), or onto a finite solid surface (e.g. drop impact onto a target smaller than the droplet). However, in practice, it is also possible for the impact onto a large surface but close to its edge (named as semi-infinite surface). In this first study of its kind, the process of drop impact onto a semi-infinite surface (both hydrophobic and hydrophilic) was investigated experimentally. During the impact process, part of the liquid lamella can spread out of the surface (free lamella). Depending on the distance between the impact point and surface edge, the free lamella can recede, or partially recede back to the surface, or completely break apart at the surface edge. The behavior of free lamella can also affect the morphology of the part of liquid lamella which remains in contact with the solid surface, especially in the receding phase (e.g. occurrence of drop rebound). Various morphologies observed for lamella breakage at the surface edge will also be discussed for surfaces of different wettabilities.

9:44AM G37.00009 Droplet impact onto a solid sphere: effect of wettability and impact velocity¹ , ALIDAD AMIRFAZLI, S.A. BANITABEI, York University — Collision of a droplet onto a still spherical particle was experimentally investigated. Effect of droplet impact velocity and wettability of the particle surface on collision outcomes were studied ($0.05 < V_0 < 5.0$ and $\theta = 70, 90, 118^\circ$). Compared to the literature, the range of Weber number variations was significantly extended ($0.1 < We < 1146$), and while focus of the previous works was only on impacts in which particle is larger than the droplet ($D_r < 1$), the drop to particle diameter ratio in this work was larger than one. Therefore, formation of a thin liquid film, i.e. lamella, was observed due to impact of a relatively high velocity droplet onto a hydrophobic particle. It was shown that for hydrophobic targets with $\theta > 710^\circ$, change in particle wettability does not affect the lamella geometry. Temporal variations of various geometrical parameters of collision outcomes including lamella length and lamella base diameter were investigated during the impact. A comprehensive map of all the available works in drop impact on a spherical target was also provided.

¹NSERC

9:57AM G37.00010 Spreading of Impacting Droplets on Wettability-Patterned Surfaces , MOHAMED ELSHARKAWY, University of Illinois at Chicago, ANTONIO RUSSO, PIETRO ASINARI, Politecnico di Torino, CONSTANTINE MEGARIDIS, University of Illinois at Chicago — Droplet collision on solid surfaces is a long-studied field that has focused mostly on droplets striking uniform-wettability surfaces. As of now, very few studies exist that analyzed droplet impact on non-uniform (spatially) wettability surfaces. More importantly, no model exists for predicting droplet impact behavior on spatially non-uniform surfaces. Using photolithographically-produced surfaces, we study droplet impact on axially-symmetric, non-uniform wettability surfaces. We expand upon previously presented models for uniform-wettability surfaces, and predict the maximum spreading diameter of droplets impacting on symmetric patterns on varying wettability surfaces. The present model is expanded to account for n annular regions of different wettabilities, and calculate the corresponding maximum spreading diameter. In addition, within the model we explore the concept of a wettability contrast barrier that must be overcome by the impacting droplets in order to continue their spreading phase. We show under which conditions a droplet can successfully overcome this barrier, and under which conditions it cannot. The model put forth makes strong use of the previously-reported droplet impact model of Passandideh-Fard et al. It draws upon geometric assumptions, such as cylindrical shape for the expanding liquid and spherical cap for the impacting droplet. The work is fundamental in nature, but offers valuable insight that helps understand droplet impact dynamics on non-uniform wettability surfaces.

Monday, November 21, 2016 8:00AM - 10:10AM –
Session G38 Flow Instability: Nonlinear Dynamics Portland Ballroom 255 - Qiqi Wang, Massachusetts Institute of Technology

8:00AM G38.00001 Lyapunov Exponents and Covariant Vectors for Turbulent Flow Simulations , PATRICK BLONIGAN, SCOTT MURMAN, NASA Ames Research Center, PABLO FERNANDEZ, QIQI WANG, Massachusetts Institute of Technology — As computational power increases, engineers are beginning to use scale-resolving turbulent flow simulations for applications in which jets, wakes, and separation dominate. However, the chaotic dynamics exhibited by scale-resolving simulations poses problems for the conventional sensitivity analysis and stability analysis approaches that are vital for design and control. Lyapunov analysis is used to study the chaotic behavior of dynamical systems, including flow simulations. Lyapunov exponents are the growth or a decay rate of specific flow field perturbations called the Lyapunov covariant vectors. Recently, the authors have used Lyapunov analysis to study the breakdown in conventional sensitivity analysis and the cost of new shadowing-based sensitivity analysis [1]. The current work reviews Lyapunov analysis and presents new results for a DNS of turbulent channel flow, wall-modeled channel flow, and a DNS of a low pressure turbine blade. Additionally, the implications of these Lyapunov analyses for computing sensitivities of these flow simulations will be discussed.

[1] Q. Wang, R. Hui, and P. Blonigan. Least squares shadowing sensitivity analysis of chaotic limit cycle oscillations. *Journal of Computational Physics*, 267:210224, June 2014.

8:13AM G38.00002 Stability of Couette flow past a viscoelastic solid , ANDREW HESS,

TONG GAO, Michigan State University — Soft materials such as polymer gels have been widely used in engineering applications such as microfluidics, micro-optics, and active surfaces. It is important to obtain fundamental understandings of the dynamics of various soft materials when interacting with fluid. Here we investigate the material behavior of a viscoelastic solid film immersed in a simple Newtonian Couette flow. An Eulerian formulation of the Zener model is used to model the solid phase with the surface tension effect. A linear stability analysis is first performed to predict the material instabilities induced by the shear flow field, and provide an analytical basis to the numerical results. The nonlinear fluid/elastic structure interactions are further explored by using the direct numerical simulations. Phase tracking is accomplished through the use of a generalized Cahn-Hilliard model for the surface tension between the gel-like material and the ambient fluid. The coupled Cahn-Hilliard/Navier-Stokes/Zener equations are then solved on a staggered grid through a finite difference method. The results are compared with previous studies for both the hyperelastic and viscoelastic materials.

8:26AM G38.00003 Lyapunov exponents, covariant vectors and shadowing sensitivity analysis of 3D wakes: from laminar to chaotic regimes , QIQI WANG, MIT, GEORGIOS RIGAS, Caltech, LUCAS ESCLAPEZ, LUCA MAGRI, Stanford, PATRICK BLONIGAN, NASA — Bluff body flows are of fundamental importance to many engineering applications involving massive flow separation and in particular the transport industry. Coherent flow structures emanating in the wake of three-dimensional bluff bodies, such as cars, trucks and lorries, are directly linked to increased aerodynamic drag, noise and structural fatigue. For low Reynolds laminar and transitional regimes, hydrodynamic stability theory has aided the understanding and prediction of the unstable dynamics. In the same framework, sensitivity analysis provides the means for efficient and optimal control, provided the unstable modes can be accurately predicted. However, these methodologies are limited to laminar regimes where only a few unstable modes manifest. Here we extend the stability analysis to low-dimensional chaotic regimes by computing the Lyapunov covariant vectors and their associated Lyapunov exponents. We compare them to eigenvectors and eigenvalues computed in traditional hydrodynamic stability analysis. Computing Lyapunov covariant vectors and Lyapunov exponents also enables the extension of sensitivity analysis to chaotic flows via the shadowing method. We compare the computed shadowing sensitivities to traditional sensitivity analysis. These Lyapunov based methodologies do not rely on mean flow assumptions, and are mathematically rigorous for calculating sensitivities of fully unsteady flow simulations.

8:39AM G38.00004 Tangent double Hopf bifurcation in a counter-rotating split cylinder¹ , PALOMA GUTIERREZ-CASTILLO, JUNA M. LOPEZ, Arizona State University — A tangent double Hopf bifurcation has been found in the flow in a counter-rotating cylinder split at its mid-plane. The cylinder of radius a and length h is completely filled with fluid of kinematic viscosity ν . Both halves rotate with the same angular speed ω , but in opposite directions. The flow, which is solved numerically via spectral methods, is dominated by the shear layer between both halves of the cylinder. For sufficiently small $Re = \omega a^2 / \nu$, the basic state, which is axisymmetry, reflection symmetric and steady, is stable. A range of aspect ratios $\Gamma = h/a$ were studied, with $\Gamma \in [0.5, 2]$. The basic state loses stability via a number of different Hopf bifurcations breaking axisymmetry, leading to waves with different azimuthal wavenumbers. For $\Gamma \in [1.3, 1.76]$, there is a double Hopf bifurcation of waves with $m = 2$ and $m = 3$. This bifurcation divides (Γ, Re) space into 6 different regions, some of which include multiple stable states due to the nonlinearities of the problem. The states in the various regions are very well described by the normal form dynamics of the bifurcation, which we use to gain deeper insights into the complicated dynamics created by the nonlinear competition between the various states.

¹U.S. National Science Foundation grant CBET- 1336410

8:52AM G38.00005 Extension to nonlinear stability theory of the circular Couette flow , PUN WONG YAU, SHIXIAO WANG, Auckland University, ZVI RUSAK, Rensselaer Polytechnic Institute — A nonlinear stability analysis of the viscous circular Couette flow to axisymmetric perturbations under axial periodic boundary conditions is developed. The analysis is based on investigating the properties of a reduced Arnol'd energy-Casimir function A_{rd} of Wang (2009). We show that all the inviscid flow effects as well as all the viscous-dependent terms related to the flow boundaries vanish. The evolution of ΔA_{rd} depends solely on the viscous effects of the perturbation's dynamics inside the flow domain. The requirement for the temporal decay of ΔA_{rd} leads to novel sufficient conditions for the nonlinear stability of the circular Couette flow in response to axisymmetric perturbations. Comparisons with historical studies show that our results shed light on the experimental measurements of Wendt (1933) and significantly extend the classical nonlinear stability results of Serrin (1959) and Joseph & Hung (1971). When the flow is nonlinearly stable and evolves axisymmetrically for all time, then it always decays asymptotically in time to the circular Couette flow determined uniquely by the setup of the rotating cylinders. This study provides new physical insights into a classical flow problem that was studied for decades.

9:05AM G38.00006 Nonlinear Pattern Selection in Bi-Modal Interfacial Instabilities¹ , JASON PICARDO, Department of Chemical Engineering, Indian Institute of Technology Madras, RANGA NARAYANAN, Department of Chemical Engineering, University of Florida — We study the evolution of two interacting unstable interfaces, with the aim of understanding the role of non-linearity in pattern selection. Specifically, we consider two superposed thin films on a heated surface, that are susceptible to thermocapillary and Rayleigh-Taylor instabilities. Due to the presence of two unstable interfaces, the dispersion curve (linear growth rate plotted as a function of the perturbation wavelength) exhibits two peaks. If these peaks have equal heights, then the two corresponding disturbance patterns will grow with the same linear growth rate. Therefore, any selection between the two must occur via nonlinear effects. The two-interface problem under consideration provides a variety of such bi-modal situations, in which the role of nonlinearity in pattern selection is unveiled. We use a combination of long wave asymptotics, numerical simulations and amplitude expansions to understand the subtle nonlinear interactions between the two peak modes. Our results offer a counter-example to Rayleigh's principle of pattern formation, that the fastest growing linear mode will dominate the final pattern. Far from being governed by any such general dogma, the final selected pattern varies considerably from case to case.

¹The authors acknowledge funding from NSF (0968313) and the Fulbright-Nehru fellowship

9:18AM G38.00007 A recurrence network approach to analyzing forced synchronization in hydrodynamic systems¹, MEENATCHIDEVI MURUGESAN, YUANHANG ZHU, LARRY K.B. LI, Hong Kong University of Science and Technology — Hydrodynamically self-excited systems can lock into external forcing, but their lock-in boundaries and the specific bifurcations through which they lock in can be difficult to detect. We propose using recurrence networks to analyze forced synchronization in a hydrodynamic system: a low-density jet. We find that as the jet bifurcates from periodicity (unforced) to quasiperiodicity (weak forcing) and then to lock-in (strong forcing), its recurrence network changes from a regular distribution of links between nodes (unforced) to a disordered topology (weak forcing) and then to a regular distribution again at lock-in (strong forcing). The emergence of order at lock-in can be either smooth or abrupt depending on the specific lock-in route taken. Furthermore, we find that before lock-in, the probability distribution of links in the network is a function of the characteristic scales of the system, which can be quantified with network measures and used to estimate the proximity to the lock-in boundaries. This study shows that recurrence networks can be used (i) to detect lock-in, (ii) to distinguish between different routes to lock-in, and (iii) as an early warning indicator of the proximity of a system to its lock-in boundaries.

¹This work was supported by the Research Grants Council of Hong Kong (Project No. 16235716 and 26202815).

9:31AM G38.00008 Unravelling the mechanism behind Swirl-Switching in turbulent bent pipes, PHILIPP SCHLATTER, LORENZ HUFNAGEL, JACOPO CANTON, RAMIS ÖRLÜ, Linné FLOW Centre, KTH Mechanics, OANA MARIN, ELIA MERZARI, Mathematics and Computer Science Division, Argonne National Laboratory — Turbulent flow through pipe bends has been extensively studied, but several phenomena still miss an exhaustive explanation. Due to centrifugal forces, the fluid flowing through a curved pipe forms two symmetric, counter-rotating Dean vortices. It has been observed, experimentally and numerically, that these vortices change their size, intensity and axis in a periodic, oscillatory fashion, a phenomenon known as swirl-switching. These oscillations are responsible for failure due to fatigue in pipes, and their origin has been attributed to a recirculation bubble, disturbances coming from the upstream straight section and others. The present study tackles the problem by direct numerical simulations (DNS) analysed, for the first time, with three-dimensional proper orthogonal decomposition (POD) as to distinguish between the spatial and temporal contributions. The simulations are performed at a friction Reynolds number of about 360 with a divergence-free synthetic turbulence inflow, as to avoid the interference of low-frequency oscillations generated by a standard recycling method. Results indicate that a single low-frequency, three-dimensional POD mode, representing a travelling wave, and previously mistaken by 2D POD for two different modes, is responsible for the swirl-switching.

9:44AM G38.00009 Homoclinic snaking in plane Couette flow: bending, skewing, and finite-size effects, JOHN GIBSON, Univ of New Hampshire, TOBIAS SCHNEIDER, cole polytechnique fdrale de Lausanne — Invariant solutions of shear flows have recently been extended from spatially periodic solutions in minimal flow units to spatially localized solutions on extended domains. One set of spanwise-localized solutions of plane Couette flow exhibits homoclinic snaking, a process by which steady-state solutions grow additional structure smoothly at their fronts when continued parametrically. In this talk, we present a numerical study of the snaking solutions, generalizing beyond the fixed streamwise wavelength of previous studies. We find a number of new solution features, including bending, skewing, and finite-size effects. We establish the parameter regions over which snaking occurs and show that the finite-size effects of the traveling-wave solution are due to a coupling between its fronts and interior that results from its shift-reflect symmetry. A new winding solution of plane Couette flow is derived from a strongly-skewed localized equilibrium.

9:57AM G38.00010 Dampening the asymmetric instability in pipe flow of shear thinning fluids using elasticity, DAVID DENNIS, CHAOFAN WEN, ROBERT POOLE, University of Liverpool — Recent experimental results have shown that the asymmetric flow of shear-thinning fluid through a cylindrical pipe, which was previously associated with the laminar-turbulent transition process, is actually a non-hysteretic and reversible, supercritical instability of the laminar base state. These experiments were performed using largely inelastic shear-thinning fluids (aqueous solutions of xanthan gum) and it was found that the greater the degree of shear-thinning the larger the magnitude of the asymmetry. In this talk we show that a viscoelastic fluid (an aqueous solution of high molecular weight polyacrylamide), with approximately the same shear-thinning characteristics as the inelastic fluid, does not exhibit the asymmetry when freshly mixed. However, once the elasticity of this fluid is degraded (by prolonged shearing) the asymmetry reappears. This suggests that the shear-thinning nature of the fluid causes the instability and the viscoelastic nature works to dampen the asymmetry. To test this hypothesis we add varyingly small amounts of polyacrylamide to xanthan gum solutions and find an inverse relationship between viscoelasticity and the magnitude of the asymmetry, although the Reynolds number at which the instability is first observed stays approximately constant.

Monday, November 21, 2016 8:00AM - 10:10AM – Session G39 Bio: Phonation and Speech Portland Ballroom 256 - Michael Krane, Pennsylvania State University

8:00AM G39.00001 Aerosol Emission during Human Speech, SIMA ASADI, WILLIAM RISTENPART, Dept. Chemical Engineering, University of California Davis — The traditional emphasis for airborne disease transmission has been on coughing and sneezing, which are dramatic expiratory events that yield easily visible droplets. Recent research suggests that normal speech can release even larger quantities of aerosols that are too small to see with the naked eye, but are nonetheless large enough to carry a variety of pathogens (e.g., influenza A). This observation raises an important question: what types of speech emit the most aerosols? Here we show that the concentration of aerosols emitted during healthy human speech is positively correlated with both the amplitude (loudness) and fundamental frequency (pitch) of the vocalization. Experimental measurements with an aerodynamic particle sizer (APS) indicate that speaking in a loud voice (95 decibels) yields up to fifty times more aerosols than in a quiet voice (75 decibels), and that sounds associated with certain phonemes (e.g., [a] or [o]) release more aerosols than others. We interpret these results in terms of the egressive airflow rate associated with each phoneme and the corresponding fundamental frequency, which is known to vary significantly with gender and age. The results suggest that individual speech patterns could affect the probability of airborne disease transmission.

8:13AM G39.00002 Flow-Structure-Acoustic Interaction Simulation of Vocalization of a Non-song Bird., WEILI JIANG, XUDONG ZHENG, QIAN XUE, University of Maine, JEPPE HAVE RASMUSSEN, COEN ELEMANS, University of Southern Denmark, COMPLEX FLOW MODELING & SIMULATION LAB TEAM, ELEMANS LAB TEAM — The myoelastic-aerodynamic mechanism of vocalization was recently evidenced in birds across a wide range of taxa, which, for a long time, was believed to generate sound based on the aerodynamic whistle mechanism. The objective of the current study is to: (1) develop a first-principle based, flow-structure-acoustics (FSA) interaction computational model of a non-song bird (rock pigeon); (2) strongly validate the computational model by comparing to the experimental data on the same bird model; (3) examine the data so as to generate new insights into the physics of vocalization of birds. In the current approach, a sharp interface immersed boundary method based incompressible flow solver is utilized to model the air flow; A finite element based solid mechanics solver is utilized to model the LVM(lateral vibratory mass) vibration; A high-order immersed boundary method based acoustics solver is utilized to directly compute sound. Geometric structure of the syrinx, including syringeal lumen, LVM, position of tracheal rings, is based on CT scan of a rock pigeon. The LVM is simulated as isotropic material according to the experimental measurements. Simulation setup about the bronchial pressure, static deformation due to air sac pressure also follows the setup in the experiments. Results including the fundamental frequency, air flow rate, gap, vibration shape will be analyzed and compared to the experimental data.

8:26AM G39.00003 Direct numerical simulation of human phonation¹, SHAKTI SAURABH, DANIEL BODONY, University of Illinois at Urbana-Champaign — A direct numerical simulation study of the generation and propagation of the human voice in a full-body domain is conducted. A fully compressible fluid flow model, anatomically representative vocal tract geometry, finite deformation model for vocal fold (VF) motion and a fully coupled fluid-structure interaction model are employed. The dynamics of the multi-layered VF tissue with varying stiffness are solved using a quadratic finite element code. The fluid-solid domains are coupled through a boundary-fitted interface and utilize a Poisson equation-based mesh deformation method. A new inflow boundary condition, based upon a quasi-1D formulation with constant sub-glottal volume velocity, linked to the VF movement, has been adopted. Simulations for both child and adult phonation were performed. Acoustic characteristics obtained from these simulation are consistent with expected values. A sensitivity analysis based on VF stiffness variation is undertaken and sound pressure level/fundamental frequency trends are established. An evaluation of the data against the commonly-used quasi-1D equations suggest that the latter are not sufficient to model phonation. Phonation threshold pressures are measured for several VF stiffness variations and comparisons to clinical data are carried out.

¹Supported by the National Science Foundation (CAREER award number 1150439)

8:39AM G39.00004 ABSTRACT WITHDRAWN —

8:52AM G39.00005 Does a pneumotach accurately characterize voice function?, GAGE WALTERS, MICHAEL KRANE, Penn State University — A study is presented which addresses how a pneumotach might adversely affect clinical measurements of voice function. A pneumotach is a device, typically a mask, worn over the mouth, in order to measure time-varying glottal volume flow. By measuring the time-varying difference in pressure across a known aerodynamic resistance element in the mask, the glottal volume flow waveform is estimated. Because it adds aerodynamic resistance to the vocal system, there is some concern that using a pneumotach may not accurately portray the behavior of the voice. To test this hypothesis, experiments were performed in a simplified airway model with the principal dimensions of an adult human upper airway. A compliant constriction, fabricated from silicone rubber, modeled the vocal folds. Variations of transglottal pressure, time-averaged volume flow, model vocal fold vibration amplitude, and radiated sound with subglottal pressure were performed, with and without the pneumotach in place, and differences noted. (Acknowledge support of NIH grant 2R01DC005642-10A1.)

9:05AM G39.00006 Power flow in normal human voice production, MICHAEL KRANE, Penn State University — The principal mechanisms of energy utilization in voicing are quantified using a simplified model, in order to better define voice efficiency. A control volume analysis of energy utilization in phonation is presented to identify the energy transfer mechanisms in terms of their function. Conversion of subglottal airstream potential energy into useful work done (vocal fold vibration, flow work, sound radiation), and into heat (sound radiation absorbed by the lungs, glottal jet dissipation) 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. (Acknowledge support of NIH grant 2R01DC005642-10A1.)

9:18AM G39.00007 A budget of energy transfer in a sustained vocal folds vibration in glottis¹, LUCY ZHANG, JUBIAO YANG, Rensselaer Polytechnic Institute, MICHAEL KRANE, Pennsylvania State University — A set of force and energy balance equations using the control volume approach is derived based on the first principles of physics for a sustained vocal folds vibration in glottis. The control volume analysis is done for compressible airflow in a moving and deforming control volume in the vicinity of the vocal folds. The interaction between laryngeal airflow and vocal folds are successfully simulated using the modified Immersed Finite Element Method (mIFEM), a fully coupled approach to simulate fluid-structure interactions. Detailed mathematical terms are separated out for deeper physical understanding and utilization of mechanical energy is quantified with the derived equation. The results show that majority of energy input is consumed for driving laryngeal airflow, while a smaller portion is for compensating viscous losses in and sustaining the vibration of the vocal folds.

¹We acknowledge the funding support of NIH 2R01DC005642-10A1

9:31AM G39.00008 A finite element study on the cause of vocal fold vertical stiffness gradient, BIAO GENG, QIAN XUE, XUDONG ZHENG, University of Maine, Orono — Vertical stiffness variation (VSV) on the vocal fold medial surface was recently reported and was hypothesized to be an important feature for phonation as it can promote the divergent angle during vibration. However, the underlying mechanism of such feature remains unclear. In our opinion, there are three primary mechanisms that could contribute to the overall stiffness variation, including the material variation in the cover layer, the superior-inferior asymmetry of the vocal fold structure and the presence of the conus elasticus. The current study aims to use the finite element method to quantify the contribution of these three mechanisms to the VSV. The preliminary results showed that the material variation and structural asymmetry can have a significant effect on the VSV, however, the presence conus elasticus had nearly negligible effect. The structural asymmetry due to the subglottal angle caused about 15%~20% increase in VSV when the subglottal angle beyond 40, and its effect was more significant at small subglottal angles.

9:44AM G39.00009 Glottal jet inertance, MICHAEL MPHAIL, MICHAEL KRANE, Penn State University —

Estimates of an inertive contribution of the glottal jet to glottal aerodynamic resistance is presented. Given that inertance of the flow in a constriction can be expressed in terms of the kinetic energy of the flow, and that a jet is a maximum kinetic energy flow pattern, it is argued that the glottal jet possesses its own inertance which is at least as large as that of the vocal tract. These arguments are supported by estimates of inertance obtained from simulations of an unsteady flow through an axisymmetric orifice, and of a compliant constriction with the approximate shape and mechanical properties of the vocal folds. It is further shown that the inertive effect of the glottal jet depends on the jet path and jet mixing, with a slowly diffusing, symmetric jet showing higher inertance than an asymmetric jet which rapidly mixes with supraglottal air. (Acknowledge support of NIH grant 2R01DC005642-10A1.)

9:57AM G39.00010 The impact of intraglottal vortices on vocal fold dynamics¹, BYRON ERATH, ALIREZA PIRNIA, Clarkson University, SEAN PETERSON, University of Waterloo —

During voiced speech a critical pressure is produced in the lungs that separates the vocal folds and creates a passage (the glottis) for airflow. As air passes through the vocal folds the resulting aerodynamic loading, coupled with the tissue properties of the vocal folds, produces self-sustained oscillations. Throughout each cycle a complex flow field develops, characterized by a plethora of viscous flow phenomena. Air passing through the glottis creates a jet, with periodically-shed vortices developing due to flow separation and the Kelvin-Helmholtz instability in the shear layer. These vortices have been hypothesized to be a crucial mechanism for producing vocal fold vibrations. In this study the effect of vortices on the vocal fold dynamics is investigated experimentally by passing a vortex ring over a flexible beam with the same non-dimensional mechanical properties as the vocal folds. Synchronized particle image velocimetry data are acquired in tandem with the beam dynamics. The resulting impact of the vortex ring loading on vocal fold dynamics is discussed in detail.

¹This work was supported by the National Science Foundation Grant CBET 1511761

Monday, November 21, 2016 8:00AM - 10:10AM –

Session G40 Porous Media Flows: Surface Wetting and Filtering Portland Ballroom 253-258-254-257 - Parisa Mirbod, Clarkson University

8:00AM G40.00001 On the effect of the interface heterogeneity between porous and free flow domain, PARISA MIRBOD, Clarkson University, NADINE FALKNER, University of Stuttgart, ZHENXING WU, Clarkson University, HOLGER STEEB, University of Stuttgart, UNIVERSITY OF STUTTGART TEAM, CLARKSON UNIVERSITY TEAM — We study microscopic velocity and shear stress profiles in interfacial transition zones that separate a free Navier-Stokes flow domain and a porous Darcy flow domain using pore scale Direct Numerical Simulation (DNS) and physical experiments using Particle Image Velocimetry (PIV). We focus on the impact of the interfacial heterogeneity and the influence of the onset of sediment transport in shallow water. While both DNS and PIV measurements show that far from the interface velocity profiles are parallel and representative of creeping flow, shear stress-induced recirculating flows exist in micro-cavities of the permeable porous surface. Local velocity and shear stresses inside recirculating flows are irregular, distinct enveloping upper and lower bounds can be constructed. The upper bound corresponds to a no-slip condition at interfaces to the solid phase of the porous material. The lower bound is related to the largest micro-cavity size.

8:13AM G40.00002 Velocity measurement of flow over random soft porous media, ISREAL SELKIRK, PARISA MIRBOD, Clarkson University — The aim of this work is to experimentally examine the flow over random soft porous media in a three-dimensional channel. Various combination of fibrous material and the morphology of the fibers were chosen to achieve void volume fraction (ε) ranging from 0.4 to 0.7. Care has been taken to keep the Reynolds number low so that the flow was laminar. The channel height was constant, however the thickness of the fibrous media was varied to achieve different filling fraction. Before starting the tests in the duct with fiber arrays, a series of tests in an empty duct (i.e., without fibers) conducted to validate the experimental measurements. We also discussed the error and uncertainty sources in the experiments and described the techniques to improve their impact. We studied detailed velocity measurements of the flow over fibrous material inside a rectangular duct using a planar particle image velocimetry (PIV) technique. Using these measurements, we determined the values of the slip velocity at the interface between the fibrous media and the flow. It was found that values of the slip velocity normalized by the maximum velocity in the flow depend on solid volume fraction, pore spaces, and fraction of channel filled by the fiber layers.

8:26AM G40.00003 Capillary-driven, spatially-directed liquid transport on and through thin porous substrates, SOUVICK CHATTERJEE, PALLAB SINHA MAHAPATRA, ALI IBRAHIM, Univ of Illinois - Chicago, RANJAN GANGULY, Jadavpur University- India, CONSTANTINE MEGARIDIS, Univ of Illinois - Chicago, LISHA YU, RICHARD DODGE, Kimberly-Clark Corporation — Thin porous substrates exhibit good wicking properties for liquid distribution. The low cost of such common substrates often makes them useful for point of care biomedical diagnostics. Isotropic and anisotropic liquid transport through porous media has been studied extensively in literature. Moreover, previous research has demonstrated spatially-directed liquid transport on textured surfaces featuring surface-tension confined track. Combining both these features, here we demonstrate and analyze capillary-driven, directional liquid transport both on the surface of, and through, a wettability-patterned, horizontal porous substrate. The vertical (through) penetration is governed by Darcy's law. The horizontal (on surface) transport is driven by the Laplace pressure gradient caused by the geometry of the meniscus on the wettability-confined track. The transport rate on the substrate is found to far exceed the liquid permeation rate through it. Consequently, the penetration resistance can be estimated using a quasi-static approach. Using a semi-analytical model, we analyze the effect of the liquid curvature on the penetration rate of a sessile drop placed on the substrate. The model accounts for the back pressure caused by the liquid on the opposing side. The transport model is validated against the experiments, and the geometry, wettability and substrate porosity parameters causing fastest transport are identified.

8:39AM G40.00004 Capillary rise in a textured channel, DANIEL BEILHARZ, CHRISTOPHE CLANET, DAVID QUERE, Ecole polytechnique - ESPCI — A wetting liquid can invade a textured material, for example a forest of micropillars. The driving and the viscous forces of this motion are determined by the texture parameters and the influence of shape, height and spacing of posts has been widely studied for the last decade. In this work, we build a channel with textured walls. Brought into contact with a reservoir of wetting liquid, we observe in some cases two advancing fronts. A first one ahead invading the forest of micropillars, and a second one behind filling the remaining gap. We study and model the conditions of existence and the dynamics of these two fronts as a function of the characteristics of both microstructure and gap of this elementary porous medium.

8:52AM G40.00005 Surface Diffusion Effect on Gas Transport in Nanoporous Materials¹, TAKUMA HORI, YUTA YOSHIMOTO, SHU TAKAGI, IKUYA KINEFUCHI, The University of Tokyo — Polymer electrolyte fuel cells are one of the promising candidates for power sources of electric vehicles. For further improvement of their efficiency in high current density operation, a better understanding of oxygen flow inside the cells, which have micro- or nanoporous structures, is necessary. Molecular simulations such as the direct simulation of Monte Carlo (DSMC) are necessary to elucidate flow phenomena in micro- or nanostructures since the Knudsen number is close to unity. Our previous report showed that the oxygen diffusion resistance in porous structures with a characteristic pore size of ~100 nm calculated by DSMC agrees well with that measured experimentally. On the other hand, when it comes to the transport in structures with much smaller pore sizes, it is expected that the surface diffusion has a significant impact on gas transport because of their higher specific surface area. Here we present the calculation of gas transport in porous structures with considering surface diffusion. The numerical porous structure models utilized in our simulations are constructed from three-dimensional imaging of materials. The effect of the distance of random walk on the total diffusion resistance in the structures is discussed.

¹This paper is based on results obtained from a project commissioned by the New Energy and Industrial Development Organization (NEDO).

9:05AM G40.00006 Optimizing internal structure of membrane filters¹, LINDA CUMMINGS, PEJMAN SANAEI, New Jersey Institute of Technology — Membrane filters are in widespread use, and manufacturers have considerable interest in improving their performance, in terms of particle retention properties, and total throughput over the filter lifetime. In this regard, it has long been known that membrane properties should not be uniform over the membrane depth; rather, membrane permeability should decrease in the direction of flow. While much research effort has been focused on investigating favorable membrane permeability gradients, this work has been largely empirical in nature. We present a simple, first-principles model for flow through and fouling of a membrane filter, accounting for permeability gradients via variable pore size. Our model accounts for two fouling modes: sieving; and particle adsorption within pores. For filtration driven by a fixed pressure drop, flux through the membrane eventually goes to zero, as fouling occurs and pores close. We address issues of filter performance as the internal pore structure is varied, by comparing the total throughput obtained with equal-resistance membranes. Within certain classes of pore profiles we are able to find the optimum pore profile that maximizes total throughput over the filter lifetime, while maintaining acceptable particle removal from the feed.

¹Partial support from NSF DMS 1261596 is gratefully acknowledged.

9:18AM G40.00007 Modeling branching pore structures in membrane filters¹, PEJMAN SANAEI, LINDA J. CUMMINGS, New Jersey Inst of Tech — Membrane filters are in widespread industrial use, and mathematical models to predict their efficacy are potentially very useful, as such models can suggest design modifications to improve filter performance and lifetime. Many models have been proposed to describe particle capture by membrane filters and the associated fluid dynamics, but most such models are based on a very simple structure in which the pores of the membrane are assumed to be simple circularly-cylindrical tubes spanning the depth of the membrane. Real membranes used in applications usually have much more complex geometry, with interconnected pores which may branch and bifurcate. Pores are also typically larger on the upstream side of the membrane than on the downstream side. We present an idealized mathematical model, in which a membrane consists of a series of bifurcating pores, which decrease in size as the membrane is traversed. Feed solution is forced through the membrane by applied pressure, and particles are removed from the feed either by sieving, or by particle adsorption within pores (which shrinks them). Thus the membrane's permeability decreases as the filtration progresses, ultimately falling to zero. We discuss how filtration efficiency depends on the characteristics of the branching structure.

¹Partial support from NSF DMS 1261596 is gratefully acknowledged.

9:31AM G40.00008 Fast Simulation of Membrane Filtration by Combining Particle Retention Mechanisms and Network Models¹, ARMIN KRUPP, IAN GRIFFITHS, COLIN PLEASE, University of Oxford — Porous membranes are used for their particle retention capabilities in a wide range of industrial filtration processes. The underlying mechanisms for particle retention are complex and often change during the filtration process, making it hard to predict the change in permeability of the membrane during the process. Recently, stochastic network models have been shown to predict the change in permeability based on retention mechanisms, but remain computationally intensive. We show that the averaged behaviour of such a stochastic network model can efficiently be computed using a simple partial differential equation. Moreover, we also show that the geometric structure of the underlying membrane and particle-size distribution can be represented in our model, making it suitable for modelling particle retention in interconnected membranes as well. We conclude by demonstrating the particular application to microfluidic filtration, where the model can be used to efficiently compute a probability density for flux measurements based on the geometry of the pores and particles.

¹A. U. K. is grateful for funding from Pall Corporation and the Mathematical Institute, University of Oxford. I.M.G. gratefully acknowledges support from the Royal Society through a University Research Fellowship.

9:44AM G40.00009 Effect of adhesion on particle clogging in fiber filtration, RAN TAO, SHUIQING LI, MENGMENG YANG, Tsinghua University — A new multi-time step approach combining a discrete element method with computational fluid dynamics is developed to investigate the clogging phenomenon in two-fiber filtration system of micro-particles. A dimensionless adhesion parameter, Ad , is introduced to characterize clogging of particles during the filtration, while the Stokes number and the interception parameter are kept constant. The results indicate that, in the adhesion-dominated regime, clogging definitely happens at $Ad=16$ or larger, which identifies two distinct zones of unclogging and clogging. Particularly, we find a best clogging range of $Ad=18$ to 36 with much shorter clogging time and fewer particles penetrating through. According to the morphological characteristics of deposited particle chains, the clogging time can be further decomposed into chain growing time and bridging time. Despite the shorter bridging time under larger Ad cases, we demonstrate that the delay of clogging can be solely attributed to the increasing chain growing time. This finding highlights that the short-range van der Waals adhesion plays a crucial part during particle clogging process, and is believed to be helpful for the understanding of filtration.

9:57AM G40.00010 Darcy permeability of hagfish slime: an ultra-soft hydrogel, GAURAV CHAUDHARY, University of Illinois at Urbana-Champaign, IL USA, DOUGLAS FUDGE, University of Guelph, ON, Canada, RANDY EWOLDT, University of Illinois at Urbana-Champaign, IL USA — When under attack from predators, hagfish produces a large amount of slime. The slime is an exceptional hydrogel, which sets-up in fraction of a second and is known to choke the predators. A small quantity of exudate, released from specialized slime glands, mixes with a large volume of sea water (99.99% w/v) and forms a mucus-like cohesive mass. The exudate has two main constituents: mucins and long intermediate filament based threads. This remarkably dilute material forms into a solid and is hypothesized to have a low hydrodynamic permeability. In this work, we present the first experimental measurements of Darcy permeability of hagfish slime. Our results explain how this ultra-soft hydrogel possesses the so-called 'gill-clogging' ability. We also investigate the roles played by individual components of slime, namely, thread cells and mucins, via a concentration-dependent permeability study. Our results provide vital insights into the roles of individual components and it is evident from our observations that mucins play a vital role in significantly reducing the permeability of the fibrous network formed by threads.

Monday, November 21, 2016 10:40AM - 12:50PM –

Session H1 Instabilities at Soft Interfaces II A105 - Chris MacMinn, Department of Engineering Science, University of Oxford

10:40AM H1.00001 Bubbles Rising Through a Soft Granular Material¹, ROBIN LE MESTRE, Département de Génie Mécanique, ENS Cachan, CHRIS MACMINN, Department of Engineering Science, University of Oxford, SUNGYON LEE, Department of Mechanical Engineering, Texas A&M University — Bubble migration through a soft granular material involves a strong coupling between the bubble dynamics and the deformation of the material. This is relevant to a variety of natural processes such as gas venting from sediments and gas exsolution from magma. Here, we study this process experimentally by injecting air bubbles into a quasi-2D packing of soft hydrogel beads and measuring the size, speed, and morphology of the bubbles as they rise due to buoyancy. Whereas previous work has focused on deformation resisted by intergranular friction, we focus on the previously inaccessible regime of deformation resisted by elasticity. At low confining stress, the bubbles are irregular and rounded, migrating via local rearrangement. At high confining stress, the bubbles become unstable and branched, migrating via pathway opening.

¹The authors thank The Royal Society for support (International Exchanges Ref IE150885)

10:53AM H1.00002 Dynamic of particle-laden liquid sheet, ALBAN SAURET, PIERRE JOP, ANTHONY TROGER, SVI, CNRS/Saint-Gobain — Many industrial processes, such as surface coating or liquid transport in tubes, involve liquid sheets or thin liquid films of suspensions. In these situations, the thickness of the liquid film becomes comparable to the particle size, which leads to unexpected dynamics. In addition, the classical constitutive rheological law cannot be applied as the continuum approximation is no longer valid. Here, we consider experimentally a transient free liquid sheet that expands radially. We characterize the influence of the particles on the shape of the liquid film as a function of time and the atomization process. We highlight that the presence of particles modifies the thickness and the stability of the liquid sheet. Our study suggests that the influence of particles through capillary effects can modify significantly the dynamics of processes that involve suspensions and particles confined in liquid films.

11:06AM H1.00003 Flow-induced compaction of soft poroelastic materials, JAPINDER S. NIJER, DUNCAN R. HEWITT, M. GRAE WORSTER, JEROME A. NEUFELD, Univ of Cambridge — Fluid flows through poroelastic materials can result in solid deformation driven by the distribution of viscous shear stresses. The porosity and permeability of the solid matrix is altered spatially through a non-trivial coupling to the fluid flow. This behaviour is studied experimentally by examining fluid flow through a packing of soft hydrogel spheres driven by an imposed pressure head. The pressure head is varied, and, for each pressure, the steady-state mass flux and solid deformation are measured. For large pressure gradients, the fluid flow is found to decrease the permeability in such way as to produce a flux that is independent of the applied pressure gradient. Measurements of the internal deformation, obtained by particle tracking, show that the medium compacts non-uniformly, with the porosity being lower at the outlet compared to the inlet. Intriguingly, we find a reproducible hysteresis of the poroelastic deformation between increasing and decreasing increments of the applied pressure head. The experimental results are compared to a simple one-dimensional model that accounts for non-linear elasticity of the solid and non-constant permeability.

11:19AM H1.00004 Droplets sliding down inclined planes: unexpected dynamics on elastomer plates, AURELIE HOURLIER-FARGETTE, ARNAUD ANTKOWIAK, SEBASTIEN NEUKIRCH, Institut Jean Le Rond d'Alembert, UPMC — Droplet dynamics on an angled surface results from a competition between the weight of the droplet, capillary forces, and viscous dissipation inside the drop. The motion of droplets on stiff surfaces has been investigated for a long time, both experimentally and theoretically, while recent studies have shown the interesting physics underlying the sliding of droplets on soft surfaces. We focus on the dynamics of water-glycerol mixture droplets sliding down vertical plates of silicone elastomers, highlighting an unexpected behavior: the droplet dynamics on such a surface includes two regimes with different constant speeds. These results contrast with those found in the literature for droplets sliding on materials such as treated glass. We investigate the universality of this behavior on various elastomers, and study in detail the two regimes and the sharp transition observed between them. Different candidates can be responsible for the sudden speed change: bistability, chemical interaction with the substrate, softness of the material, etc. Our experiments to clarify the role of each of them reveal an unexpected link between microscopic phenomena at the scale of the polymer matrix and the macroscopic dynamics of a droplet.

11:32AM H1.00005 Formation and destabilization of the particle band on the fluid-fluid interface, JUNGCHUL KIM, FENG XU, SUNGYON LEE, Texas A&M University — An inclusion of spherical particles in a viscous fluid can fundamentally change the interfacial dynamics and even cause interfacial instabilities. For instance, particle-induced viscous fingering has been previously observed even in the absence of the destabilizing viscosity ratio, when particles are added to the viscous invading fluid inside a Hele-Shaw cell. In the same flow configuration, the effects of channel confinement lead to the appearance of a novel fingering regime which consists of the formation and break-up of a dense particle band on the interface. In this talk, we experimentally characterize the evolution of the fluid-fluid interface in this new physical regime and propose a simple model of the particle band that successfully captures the onset of fingering as a function of the particle concentrations and particle size.

11:45AM H1.00006 Influence of surface tension on the instabilities and bifurcations of a particle in a drop under shear¹, FRANCOIS GALLAIRE, LAILAI ZHU, EPFL - Lausanne — While the deformation regimes under flow of anuclear cells, like red blood cells, have been widely analyzed, the dynamics of nuclear cells are less explored. The objective of this work is to investigate the interplay between the stiff nucleus, modeled here as a rigid spherical particle and the surrounding deformable cell membrane, modeled for simplicity as an immiscible droplet, subjected to an external unbounded plane shear flow. A three-dimensional boundary integral implementation is developed to describe the interface-structure interaction characterized by two dimensionless numbers: the capillary number Ca , defined as the ratio of shear to capillary forces and the particle-droplet size ratio. For large Ca , i.e. very deformable droplets, the particle has a stable equilibrium position at the center of the droplet. However, for smaller Ca , both the plane symmetry and the time invariance are broken and the particle migrates to a closed orbit located off the symmetry plane, reaching a limit cycle. For even smaller capillary numbers, the time invariance is restored and the particle reaches a steady equilibrium position off the symmetry plane. This series of bifurcations is analyzed and possible physical mechanisms from which they originate are discussed.

¹Financial support by ERC grant SimCoMiCs 280117 is gratefully acknowledged.

11:58AM H1.00007 Soft Plumbing: Direct-Writing and Controllable Perfusion of Tubular Soft Materials, AXEL GUENTHER, PATRICIA OMORUWA, HAOTIAN CHEN, ARIANNA MCALLISTER, MARK JERONIMO, SHASHI MALLADI, NAVID HAKIMI, University of Toronto, LI CAO, University of California, Berkeley, ARUN RAMCHANDRAN, University of Toronto — Tubular and ductular structures are abundant in tissues in a wide variety of diameters, wall thicknesses, and compositions. In spite of their relevance to engineered tissues, organs-on-chips and soft robotics, the rapid and consistent preparation of tubular structures remains a challenge. Here, we use a microfabricated printhead to direct-write biopolymeric tubes with dimensional and compositional control. A biopolymer solution is introduced to the center layer of the printhead, and the confining fluids to the top and the bottom layers. The radially flowing biopolymer solution is sandwiched between confining solutions that initiate gelation, initially assuming the shape of a funnel until emerging through a cylindrical confinement as a continuous biopolymer tube. Tubular constructs of sodium alginate and collagen I were obtained with inner diameters (0.6-2.2mm) and wall thicknesses (0.1-0.4mm) in favorable agreement with predictions of analytical models. We obtained homogeneous tubes with smooth and buckled walls and heterotypic constructs that possessed compositions that vary along the tube circumference or radius. Ductular soft materials were reversibly hosted in 3D printed fluidic devices for the perfusion at well-defined transmural pressures to explore the rich variety of dynamical features associated with collapsible tubes that include buckling, complete collapse, and self-oscillation

12:11PM H1.00008 Fracture Phenomena in Foams: From Film Instability to Wave Propagation, SASCHA HILGENFELDT, University of Illinois at Urbana-Champaign, PETER STEWART, University of Glasgow — Injection of a gas into a gas/liquid foam is known to give rise to instability phenomena on a variety of time and length scales. Macroscopically, one observes a propagating gas-filled structure that can display properties of liquid finger propagation as well as of fracture in solids. The observation of both large-scale, finger-like cracks (without film breakage) and brittle cleavage phenomena (consisting of successive film ruptures) is explained through careful modeling of phenomena ranging from thin-film instabilities to friction between bubbles and confining plates. Whereas we use a network approach with full representation of the foam microstructure to model the cracks, we also derive a continuum limit description in order to investigate possible modes of wave propagation and their feedback on the fracture process.

12:24PM H1.00009 Instability in poroelastic media, SATYAJIT PRAMANIK, NORDITA, JOHN WETTLAUFER, NORDITA, Yale University, University of Oxford — Fluid flow in deformable porous materials, which play significant role in different biological and geological systems of wide range of scales, is a highly nonlinear problem. Feedback from the elastic deformation of the solid skeleton on the fluid flow and vice-versa gives rise to pattern formation in the porosity structure of the skeleton [1]. We view some of these patterns as instabilities of the coupled fluid-solid system. Due to highly nonlinear nature of the problem, very little has been understood about this instability. Here, we use a minimal poroelastic theory to understand the pattern formation in a fluid-saturated poroelastic material and discuss the similarities/differences with viscous fingering in non-deformable porous media.

[1] C. M. MacMinn, E. R. Dufresne, and J. Wetlaufer, "Fluid-driven deformation of a soft granular material," *Phys. Rev. X* 5, 011020 (2015).

12:37PM H1.00010 Elasticity-Driven Backflow of Fluid-Driven Cracks¹, CHING-YAO LAI, Princeton University, EMILIE DRESSAIRE, New York University, GUY RAMON, Israel Institute of Technology, HERBERT HUPPERT, HOWARD A. STONE, Princeton University — Fluid-driven cracks are generated by the injection of pressurized fluid into an elastic medium. Once the injection pressure is released, the crack closes up due to elasticity and the fluid in the crack drains out of the crack through an outlet, which we refer to as backflow. We experimentally study the effects of crack size, elasticity of the matrix, and fluid viscosity on the backflow dynamics. During backflow, the volume of liquid remaining in the crack as a function of time exhibits a transition from a fast decay at early times to a power law behavior at late times. Our results at late times can be explained by scaling arguments balancing elastic and viscous stresses in the crack. This work may relate to the environmental issue of flowback in hydraulic fracturing.

¹This work is supported by National Science Foundation via grant CBET-1509347 and partially supported by Andlinger Center for Energy and the Environment at Princeton University.

Monday, November 21, 2016 10:40AM - 12:50PM –
Session H2 Wind Turbines: Experiments A106 - Leonardo Chamorro, University of Illinois at Urbana-Champaign

10:40AM H2.00001 Fractional Flow Speedup from Porous Windbreaks for Enhanced Wind Turbine Power., NICOLAS TOBIN, ALI M. HAMED, LEONARDO P. CHAMORRO, University of Illinois at Urbana-Champaign — A wind tunnel experiment was performed to investigate the potential of porous windbreaks to increase the momentum into the swept area of a wind turbine, and thus power output. Planar particle-image velocimetry (PIV) along with linear perturbation theory is used quantify the effect of windbreak height in the changes in power output. Results show that far above the windbreak, perturbations reduce to potential flow, with a near-ground boundary condition defined by the recirculation zone behind the windbreak. Similarity in the windbreak flow is investigated and used to predict an increase in power which depends roughly linearly with windbreak height, which is corroborated by direct measurements of power from a model wind turbine. The flow field predicted by the linear theory is in broad agreement with the PIV measurements. By incorporating this result with a top-down wind turbine boundary layer approach which treats the windbreaks as additional roughness, it is found that there exists an inter-turbine spacing, on the order of 10 rotor diameters, for which windbreaks induce a net positive effect. This break-even spacing is dependent on surface roughness and the spanwise width of the windbreaks.

10:53AM H2.00002 ABSTRACT WITHDRAWN –

11:06AM H2.00003 Wake flow control using a dynamically controlled wind turbine¹, RICARDO CASTILLO, YEQIN WANG, SUHAS POL, ANDY SWIFT, FAZLE HUSSAIN, CARSTEN WESTERGAARD, Texas Tech University, TEXAS TECH UNIVERSITY TEAM — A wind tunnel based “Hyper Accelerated Wind Farm Kinematic-Control Simulator” (HAWKS) is being built at Texas Tech University to emulate controlled wind turbine flow physics. The HAWKS model turbine has pitch, yaw and speed control which is operated in real model time, similar to that of an equivalent full scale turbine. Also, similar to that of a full scale wind turbine, the controls are developed in a Matlab Simulink environment. The current diagnostic system consists of power, rotor position, rotor speed measurements and PIV wake characterization with four cameras. The setup allows up to 7D downstream of the rotor to be mapped. The purpose of HAWKS is to simulate control strategies at turnaround times much faster than CFD and full scale testing. The fundamental building blocks of the simulator have been tested, and demonstrate wake steering for both static and dynamic turbine actuation. Parameters which have been studied are yaw, rotor speed and combinations hereof. The measured wake deflections for static yaw cases are in agreement with previously reported research implying general applicability of the HAWKS platform for the purpose of manipulating the wake. In this presentation the general results will be introduced followed by an analysis of the wake turbulence and coherent structures when comparing static and dynamic flow cases. The outcome of such studies could ultimately support effective wind farm wake flow control strategies.

¹Texas Emerging Technology Fund (ETF)

11:19AM H2.00004 Effective solidity in vertical axis wind turbines, COLIN M. PARKER, MEGAN C. LEFTWICH, The George Washington University — The flow surrounding vertical axis wind turbines (VAWTs) is investigated using particle imaging velocimetry (PIV). This is done in a low-speed wind tunnel with a scale model that closely matches geometric and dynamic properties tip-speed ratio and Reynolds number of a full size turbine. Previous results have shown a strong dependence on the tip-speed ratio on the wake structure of the spinning turbine. However, it is not clear whether this is a speed or solidity effect. To determine this, we have measured the wakes of three turbines with different chord-to-diameter ratios, and a solid cylinder. The flow is visualized at the horizontal mid-plane as well as the vertical mid-plane behind the turbine. The results are both ensemble averaged and phase averaged by syncing the PIV system with the rotation of the turbine. By keeping the Reynolds number constant with both chord and diameter, we can determine how each effects the wake structure. As these parameters are varied there are distinct changes in the mean flow of the wake. Additionally, by looking at the vorticity in the phase averaged profiles we can see structural changes to the overall wake pattern.

11:32AM H2.00005 Low order physical models of vertical axis wind turbines¹, ANNA CRAIG, JOHN DABIRI, JEFFREY KOSEFF, Stanford University — In order to examine the ability of low-order physical models of vertical axis wind turbines to accurately reproduce key flow characteristics, experiments were conducted on rotating turbine models, rotating solid cylinders, and stationary porous flat plates (of both uniform and non-uniform porosities). From examination of the patterns of mean flow, the wake turbulence spectra, and several quantitative metrics, it was concluded that the rotating cylinders represent a reasonably accurate analog for the rotating turbines. In contrast, from examination of the patterns of mean flow, it was found that the porous flat plates represent only a limited analog for rotating turbines (for the parameters examined). These findings have implications for both laboratory experiments and numerical simulations, which have previously used analogous low order models in order to reduce experimental/computational costs.

¹NSF GRF and SGF to A.C; ONR N000141211047 and the Gordon and Betty Moore Foundation Grant GBMF2645 to J.D.; and the Bob and Norma Street Environmental Fluid Mechanics Laboratory at Stanford University

11:45AM H2.00006 Velocity Data in a Fully Developed Wind Turbine Array Boundary Layer, JOHN TURNER, MARTIN WOSNIK, University of New Hampshire — Results are reported from an experimental study of an array of porous disks simulating offshore wind turbines. The disks mimic power extraction of similarly scaled wind turbines via drag matching, and the array consists of 19x5 disks of 0.25 m diameter. The study was conducted in the UNH Flow Physics Facility (FPF), which has test section dimensions of 6.0 m wide, 2.7 m high and 72.0 m long. The FPF can achieve a boundary layer height on the order of 1 m at the entrance of the wind turbine array which puts the model turbines in the bottom third of the boundary layer, which is typical of field application. Careful consideration was given to an expanded uncertainty analysis, to determine possible measurements in this type of flow. For a given configuration (spacing, initial conditions, etc.), the velocity levels out and the wind farm approaches fully developed behavior, even within the maintained growth of the simulated atmospheric boundary layer. Benchmark pitot tube data was acquired in vertical profiles progressing streamwise behind the centered column at every row in the array.

11:58AM H2.00007 Wind tunnel measurements of wake structure and wind farm power for actuator disk model wind turbines in yaw¹, MICHAEL HOWLAND, Johns Hopkins University, JULIAAN BOSSUYT, KU Leuven, JUSTIN KANG, Johns Hopkins University, JOHAN MEYERS, KU Leuven, CHARLES MENEVEAU, Johns Hopkins University — Reducing wake losses in wind farms by deflecting the wakes through turbine yawing has been shown to be a feasible wind farm control approach. In this work, the deflection and morphology of wakes behind a wind turbine operating in yawed conditions are studied using wind tunnel experiments of a wind turbine modeled as a porous disk in a uniform inflow. First, by measuring velocity distributions at various downstream positions and comparing with prior studies, we confirm that the non-rotating wind turbine model in yaw generates realistic wake deflections. Second, we characterize the wake shape and make observations of what is termed a “curled wake,” displaying significant spanwise asymmetry. Through the use of a 100 porous disk micro-wind farm, total wind farm power output is studied for a variety of yaw configurations. Strain gages on the tower of the porous disk models are used to measure the thrust force as a substitute for turbine power. The frequency response of these measurements goes up to the natural frequency of the model and allows studying the spatiotemporal characteristics of the power output under the effects of yawing.

¹This work has been funded by the National Science Foundation (grants CBET-113380 and IIA-1243482, the WINDINSPIRE project). JB and JM are supported by ERC (ActiveWindFarms, grant no. 306471).

12:11PM H2.00008 Wind tunnel measurements of a large wind farm model approaching the infinite wind farm regime¹, JULIAAN BOSSUYT, KU Leuven and Johns Hopkins University, MICHAEL HOWLAND, CHARLES MENEVEAU, Johns Hopkins University, JOHAN MEYERS, KU Leuven — A scaled wind farm, with 100 porous disk models of wind turbines, is used to study the effect of wind farm layout on the wind farm power output and its variability, in a wind tunnel study. The wind farm consists of 20 rows and 5 columns. The porous disk models have a diameter of $0.03m$ and are instrumented with strain gages to measure the thrust force, as a surrogate for wind turbine power output. The frequency response of the measurements goes up to the natural frequency of the models and allows studying the spatio-temporal characteristics of the power output for different layouts. A variety of layouts are considered by shifting the individual rows in the spanwise direction. The reference layout has a regular streamwise spacing of $S_x/D = 7$ and a spanwise spacing of $S_y/D = 5$. The parameter space is further expanded by considering layouts with an uneven streamwise spacing: $S_x/D = 3.5 \& 10.5$ and $S_x/D = 1.5 \& 12.5$. We study how the mean row power changes as a function of wind farm layout and investigate the appearance of an asymptotic limiting behavior as previously described in the literature by application of the top-down model for the spatially averaged wind farm - boundary layer interaction.

¹Work supported by ERC (grant no. 306471, the ActiveWindFarms project) and by NSF (OISE-1243482, the WINDINSPIRE project).

12:24PM H2.00009 ABSTRACT WITHDRAWN —

12:37PM H2.00010 Achieving Full Dynamic Similarity with Small-Scale Wind Turbine Models¹, MARK MILLER, JANIK KIEFER, Princeton University, CARSTEN WESTERGAARD, Texas Tech University, MARCUS HULTMARK, Princeton University — Power and thrust data as a function of Reynolds number and Tip Speed Ratio are presented at conditions matching those of a full scale turbine. Such data has traditionally been very difficult to acquire due to the large length-scales of wind turbines, and the limited size of conventional wind tunnels. Ongoing work at Princeton University employs a novel, high-pressure wind tunnel (up to 220 atmospheres of static pressure) which uses air as the working fluid. This facility allows adjustment of the Reynolds number (via the fluid density) independent of the Tip Speed Ratio, up to a Reynolds number (based on chord and velocity at the tip) of over 3 million. Achieving dynamic similarity using this approach implies very high power and thrust loading, which results in mechanical loads greater than 200 times those experienced by a similarly sized model in a conventional wind tunnel. In order to accurately report the power coefficients, a series of tests were carried out on a specially designed model turbine drive-train using an external testing bench to replicate tunnel loading. An accurate map of the drive-train performance at various operating conditions was determined. Finally, subsequent corrections to the power coefficient are discussed in detail.

¹Supported by: National Science Foundation grant CBET-1435254 (program director Gregory Rorrer)

Monday, November 21, 2016 10:40AM - 12:50PM — Session H3 Vortex Dynamics: Vortex Rings B110-111 - C.H.K Williamson, Cornell University

10:40AM H3.00001 Impingement of a Vortex Pair on a Wavy Wall¹, SARAH MORRIS, C.H.K. WILLIAMSON, Cornell University — In this research we examine the impingement of a vortex pair onto a wavy wall. Isolated vortex pairs, not in ground effect, can become unstable to short-wave (Widnall, 1974) or long-wave instability (Crow, 1970). When a vortex pair approaches a ground plane, the boundary layer that forms on the surface separates, generating secondary vorticity and causing the primary pair to 'rebound'. When a vortex pair with the long-wave instability interacts with a flat boundary, the topology of the pair changes, resulting in rebounding vortical structures whose form is dependent on the extent of the instability prior to wall interaction (Asselin & Williamson, 2013, 2016). By using PIV and LIF to consider the "complementary" experiment, a straight vortex pair encountering a wavy wall (rather than a wavy pair impinging on a flat wall), certain critical features of the two flows are found to be similar. The 2D vortex pair first interacts with the "hills" of the boundary, triggering accelerated vorticity cancellation in this area compared to the corresponding "valley" regions. An axial pressure gradient forms between the two regions, giving rise to strong axial flow. This leads to the interaction of primary and secondary vortices in the valleys, wherein reconnection results in "rebounding" vortex rings, two per fundamental wavelength. The resulting flowfield forms distinctly different vortex structures than are classically found for 2D vortex pair wall impingement or for the long-wave instability out of ground effect.

¹This work was supported by the Office of Naval Research Award No. N00014-12-421-0712, monitored by Dr. Ron Joslin.

10:53AM H3.00002 Formation number for vortex dipoles¹, VAHID SADRI, PAUL S KRUEGER, SMU — This investigation considers the axisymmetric formation of two opposite sign concentric vortex rings from jet ejection between concentric cylinders. This arrangement is similar to planar flow in that the vortex rings will travel together when the gap between the cylinders is small, similar to a vortex dipole, but it has the advantage that the vortex motion is less constrained than the planar case (vortex stretching and vortex line curvature is allowed). The flow was simulated numerically at a jet Reynolds number of 1,000 (based on ΔR and the jet velocity), jet pulse length-to-gap ratio ($L/\Delta R$) in the range 10–20, and gap-to-outer radius ratio ($\Delta R/R_o$) in the range 0.01–0.1. Small gap ratios were chosen for comparison with 2D results. In contrast with 2D results, the closely paired vortices in this study exhibited pinch-off from the generating flow and finite formation numbers. The more complex flow evolution afforded by the axisymmetric model and its influence on the pinch-off process will be discussed.

¹This material is based on work supported by the National Science Foundation under Grant No. 1133876 and SMU. This supports are gratefully acknowledged.

11:06AM H3.00003 Vortex ring formation in starting forced plumes with negative and positive buoyancy¹, LEI GAO, Sichuan University, SIMON CHING-MAN YU, Singapore Institute of Technology — The limiting process of vortex formation in starting forced plumes, with Richardson number in the range of $-0.06 \leq Ri \leq 0.06$, was studied numerically. As Ri increases, three regimes can be identified in terms of the vortex interaction patterns, i.e., the weak-interaction regime ($-0.06 < Ri < -0.02$), the transition regime ($-0.02 \leq Ri < 0$) and the strong-interaction regime ($0 \leq Ri < 0.06$). The numerical results show that the variation trends of formation number and separation number against Ri change near the critical value of -0.02 . In the weak-interaction regime, both formation number and separation number increase rapidly against Ri . In the transition and strong-interaction regimes alike, the formation number increases at a much slower rate, while the separation number declines dramatically as Ri increases. A qualitative explanation on the variation patterns of formation number and separation number is proposed based on the buoyancy effects on the dynamic properties of the leading vortex ring and the vortex interaction patterns.

¹We acknowledge the financial support provided by National Natural Science Foundation of China (Grant No. 11502153)

11:19AM H3.00004 Drift due to viscous vortex rings¹ , THOMAS MORRELL, SAVERIO SPAGNOLIE,

JEAN-LUC THIFFEAULT, Department of Mathematics, University of Wisconsin - Madison — Biomixing is the study of fluid mixing due to swimming organisms. While large organisms typically produce turbulent flows in their wake, small organisms produce less turbulent wakes; the main mechanism of mixing is the induced net particle displacement (drift). Several experiments have examined this drift for small jellyfish, which produce vortex rings that trap and transport a fair amount of fluid. Inviscid theory implies infinite particle displacements for the trapped fluid, so the effect of viscosity must be included to understand the damping of real vortex motion. We use a model viscous vortex ring to compute particle displacements and other relevant quantities, such as the integrated moments of the displacement. Fluid entrainment at the tail end of a growing vortex 'envelope' is found to play an important role in the total fluid transport and drift.

¹Partially supported by NSF grant DMS-1109315

11:32AM H3.00005 Buoyant vortex rings and knots with thin core , CHING CHANG, STEFAN LLEWELLYN SMITH, University of California San Diego — One challenge of studying the motion of vortex filaments arises from the singular nature of the Biot-Savart integral. We employ the momentum balance investigated by Moore and Saffman for thin-core vortex filaments to obtain the self-induced velocity of filaments, rings and knots. A key feature of the approach is the possibility of incorporating buoyancy forces. The numerical scheme used is discussed and compared to previous analytical and numerical results in the literature. The effect of geometry and buoyancy on the motion of such vortices is examined.

11:45AM H3.00006 Experimental study of vortex ring interactions with a flexible beam; investigating the role of viscous effects.¹ , ALIREZA PIRNIA, Clarkson University, JIACHENG HU, SEAN PETERSON, University of Waterloo, BYRON ERATH, Clarkson University — Energy can be extracted from flow instabilities in the environment for powering low consumption devices. When vortices pass tangentially over a flexible beam the lower pressure in the vortex core causes the beam to deflect, and induces sustained oscillations which can be converted into energy via piezoelectric materials. The beam dynamics can be parameterized according to the beam properties (nondimensional mass and stiffness ratios) as well as the vortex properties (size, vortex circulation strength and advection velocity). Recently, inviscid models have been developed to solve this fluid-structure interaction problem but they do not capture viscous interactions; features that become more prominent when the beam is positioned close to the vortex core. In this study the interaction of a vortex ring passing tangentially over a flexible beam as a function of circulation strength, beam properties, and offset distance are investigated to identify how viscous interactions influence the energy exchange process. Particle image velocimetry is acquired in tandem with the beam dynamics. The velocity and pressure fields, and transient beam dynamics are compared and contrasted with an inviscid model to identify the role of viscous interactions.

¹This work was supported by the National Science Foundation Grant CBET 1511761

11:58AM H3.00007 Direct numerical simulations of vortex ring collisions , RODOLFO OSTILLA MONICO, Harvard University, ALAIN PUMIR, ENS Lyon, MICHAEL BRENNER, Harvard University — We numerically simulate the ring vortex collision experiment of Lim and Nickels (Nature, 357:225-227, 1992) in an attempt to understand the rapid formation of very fine scale turbulence (or 'smoke') from relatively smooth initial conditions. Reynolds numbers of up to $Re = \Gamma/\nu = 7500$, where Γ is the vortex ring circulation and ν the kinematic viscosity of the fluid are reached, which coincide with the highest Reynolds number case of the experiments. Different perturbations to the ring vortex are added, and their effect on the generation and amplification of turbulence is quantified. The underlying dynamics of the vortex core is analyzed, and compared to the dynamics arising from a simple Biot-Savart filament model for the core.

12:11PM H3.00008 Numerical study of asymmetrical modes in a vortex ring impacting a conical surface¹ , JOSE ANTONIO TREJO GUTIERREZ, ERICK JAVIER LOPEZ SANCHEZ, SERGIO HERNANDEZ ZAPATA, GERARDO RUIZ CHAVARRIA, Facultad de Ciencias, Universidad Nacional Autonoma de Mexico — In this work we investigate the impact of an annular vortex on a conical surface when their symmetry axes are parallel but they do not coincide. For this purpose we solve the Navier-Stokes and continuity equations in cylindrical coordinates. We use a finite difference scheme for r and z coordinates whereas for the angular coordinate we use a Fourier spectral method. We study the development of asymmetrical modes when the vortex approaches the inner surface of the cone. The presence of the vortex ring induces the formation of a boundary layer which detaches and leads to the formation of a secondary vortex of opposite sign which moves away the cone. This secondary vortex also exhibits asymmetrical modes, which are attenuated as it moves. We present some results as the trajectories of the primary and the secondary vortices, their circulations as a function of time, the development of asymmetrical modes and the dependence of these properties on the Reynolds number and the distance between both symmetry axes. Finally we made a comparison of primary and secondary vortices with a free vortex.

¹Authors acknowledge DGAPA-UNAM by support under project IN115315 Ondas y estructuras coherentes en dinamica de fluidos.

12:24PM H3.00009 Gap vortex streets and turbulence in time-dependent streams¹ , DAN DUONG, STAVROS TAVOURARIS, University of Ottawa — Gap vortex streets form in axial flows in highly eccentric annular channels, tightly packed rod bundles and other channels having narrow gap regions flanked by wider ones. The characteristics of these vortices and the flow and turbulence distributions in some of these channels have in the past documented for steady streams; in particular, the vortex generation frequency was found to be proportional to the bulk Reynolds number. The present study extends these findings to both accelerating and decelerating air flows in a large-scale rod bundle, configured as a wind tunnel with a by-pass branch equipped with a controlled movable flap just downstream of the blower. Time-dependent statistical properties in a gap and a subchannel centre were determined by phase-averaging velocity measurements collected with hot-wire anemometers and the time history of the phase-averaged vortex street frequency was determined with the use of a wavelet transform. Contrary to expectations, the results show that deviations of the vortex frequency and other flow characteristics from the corresponding values in steady flows at the same bulk Reynolds number were significant during acceleration and much less so during deceleration.

¹Supported by the Natural Sciences and Engineering Research Council of Canada and the Canadian Nuclear Laboratories

12:37PM H3.00010 Stratified Vortex Rings: Visualization of the Density Evolution¹

, JASON OLSTHOORN, STUART DALZIEL, University of Cambridge — The study of vortex-ring induced stratified mixing has played a key role in understanding stratified turbulent mixing. In this study, we present an experimental investigation of the mechanical evolution and the stratification-modified three-dimensional instability of these vortex rings. Using a stereoscopic particle image velocimetry setup, we reconstruct a full, three-dimensional, time-resolved velocity field of the interaction of a vortex ring with a stratified interface. This reconstruction agrees with previous two-dimensional studies, while capturing the three-dimensional instabilities of the dynamical evolution. The stratified three-dimensional instability of a vortex ring is similar to the unstratified instability, but here the instability occurs much earlier. Through the use of numerical integration, we use the experimentally determined velocity field to simulate the kinematic evolution of the density stratification. This technique allows us to evaluate the vertical buoyancy flux throughout the vortex-ring interaction, providing a quantitative explanation for the interface sharpening observed within the experiments. Understanding the sharpening mechanism in the context of a vortex ring has direct relevance to understanding the layer formation found in stratified turbulence

¹NSERC, EPSRC

Monday, November 21, 2016 10:40AM - 12:50PM – Session H4 Acoustics IV: General B112 - Carlo Scalo, Purdue University

10:40AM H4.00001 The acoustics of short circular holes with finite expansion ratio¹

, DONG YANG, AIMEE MORGANS, Department of Mechanical Engineering, Imperial College London — The acoustic response of a circular hole with mean flow passing through it is highly relevant to Helmholtz resonators, fuel injectors, perforated liners, perforated plates and many other engineering applications. Analytical models for the acoustic response of these holes often ignore the impact of a finite expansion ratio either side, or account for it simply by adding an end mass inertial correction derived from the no mean flow assumption. The vortex-sound interaction within a short hole has been recently shown to strongly affect the acoustic response in the low frequency region. The present study uses an analytical model based on the Greens function method to investigate how the expansion ratios either side of a short hole affect the vortex-sound interaction within it something neglected by previous models. This model is then incorporated into a Helmholtz resonator model, allowing us to consider the effect of a finite neck-to-cavity expansion ratio and the vortex-sound interaction within the finite length neck. Large resistance and acoustic energy absorption performance variations are seen even for small changes in the resonator neck length. Reducing the neck-to-cavity expansion ratio is found to decrease the resonators sound absorption when the expansion ratio is low.

¹CSC-Imperial PhD Scholarship; ERC Starting Grant, ACOULOMODE (2013-18)

10:53AM H4.00002 Thermo-mechanical concepts applied to modeling liquid propellant rocket engine stability¹ , DAVID R KASSOY, University of Colorado, Boulder (retired), ADAM NORRIS, University of Colorado, Boulder, Applied Mathematics Department — The response of a gas to transient, spatially distributed energy addition can be quantified mathematically using thermo-mechanical concepts available in the literature. The modeling demonstrates that the ratio of the energy addition time scale to the acoustic time scale of the affected volume, and the quantity of energy added to that volume during the former determine the whether the responses to heating can be described as occurring at nearly constant volume, fully compressible or nearly constant pressure. Each of these categories is characterized by significantly different mechanical responses. Application to idealized configurations of liquid propellant rocket engines provides an opportunity to identify physical conditions compatible with gasdynamic disturbances that are sources of engine instability.

¹Air Force Office of Scientific Research

11:06AM H4.00003 Acoustic impedance characterization via numerical resolution of the inverse Helmholtz problem , CARLO SCALO, DANISH PATEL, PRATEEK GUPTA, Purdue University — Impedance boundary conditions (IBCs) regulate the relative phasing and amplitudes of pressure and velocity fluctuations and, therefore, the acoustic energy flux. We present a numerical method to determine the acoustic impedance at the surface of an arbitrarily shaped cavity as seen by a generically oriented incident external harmonic planar wave. The proposed method (conceptually) inverts the usual eigenvalue-solving procedure underlying Helmholtz solvers: the impedance at one or multiple (but not all) boundaries is an output of the calculation and is obtained via implicit reconstruction the linear acoustic waveform at the frequency of the incident wave. The linearized governing equations are discretized via a mixed finite-difference/finite-volume approach and are closed with a generalized equation of state. Results are validated against quasi one-dimensional cases derived via direct application of Rott's linear thermoacoustic theory and by comparison against fully compressible Navier-Stokes simulations. This work is motivated by the need to develop a comprehensive suite of predictive tools capable of performing high-fidelity simulations of compressible boundary layers over assigned IBCs, accurately representing the acoustic response of arbitrarily shaped porous cavities.

11:19AM H4.00004 Characterization of Atmospheric Infrasound for Improved Weather Monitoring¹ , ARNESHA THREATT, BRIAN ELBING, Oklahoma State University — Collaboration Leading Operational UAS Development for Meteorology and Atmospheric Physics (CLOUD MAP) is a multi-university collaboration focused on development and implementation of unmanned aircraft systems (UAS) and integration with sensors for atmospheric measurements. A primary objective for this project is to create and demonstrate UAS capabilities needed to support UAS operating in extreme conditions, such as a tornado producing storm system. These storm systems emit infrasound (acoustic signals below human hearing, <20 Hz) up to 2 hours before tornadogenesis. Due to an acoustic ceiling and weak atmospheric absorption, infrasound can be detected from distances in excess of 300 miles. Thus infrasound could be used for long-range, passive monitoring and detection of tornadogenesis as well as directing UAS resources to high-decision-value-information. To achieve this the infrasonic signals with and without severe storms must be understood. This presentation will report findings from the first CLOUD MAP field demonstration, which acquired infrasonic signals while simultaneously sampling the atmosphere with UAS. Infrasonic spectra will be shown from a typical calm day, a continuous source (pulsed gas-combustion torch), singular events, and UAS flights as well as localization results from a controlled source and multiple microphones.

¹This work was supported by NSF Grant 1539070: CLOUD MAP Collaboration Leading Operational UAS Development for Meteorology and Atmospheric Physics

11:32AM H4.00005 Laboratory measurements of the effect of internal waves on sound propagation, LIKUN ZHANG, University of Texas at Austin and University of Mississippi, HARRY L. SWINNEY, University of Texas at Austin, YING-TSONG LIN, Woods Hole Oceanographic Institution — The fidelity of acoustic signals used in communication and imaging in the oceans is limited by density fluctuations arising from many sources, particularly from internal waves. We present results from laboratory experiments on sound propagation through an internal wave field produced by a wave generator consisting of multiple oscillating plates. The fluid density as a function of height is measured and used to determine the sound speed as a function of the height. Sound pulses from a transducer propagate through the fluctuating stratified density field and are detected to determine sound refraction, pulse arrival time, and sound signal distortion. The results are compared with sound ray model and numerical models of underwater sound propagation. The laboratory experiments can explore the parameter dependence by varying the fluid density profile, the sound pulse signal, and the internal wave amplitude and frequency. The results lead to a better understanding of sound propagation through and scattered by internal waves.

11:45AM H4.00006 Low order models for uncertainty quantification in acoustic propagation problems, CHRISTOPHE MILLET, CEA, DAM, DIF — Long-range sound propagation problems are characterized by both a large number of length scales and a large number of normal modes. In the atmosphere, these modes are confined within waveguides causing the sound to propagate through multiple paths to the receiver. For uncertain atmospheres, the modes are described as random variables. Concise mathematical models and analysis reveal fundamental limitations in classical projection techniques due to different manifestations of the fact that modes that carry small variance can have important effects on the large variance modes. In the present study, we propose a systematic strategy for obtaining statistically accurate low order models. The normal modes are sorted in decreasing Sobol indices using asymptotic expansions, and the relevant modes are extracted using a modified iterative Krylov-based method. The statistics of acoustic signals are computed by decomposing the original pulse into a truncated sum of modal pulses that can be described by a stationary phase method. As the low-order acoustic model preserves the overall structure of waveforms under perturbations of the atmosphere, it can be applied to uncertainty quantification. The result of this study is a new algorithm which applies on the entire phase space of acoustic fields.

11:58AM H4.00007 Numerical study on nonlinear acoustic pulse propagation for parametric array with different fluid layer, KEI FUJISAWA, AKIRA ASADA, The University of Tokyo — We present numerical results of nonlinear acoustic pulse propagation emitted from parametric array utilizing different fluid layer in water environment. A numerical simulation was carried out using the compressible forms fluid dynamic equations in cylindrical coordinate system for the parametric sound propagation in water with and without different fluid layer composed of ethanol. The numerical results indicated that the asymmetry of the acoustic pulse increased with different fluid layer due to the combined effect of diffraction and nonlinearity in the propagation.

12:11PM H4.00008 Modes of targets in water excited and identified using radiation pressure of modulated focused ultrasound¹, TIMOTHY DANIEL, AUBERRY FORTUNER, Washington State Univ, AHMAD ABAWI, HLS Research, IVARS KIRSTEINS, NUWC, PHILIP MARSTON, Washington State Univ — The modulated radiation pressure (MRP) of ultrasound has been widely used to selectively excite low frequency modes of fluid objects [1,2]. We previously used MRP to excite less compliant metallic object in water including the low frequency modes of a circular metal plate in water. A larger focused ultrasonic transducer allows us to drive modes of larger more-realistic targets. In our experiments solid targets are suspended by strings or supported on sand and the modulated ultrasound is focused on the target's surface. Target sound emissions were recorded and a laser vibrometer was used to measure the surface velocity of the target to give the magnitude of the target response. The source transducer was driven with a doublesideband suppressed carrier voltage as in [1]. By varying the modulation frequency and monitoring target response, resonant frequencies can be measured and compared to finite element models. We also demonstrate the radiation torque of a focused first-order acoustic vortex beam associated with power absorption in the Stokes layer adjacent to a sphere. [1] P. L. Marston and R. E. Apfel, J. Acoust. Soc. Am. 67, 27–37 (1980). [2] S. F. Morse, D. B. Thiessen, and P. L. Marston, Phys. Fluids 8, 35 (1996).

¹Funded by ONR

12:24PM H4.00009 Towards a Coupled Vortex Particle and Acoustic Boundary Element Solver to Predict the Noise Production of Bio-Inspired Propulsion, NATHAN WAGENHOFFER, KEITH MOORED, JUSTIN JAWORSKI, Lehigh Univ — The design of quiet and efficient bio-inspired propulsive concepts requires a rapid, unified computational framework that integrates the coupled fluid dynamics with the noise generation. Such a framework is developed where the fluid motion is modeled with a two-dimensional unsteady boundary element method that includes a vortex-particle wake. The unsteady surface forces from the potential flow solver are then passed to an acoustic boundary element solver to predict the radiated sound in low-Mach-number flows. The use of the boundary element method for both the hydrodynamic and acoustic solvers permits dramatic computational acceleration by application of the fast multipole method. The reduced order of calculations due to the fast multipole method allows for greater spatial resolution of the vortical wake per unit of computational time. The coupled flow-acoustic solver is validated against canonical vortex-sound problems. The capability of the coupled solver is demonstrated by analyzing the performance and noise production of an isolated bio-inspired swimmer and of tandem swimmers.

12:37PM H4.00010 Rotation of a metal gear disk in an ultrasonic levitator, PABLO L RENDON, RICARDO R BOULLOSA, CCADET, Universidad Nacional Autonoma de Mexico, LAURA SALAZAR, Facultad de Ciencias, Universidad Nacional Autonoma de Mexico — The phenomenon known as acoustic radiation pressure is well-known to be associated with the time-averaged momentum flux of an acoustic wave, and precisely because it is a time-averaged effect, it is relatively easy to observe experimentally. An ultrasonic levitator makes use of this effect to levitate small particles. Although it is a less-well studied effect, the transfer of angular momentum using acoustic waves in air or liquids has nonetheless been the subject of some recent studies. This transfer depends on the scattering and absorbing properties of the object and is achieved, typically, through the generation of acoustic vortex beams. In the present study, we examine the manner in which the acoustic standing wave located between two disks of an ultrasonic levitator in air may transfer angular momentum to objects with different shapes. In this case, a non-spherical object is subjected to, in addition to the radiation force, a torque which induces rotation. Analytical solutions for the acoustic force and torque are available, but limited to a few simple cases. In general, a finite element model must be used to obtain solutions. Thus, we develop and validate a finite element simulation in order to calculate directly the torque and radiation force.

Monday, November 21, 2016 10:40AM - 12:50PM – Session H5 Compressible Flow: Particle-shock Interactions and Explosions B113 - Elaine Oran, University of Maryland

10:40AM H5.00001 The role of granular shocks in dust-layer dispersal by shock waves, RYAN HOUIM, ORLANDO UGARTE, ELAINE ORAN, University of Maryland — Exactly how dust-layers are lifted and dispersed by shocks has been a longstanding question in compressible multiphase flow. Understanding the mechanism for this, however, is extremely important for early control of dust explosions. We address this problem by numerically solving a set of equations that couples a fully compressible representation of a gas with a kinetic-theory model for a granular medium (see Journal of Fluid Mechanics, (2016) 789:166-220) to simulate a shock propagating along the surface of a dust layer. The results show that the majority of the dispersed dust is lifted by hydrodynamic shear directly behind the shock wave. Simultaneously, large forces are produced behind the shock that compact the dust layer and create a granular shock. The effects from this granular shock on the surface of the dust layer destabilize the gas-dust boundary layer, which, in turn, enhances turbulence and the rate of dust dispersal.

10:53AM H5.00002 Simulations of Shock Wave Interaction with a Particle Cloud¹, RAHUL KONERU, CCMT-University of Florida, BERTRAND ROLLIN, Embry-Riddle Aeronautical University, FREDERICK OUELLET, SUBRAMANIAN ANNAMALAI, S. 'BALA' BALACHANDAR, CCMT-University of Florida — Simulations of a shock wave interacting with a cloud of particles are performed in an attempt to understand similar phenomena observed in dispersal of solid particles under such extreme environment as an explosion. We conduct numerical experiments in which a particle curtain fills only 87% of the shock tube from bottom to top. As such, the particle curtain upon interaction with the shock wave is expected to experience Kelvin-Helmholtz (KH) and Richtmyer-Meshkov (RM) instabilities. In this study, the initial volume fraction profile matches with that of Sandia Multiphase Shock Tube experiments, and the shock Mach number is limited to M=1.66. In these simulations we use a Eulerian-Lagrangian approach along with state-of-the-art point-particle force and heat transfer models. Measurements of particle dispersion are made at different initial volume fractions of the particle cloud. A detailed analysis of the evolution of the particle curtain with respect to the initial conditions is presented.

¹This 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.

11:06AM H5.00003 Shock Interaction with Random Spherical Particle Beds¹, CHRIS NEAL, YASH MEHTA, Univ of Florida - Gainesville, KAMBIZ SALARI, Lawrence Livermore National Lab, THOMAS L. JACKSON, S. "BALA" BALACHANDAR, SIDDHARTH THAKUR, Univ of Florida - Gainesville — In this talk we present results on fully resolved simulations of shock interaction with randomly distributed bed of particles. Multiple simulations were carried out by varying the number of particles to isolate the effect of volume fraction. Major focus of these simulations was to understand 1) the effect of the shockwave and volume fraction on the forces experienced by the particles, 2) the effect of particles on the shock wave, and 3) fluid mediated particle-particle interactions. Peak drag force for particles at different volume fractions show a downward trend as the depth of the bed increased. This can be attributed to dissipation of energy as the shockwave travels through the bed of particles. One of the fascinating observations from these simulations was the fluctuations in different quantities due to presence of multiple particles and their random distribution. These are large simulations with hundreds of particles resulting in large amount of data. We present statistical analysis of the data and make relevant observations. Average pressure in the computational domain is computed to characterize the strengths of the reflected and transmitted waves. We also present flow field contour plots to support our observations.

¹U.S. Department of Energy, National Nuclear Security Administration, Advanced Simulation and Computing Program, as a Cooperative Agreement under the Predictive Science Academic Alliance Program, under Contract No. DE-NA0002378

11:19AM H5.00004 Shock Particle Interaction - Fully Resolved Simulations and Modeling¹, YASH MEHTA, CHRIS NEAL, THOMAS L. JACKSON, S. "BALA" BALACHANDAR, SIDDHARTH THAKUR, Univ of Florida - Gainesville — Currently there is a substantial lack of fully resolved data for shock interacting with multiple particles. In this talk we will fill this gap by presenting results of shock interaction with 1-D array and 3-D structured arrays of particles. Objectives of performing fully resolved simulations of shock propagation through packs of multiple particles are twofold, 1) To understand the complicated physical phenomena occurring during shock particle interaction, and 2) To translate the knowledge from microscale simulations in building next generation point-particle models for macroscale simulations that can better predict the motion (forces) and heat transfer for particles. We compare results from multiple particle simulations against the single particle simulations and make relevant observations. The drag history and flow field for multiple particle simulations are markedly different from those of single particle simulations, highlighting the effect of neighboring particles. We propose new models which capture this effect of neighboring particles. These models are called Pair-wise Interaction Extended Point Particle models (PIEP). Effect of multiple neighboring particles is broken down into pair-wise interactions, and these pair-wise interactions are superimposed to get the final model

¹U.S. Department of Energy, National Nuclear Security Administration, Advanced Simulation and Computing Program, as a Cooperative Agreement under the Predictive Science Academic Alliance Program, under Contract No. DE-NA0002378

11:32AM H5.00005 The Utility of Gas Gun Experiments in Developing Equations of State, EMILY PITTMAN, CARL HAGELBERG, SCOTT RAMSEY, Los Alamos National Laboratory — Gas gun experiments have the potential to investigate material properties in various well defined shock conditions, making them a valuable research tool for the development of equations of state (EOS) and material response under shock loading. Gas guns have the ability to create shocks for loading to pressures ranging from MPa to GPa. A variety of diagnostics techniques can be used to gather data from gas gun experiments; resulting data from these experiments is applicable to many fields of study. The focus of this set of experiments is the development of data on the Hugoniot for the overdriven products EOS of PBX 9501 to extend data from which current computational EOS models draw. This series of shots was conducted by M-9 using the two-stage gas-guns at LANL and aimed to gather data within the 30-120 GPa pressure regime. The experiment was replicated using FLAG, a Langrangian multiphysics code, using a one-dimensional setup which employs the Wescott Stewart Davis (WSD) reactive burn model. Prior to this series, data did not extend into this higher range, so the new data allowed for the model to be re-evaluated. A comparison of the results to the experimental data reveals that the model is a good fit to the data below 40 GPa. However, the model did not fall within the error bars for pressures above this region. This is an indication that the material models or burn model could be modified to better match the data.

11:45AM H5.00006 A Scale Invariant Equation of State for Gruneisen Materials, EMMA SCHMIDT, JENNIFER LILIEHOLM, SCOTT RAMSEY, ZACHARY BOYD, Los Alamos National Laboratory — Scale-invariant equations of state are required for the existence of the Noh, Sedov, and Guderley compressible flow similarity solutions in the general case. All of these problems are self-similar and their solutions are independent of space, time, and the hydrodynamic state of the system. This work establishes a new equation of state with hydrodynamic scaling properties that may be used to approximate Gruneisen materials. The Gruneisen equation of state is relevant for materials whose atoms are limited to small vibrations; however the Gruneisen EOS is shown to lack the form necessary to yield a scaling solution in the general case. A Virial EOS with coefficients reminiscent of Gruneisen materials is proposed and derived, and shown to possess the desired hydrodynamic scaling properties. The divergence of the approximation from the true Gruneisen EOS under strong shock conditions is discussed.

11:58AM H5.00007 Analysis of Computational Models of Shaped Charges for Jet Formation and Penetration¹, JONAH HAEFNER, JIM FERGUSON, Los Alamos National Laboratory — Shaped charges came into use during the Second World War demonstrating the immense penetration power of explosively formed projectiles and since has become a tool used by nearly every nation in the world. Penetration is critically dependent on how the metal liner is collapsed into a jet. The theory of jet formation has been studied in depth since the late 1940s, based on simple models that neglect the strength and compressibility of the metal liner. Although attempts have been made to improve these models, simplifying assumptions limit the understanding of how the material properties affect the jet formation. With a wide range of material and strength models available for simulation, a validation study was necessary to guide code users in choosing models for shaped charge simulations. Using PAGOSA, a finite-volume Eulerian hydrocode designed to model hypervelocity materials and strong shock waves developed by Los Alamos National Laboratory, and experimental data, we investigated the effects of various equations of state and material strength models on jet formation and penetration of a steel target. Comparing PAGOSA simulations against modern experimental data, we analyzed the strengths and weaknesses of available computational models. LA-UR-16-25639

¹Los Alamos National Laboratory

12:11PM H5.00008 Physical and Electrical Measurements of Different Metals used in Exploding Wires, COLE VALANCIUS, JOE BAINBRIDGE, CODY LOVE, DUANE RICHARDSON, Sandia Natl Labs — Implementation of the energy density metric has made it possible to better understand the physics of exploding wires. When applying the energy density metric to exploding wire experiments in a porous material bed, results suggest a link between characteristics of wire materials (e.g. their electrical properties during burst and the physical work done by the bursting wire). Previous work has focused on qualitative comparisons of current and voltage waveforms and the qualitative comparison of Schlieren images of wire shocks in air. In these experiments, the wires were all buried in a porous media allowing the simultaneous capture of accurate current and voltage to observe the energy density at burst, while simultaneously observing the amount of time the wire took to compress the porous media to a 1 mm deep crater. Observing the physical compression of the porous bed in time in concert with the energy density allows a link to be established between the measured electrical signals and the physical work done by the exploding wire. This research allows a more quantitative link to be established between the electrical energy and the physical energy expended by an exploding wire, allowing for the development of more accurate models and a better understanding of exploding wire physics.

12:24PM H5.00009 Jetting instability of a shocked cylindrical shell of solid particles, ANDREAS NYGAARD OSNES, University Graduate Center at the University of Oslo, MAGNUS VARTDAL, Norwegian Defence Research Establishment (FFI), BJOERN ANDERS PETTERSSON REIF, University Graduate Center at the University of Oslo — Explosive dispersion of cylindrical or spherical shells of liquid or solid particles features jetting phenomena and the underlying mechanism generating these is not well understood. Previous studies have indicated that the jets originate from structures formed during a very short time interval after the passing of the shock wave. This study utilizes LES simulations to examine the jetting phenomena occurring during impulsive dispersal of solid particles. The problem considered is the dispersion of a ring of particles inside a Hele-Shaw cell by a shock-wave, as previously examined experimentally by [Rodriguez, V., Saurel, R., Jourdan, G., & Houas, L. (2013). Solid-particle jet formation under shock-wave acceleration. Physical Review E, 88(6), 063011]. Various degrees of coupling between the flow field and the solid particles are tested, and results are compared. The simpler case of replacing the shock-tube by a constant high pressure and high density region inside the solid particle ring is also examined and compared to the original problem.

12:37PM H5.00010 Analysis of Xrage and Flag High Explosive Burn Models with PBX 9404 Cylinder Tests, DANIELLE HARRIER, JULIANNA FESSENDEN, SCOTT RAMSEY, Los Alamos National Laboratory — High explosives are energetic materials that release their chemical energy in a short interval of time. They are able to generate extreme heat and pressure by a shock driven chemical decomposition reaction, which makes them valuable tools that must be understood. This study investigated the accuracy and performance of two Los Alamos National Laboratory hydrodynamic codes, which are used to determine the behavior of explosives within a variety of systems: xRAGE which utilizes an Eulerian mesh, and FLAG with utilizes a Lagrangian mesh. Various programmed and reactive burn models within both codes were tested, using a copper cylinder expansion test. The test was based off of a recent experimental setup which contained the plastic bonded explosive PBX 9404. Detonation velocity versus time curves for this explosive were obtained from the experimental velocity data collected using Photon Doppler Velocimetry (PDV). The modeled results from each of the burn models tested were then compared to one another and to the experimental results using the Jones-Wilkins-Lee (JWL) equation of state parameters that were determined and adjusted from the experimental tests. This study is important to validate the accuracy of our high explosive burn models and the calibrated EOS parameters, which are important for many research topics in physical sciences.

Monday, November 21, 2016 10:40AM - 12:37PM – Session H6 Aerodynamics: Flexible Structures B114 - Thomas Ward, Iowa State University

10:40AM H6.00001 Vibrating cantilever beam in a flowing soap film, VEERA SAJJANAPU, THOMAS WARD, Iowa State University — We present an experimental study of the interaction between a flexible cantilever beam and a flowing fluid medium using a soap film. The vertically falling soap film is capable of attaining speeds ranging from 1.5 - 3 m/s with an operating test section width of 7.5 cm. Experiments were conducted for flexible cantilever beams of length $L \leq 10$ mm yielding Reynolds number 5000 $< Re < 10000$ and of cantilever beam thickness ranging from 0.03 - 0.08 mm were placed at angles of attack ranging from $10^\circ - 50^\circ$. We visualize the beam displacements and wake with a high-speed camera. Assuming small vibrational amplitudes, we consider the Euler-Bernoulli beam theory to understand the dynamics. From the analysis we find that the normalized average displacement is linear with respect to the square of the free-stream velocity. The vibrational amplitude is also discussed using a similar scaling. Finally, visualization of the downstream vortex structure is related to a beams displacement and vibrational frequency using dimensional analysis.

10:53AM H6.00002 Interaction of a highly flexible cantilever beam with grid-generated turbulent flow, OLEG GOUSHCHA, Manhattan College, YIANNIS ANDREOPoulos, The City College of New York — Experiments have been performed to study the fluid-structure interaction of a flexible cantilever beam with the free end facing upstream in anisotropic turbulent flow. Velocity fluctuations in the wind tunnel flow were generated by a turbulence grid. Time-Resolved Particle Image Velocimetry (TR-PIV) techniques were used to acquire velocity data on the plane of a CW laser illumination. Forces exerted on the beam were estimated based on the PIV data by analytically solving the Pressure Poisson Equation (PPE). Two types of interaction were observed. At a lower Reynolds number, fluid forces excite the beam into oscillations of small magnitude. At higher Reynolds number, the excitation is stronger, deflecting the beam sufficiently to cause flow separation and vortex shedding on one side of the beam. The resultant vortices exert additional forces on the beam producing large magnitude oscillations of the beam.

11:06AM H6.00003 Aeroelastic Flutter Behavior of a Cantilever and Elastically Mounted Plate within a Nozzle-Diffuser Geometry, LUIS PHILLIPE TOSI, TIM COLONIUS, California Institute of Technology, HYEONG JAE LEE, STEWART SHERRIT, Jet Propulsion Laboratory, California Institute of Technology, JET PROPULSION LABORATORY COLLABORATION, CALIFORNIA INSTITUTE OF TECHNOLOGY COLLABORATION — 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 interesting 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. Similar behavior has been also observed in elastically mounted rigid plates, enabling new designs for such devices. This work explores the relationship between the aeroelastic flutter instability boundaries and relevant non-dimensional parameters via experiments, numerical, and stability analyses. Parameters explored consist of a non-dimensional stiffness, a non-dimensional mass, non-dimensional throat size, and Reynolds number. A map of the system response in this parameter space may serve as a guide to future work concerning possible electrical output and failure prediction in harvesting devices.

11:19AM H6.00004 Wake reconfiguration downstream of an inclined flexible cylinder at the onset of vortex-induced vibrations, REMI BOURGUET, IMFT / CNRS, MICHAEL TRIANTAFYLLOU, MIT — Slender flexible cylinders immersed in flow are common in nature (e.g. plants and trees in wind) and in engineering applications, for example in the domain of offshore engineering, where risers and mooring lines are exposed to ocean currents. Vortex-induced vibrations (VIV) naturally develop when the cylinder is placed at normal incidence but they also appear when the body is inclined in the current, including at large angles. In a previous work concerning a flexible cylinder inclined at 80 degrees, we found that the occurrence of VIV is associated with a profound alteration of the flow dynamics: the wake exhibits a slanted vortex shedding pattern in the absence of vibration, while the vortices are shed parallel to the body once the large-amplitude VIV regime is reached. The present study aims at bridging the gap between these two extreme configurations. On the basis of direct numerical simulations, we explore the intermediate states of the flow-structure system. We identify two dominant components of the flow: a high-frequency component that relates to the stationary body wake and a low-frequency component synchronized with body motion. We show that the scenario of flow reconfiguration is driven by the opposite trends of these two component contributions.

11:32AM H6.00005 Influence of flexible fins on vortex-induced load over a circular cylinder at low Reynolds number, RAJEEV K JAUMAN, PARDHA S GURUGUBELLI, Natl Univ of Singapore — Rigid fins/fairings are known to reduce the vortex induced periodic forces exerting on a cylinder by extending the shear layers interaction further downstream to avoid alternate oppositely-signed shed vortices in the afterbody region. In this work, we present a numerical analysis on the effect of flexible fins with their leading edges fixed tangentially to the cylinder and the trailing edges are free to flap in the wake of two-dimensional (2D) cylinder. Two-dimensional simulations are carried out with varying non-dimensional flexural rigidity, $K_B \in [0.01, 1]$ at a fixed a non-dimensional mass ratio, $m^* = 0.1$ and Reynolds number, $Re = 100$, defined based on the cylinder diameter. We investigate the role of flexibility in altering the wake flow and load generation over the cylinder body. As the K_B is reduced, there exists a critical K_B below which the flexible fins lose their stability to perform flapping and the drag acting on combined cylinder flexible fins begins to increase. However surprisingly, we observe that due to the flexible fin flapping the periodic lift forces acting on the cylinder drops significantly. We show that we can achieve an approx. 62.5% decrease in the nett periodic lift forces when compared to the bare cylinder.

11:45AM H6.00006 Why do inverted-flags flap in a uniform steady flow?¹, JOHN SADER, The University of Melbourne, JULIA COSSE, California Institute of Technology, DAEGYOUNG KIM, KAIST, BOYU FAN, MORTEZA GHARIB, California Institute of Technology — The dynamics of a cantilevered elastic sheet, with a uniform steady flow impinging on its clamped-end, have been studied widely and provide insight into the stability of flags and biological phenomena. Recent measurements by Kim *et al.* J. Fluid Mech. **736**, R1 (2013) show that reversing the sheet's orientation, with the flow impinging on its free-edge, dramatically alters its dynamics. In this talk, we use a combination of mathematical theory, scaling analysis and measurement to explore the physical mechanisms driving the observed large-amplitude flapping motion of an inverted-flag. Flapping is found to be periodic predominantly, with a transition to chaos as flow speed increases. These findings have implications to leaf motion and other biological processes, such as the dynamics of individual hairs, because they also can present an inverted-flag configuration.

¹Gordon and Betty Moore Foundation; Ministry of Trade, Industry and Energy, Republic of Korea; Australian Research Council grants scheme.

11:58AM H6.00007 Stability of low aspect ratio inverted flags and rods in a uniform flow¹, CECILIA HUERTAS-CERDEIRA, California Institute of Technology, JOHN E. SADER, The University of Melbourne, MORTEZA GHARIB, California Institute of Technology — Cantilevered elastic plates and rods in an inverted configuration, where the leading edge is free to move and the trailing edge is clamped, undergo complex dynamics when subjected to a uniform flow. The stability of low aspect ratio inverted plates and rods is theoretically examined, showing that it is markedly different from that of their large aspect ratio counterpart. In the limit of zero aspect ratio, the undeflected equilibrium position is found to be stable for all wind speeds. A saddle-node bifurcation emerges at finite wind speed, giving rise to a strongly deflected stable and a weakly deflected unstable equilibria. This theory is compared to experimental measurements, where good agreement is found.

¹This research was supported by a grant of the Gordon and Betty Moore Foundation, the Australian Research Council grants scheme and a “la Caixa” Fellowship Grant for Post-Graduate Studies of “la Caixa” Banking Foundation.

12:11PM H6.00008 Flow-Induced Flutter of Multi-Inverted Flag Configurations: Vortex Dynamics and Flutter Behaviors¹, AARON RIPS, KOUROSH SHOOLE, RAJAT MITTAL, Johns Hopkins University — Flow-induced flutter of “inverted” flags has potential application in energy-harvesting, enhanced mixing and heat transfer enhancement. While a number of previous studies have explored the dynamics of single inverted flags, the current study examines the issue of the coupled dynamics of multi-inverted flag configurations. The primary configurations investigated here have two inverted filaments in either side-by-side or tandem formations. The flapping behavior, dynamics, and flow-structure interaction of each filament as well as the synchronization between the dynamics of the filaments was studied. The investigation of the tandem configuration shows coupling of the flapping dynamics for a variety of separations as well as the ability of the trailing filament to drive the behavior of the leading filament. The behavior of the side-by-side configuration suggests the flapping dynamics in this configuration is dominated by the vortex shedding at the fixed trailing edges. The side-by-side configuration also exhibits multiple distinctive flutter behavior regimes depending on separation distance. Finally, simulations of flag formations with >2 flags are also explored. Implication of the findings on energy-harvesting applications is discussed.

¹The authors would like to acknowledge financial support from NSF Grant CBET 1357819

12:24PM H6.00009 The Stability of Tip Vortices Generated by a Flexible Wind Turbine, STEVEN RODRIGUEZ¹, JUSTIN JAWORSKI, Lehigh University — The influence of root-vortices and a trailing vortex sheet on tip-vortex dynamics of a flexible onshore and floating-offshore wind turbine configurations are investigated numerically. The rotor near-wake is generated using a lifting-line free vortex wake method, which is coupled to a finite element solver for linear flapwise bending deformations. A synthetic time series of rigid-body rotor motions emulates the offshore environment for the NREL 5MW reference wind turbine. To evaluate the influence of root vortices and the trailing vortex sheet, a linear stability analysis is first performed for a rotor wake consisting only of the tip vortices. The stability analysis is then modified to account for the presence of the root vortices and trailing vortex sheet. Stability trends of the two analyses are compared to identify any influence that the root vortices and the trailing vortex sheet have on the tip-vortex dynamics. Lastly, the aforementioned stability analyses are conducted for varying tip speed ratios to identify intrinsically stable helical structures.

Monday, November 21, 2016 10:40AM - 12:50PM –
Session H7 Flow Control: Vortices, Coherent Structures and Surfaces B115 - Robert Breidenthal, University of Washington

10:40AM H7.00001 Active Control of Stationary Vortices.¹, GIOVANNI NINO, Quest Integrated LLC, ROBERT BREIDENTHAL, ADITI BHIDE, ADITYA SRIDHAR, University of Washington — A system for active stationary vortex control is presented. The system uses a combination of plasma actuators, pressure sensors and electrical circuits deposited on aerodynamic surfaces using printing electronics methods. Once the pressure sensors sense a change on the intensity or on the position of the stationary vortices, its associated controller activates a set of plasma actuator to return the vortices to their original or intended positions. The forces produced by the actuators act on the secondary flow in the transverse plane, where velocities are much less than in the streamwise direction. As a demonstration case, the active vortex control system is mounted on a flat plate under low speed wind tunnel testing. Here, a set of vortex generators are used to generate the stationary vortices and the plasma actuators are used to move them. Preliminary results from the experiments are presented and compared with theoretical values.

¹Thanks to the USAF AFOSR STTR support under contract FA9550-15-C-0007

10:53AM H7.00002 Modifying formation and merging of shear-layer vortices using local periodic heating¹, CHI-AN YEH, PHILLIP MUNDAY, KUNIHIKO TAIRA, Florida State University — The flow physics of a thermally forced shear layer downstream of a finite-thickness splitter plate is examined with 2D compressible DNS. Unsteady forcing is introduced at the tip of the plate with an oscillatory heat flux boundary condition. We observe that the forcing can introduce small-level oscillatory surface vorticity flux and generates volumetric baroclinic vorticity at the actuation frequency in the vicinity of the tip, which in turn is able to modify the vortex dynamics of the shear layer downstream. When using forcing frequency near the first subharmonic of the baseline flow, the strength of each roll-up vortex appears to have greater fluctuation, with its mean remaining unchanged from that of the baseline. The fluctuation added to each vortex leads to a wider shear layer by either encouraging the vortex deviating from the centerline while convecting downstream, or encouraging the merging process to take place earlier upstream. When forcing excites the roll-up and lock the roll-up frequency onto that of actuation, the mean strength of the vortices can be accordingly modified by controlling the amount of vorticity fed into each formed vortex. Consequently, the modified strength alters the shear layer thickness while vortices convecting along the centerline. Steady forcing

¹This work was supported by the US Army Research Office (Grant W911NF-14-1-0224).

11:06AM H7.00003 Controlled Flow Distortion in an Offset Diffuser using Hybrid Trapped Vorticity¹, T.J. BURROWS, B. VUKASINOVIC, A. GLEZER, Georgia Institute of Technology — Trapped vorticity concentration engendered by deliberate modification of the internal surface of an offset diffuser is coupled with a spanwise array of surface-integrated fluidic-oscillating jets for hybrid flow control of streamwise vorticity concentrations that dominate the base flow and give rise to flow distortions at the engine inlet. The local and global characteristics of the diffuser flow in the absence and presence of the actuation are investigated at Mach numbers up to $M = 0.7$, using surface oil-flow visualization and pressure distributions, and particle image velocimetry. It is shown that two sources of streamwise vorticity dominate the base flow distortion, namely, corner and a central pair of counter-rotating vortices. The present investigations demonstrate that the actuation affects the topology, strength and scale of the trapped vorticity and thereby its coupling to and interaction with the counter rotating streamwise vortices, where the central vortex pair becomes fully suppressed. As a result, the actuation significantly alters the evolution of the flow within the diffuser, and leads to significant suppression of pressure distortion at the engine inlet (by about 80%) at actuation level that is less than 0.7% of the diffuser's mass flow rate. These findings indicate the utility of hybrid trapped vorticity actuation for mitigating adverse effects of secondary vorticity concentrations formed by local separation and corner flows.

¹Supported by ONR

11:19AM H7.00004 Analysis of vortical structure over sinusoidal riblet surface in turbulent channel flow by means of Dual-plane stereoscopic PIV measurement, HIROYA MAMORI, Tokyo University of Science, KYOTARO YAMAGUCHI, Tokyo University of Agriculture and Technology, MONAMI SASAMORI, Japan Aerospace Exploration Agency, KAORU IWAMOTO, AKIRA MURATA, Tokyo University of Agriculture and Technology — We perform a dual-plane stereoscopic particle image velocimetry (DPS-PIV) measurement to investigate vortical structure over a sinusoidal riblet surface in the turbulent channel flow. In the sinusoidal riblet surface, its lateral spacing of the adjacent walls varies in the streamwise direction and 12% of the drag reduction rate has been confirmed in the turbulent channel flow. The DPS-PIV measurement system consists of four high-speed CCD cameras and the two laser sheets. In the flat case, the profile of the velocity statistics shows a good agreement with previous data. In the riblet case, the velocity statistics decrease in the region close to the wall as compared with that of the flat case. Since all velocity components are measured on adjacent laser sheets simultaneously, vortical structures can be obtained by a second invariant of the tensor i.e. the Q value. According to an analysis for the Q value, we found that the vortical structure is shifted up and attenuated owing to the riblet. Moreover, the riblet prevents the approaching of the vortical structure: the upward and downward flows in the region near the wall are generated by the riblet; if the vortical structure approaches the wall, it is shifted away from the wall due to the upward flow.

11:32AM H7.00005 Vortex-Induced Vibration of a Circular Cylinder Fitted with a Single Spanwise Tripwire, EHSAN VAZIRI, ALIS EKMEKCI, University of Toronto — A spanwise tripwire can be used to alter the coherence and strength of the vortex shedding from cylindrical structures. While this has been well-documented for cylinders in stationary state, there exists a lack of understanding regarding the control induced by spanwise tripwires for cylinders undergoing vortex-induced vibration (VIV). The current experimental research investigates the consequences of spanwise tripping on VIV of a cylinder. Experiments are conducted in a recirculating water tunnel at a Reynolds number of 10,000. The test setup allows the rigid test cylinder to have one-degree-of-freedom vibration in the cross-flow direction as a result of fluid forcing. To measure the cylinder motion, a high-resolution laser displacement sensor is used. The tripwire diameter to cylinder diameter ratio is fixed at 6.1%. Various angular positions of tripwire are studied ranging from 40 to 90 degrees. It is shown that the tripwire location controls the pattern, amplitude, frequency, and mid-position of oscillations significantly. Different oscillation modes are classified based on the observed oscillation pattern, amplitude and frequency. Oscillation amplitude can be reduced by 61% with respect to the amplitude of a clean cylinder undergoing VIV under the same flow condition.

11:45AM H7.00006 A single bubble in a turbulent channel flow: Towards understanding drag reduction, RAGHURAMAN N. GOVARDHAN, NARSING KUMAR JHA, Indian Institute of Science — Two phase turbulent flows are ubiquitous in natural systems such as in clouds and oceans, besides in the chemical process industry. There is also interest in such flows from the drag reduction perspective of marine vehicles through the injection of bubbles into the boundary layer. In these flows, the bubbles interact with the turbulent structures present in the flow resulting in complex bubble paths and modification of the structures, the physical mechanisms of which are not well understood. In the present work, we experimentally study the interaction of a single air bubble with turbulent structures in a fully developed horizontal turbulent channel flow. The bubble path is tracked using high speed imaging in two perpendicular planes, while the vortical structures are tracked using time resolved particle image velocimetry (PIV). We observe different bubble paths even at the same incoming flow Reynolds number and bubble size, which is likely related to the interaction of the bubble with the incoming turbulent structures. The effect of bubble size on the bubble motion trajectories and its relation to structures will be presented in the meeting. This simplified study of interaction of a single bubble can help in understanding more complex interaction of multiple bubbles with multi-scale turbulent flows.

11:58AM H7.00007 Solution to Shape Identification of Unsteady Natural Convection Fields to Control Temperature Distribution, EIJI KATAMINE, Department of Mechanical Engineering, National Institute of Technology, Gifu College, SHINYA IMAI, Department of Mechanical Engineering, Nagaoka University of Technology, MATHEMATICAL DESIGN TEAM, COMPUTATIONAL MECHANICS TEAM — This paper presents a numerical solution to shape identification of unsteady natural convection fields to control temperature to a prescribed distribution. The square error integral between the actual temperature distributions and the prescribed temperature distributions on the prescribed sub-boundaries during the specified period of time 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. Numerical analyses program for the shape identification is developed based on FreeFem++, and the validity of proposed method is confirmed by results of 2D numerical analyses.

12:11PM H7.00008 Experimental study of turbulent structures over hairy poro-elastic surfaces, MARIE COULIOU, Department of Mechanics, KTH, Stockholm, Sweden, JONAS HANSSON, WOUTER VAN DER WIJNGAART, Micro and Nanosystems, KTH, Stockholm, Sweden, FREDRIK LUNDELL, SHERVIN BAGHERI, Department of Mechanics, KTH, Stockholm, Sweden — Flows over slender, deformable and dense structures are ubiquitous in both nature and technological applications, ranging from the atmospheric flow over trees to the flow over the skin of organisms. In order to create a fundamental understanding of how poro-elastic surface can be used for flow control purposes, our work focuses on the behaviour of wall-bounded turbulent flows over fibrous poro-elastic surfaces. We fabricate the coatings using Off-Stoichiometry-Thiolene-Epoxy (OSTE+) polymers and multidirectional UV-lithography which enables us to design arrays of flexible pillars with various geometrical parameters (aspect ratio, pitch, inclinaison, etc.). We assess the effects of these coatings on an overlying low-Reynolds number turbulent flow using a water-table facility and PIV measurements. In particular, we focus on the modification of near wall turbulent structures in both space and time due to the presence of the poro-elastic coatings.

12:24PM H7.00009 Manipulation of flow around bluff bodies by flexible slender filaments, MOHAMMAD OMIDYEGANEH, ALFREDO PINELLI, City University London — Manipulation of bluff bodies wakes to control the intensity of fluid forces and the induced solid vibrations is of paramount importance. A biomimetic passive control based on the use of flexible slender appendages protruding from the body into the separated region has shown promising achievements in drag reduction and moderating force fluctuations. The present research aimed at understating and optimizing the physical properties and the arrangement of elongated flexible filaments to delay the 3D transition of the wake in terms of Reynolds number, mean drag reduction, and mitigation of the force fluctuations. The numerical campaign unveiled the role of flexural stiffness of the filaments: matching the natural frequency with the vortex shedding frequency enhances the mixing at the lee side. However, softer filaments (i.e. larger time scales) lock-in on either side of mid plane breaking the symmetry of the flow field (inducing a net lift force). In addition to 2D effects, the presence of filaments can interfere with the 3D bifurcation process resulting in a delay of the spanwise destabilization of the wake. The most effective parameter for this transitional interference is the spacing between filaments that should be smaller than the wavelength of the dominant 3D unstable mode.

12:37PM H7.00010 Control of Flow Structure on Non-Slender Delta Wing: Bio-inspired Edge Modifications, Passive Bleeding, and Pulsed Blowing, MEHMET METIN YAVUZ, ALPER CELIK, CENK CETIN, Middle East Technical University — In the present study, different flow control approaches including bio-inspired edge modifications, passive bleeding, and pulsed blowing are introduced and applied for the flow over non-slender delta wing. Experiments are conducted in a low speed wind tunnel for a 45 degree swept delta wing using qualitative and quantitative measurement techniques including laser illuminated smoke visualization, particle image velocimetry (PIV), and surface pressure measurements. For the bio-inspired edge modifications, the edges of the wing are modified to dolphin fluke geometry. In addition, the concept of flexion ratio, a ratio depending on the flexible length of animal propulsors such as wings, is introduced. For passive bleeding, directing the free stream air from the pressure side of the planform to the suction side of the wing is applied. For pulsed blowing, periodic air injection through the leading edge of the wing is performed in a square waveform with 25% duty cycle at different excitation frequencies and compared with the steady and no blowing cases. The results indicate that each control approach is quite effective in terms of altering the overall flow structure on the planform. However, the success level, considering the elimination of stall or delaying the vortex breakdown, depends on the parameters in each method.

Monday, November 21, 2016 10:40AM - 12:50PM –
Session H8 Nonlinear Dynamics: Chaos and Bifurcations B116 - Marc Pradas, Open University

10:40AM H8.00001 Stochastic versus chaotic behaviour in the noisy generalized Kuramoto-Sivashinsky equation, HIROSHI GOTODA, Department of Mechanical Engineering, Tokyo University of Science, MARC PRADAS, Department of Mathematics and Statistics, The Open University, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London — Random fluctuations are well-known to have significant impact on the formation of complex spatiotemporal patterns in a wide spectrum of biological, engineering and physical environments, including fluid systems such Rayleigh-Bénard convection, contact line dynamics, or waves in free-surface thin film flows. Many of these systems can be modeled by stochastic partial differential equations in large or unbounded domains, a simple prototype of which is the generalised Kuramoto-Sivashinsky (gKS) equation. Its deterministic version has been used in a wide variety of fluid flow contexts, such as two-phase flows with surfactants, free falling films and films in the presence of chemical reactions, heating effects and curved substrates, amongst others. Here we study the dynamical states of the noisy gKS equation by making use of time series techniques based on chaos theory, in particular permutation entropy and nonlinear forecasting. We focus on analyzing temporal signals of global measure in the spatiotemporal pattern as the dispersion parameter of the gKS equation and the strength of the noise are varied, observing a rich variety of different emerging regimes, from high-dimensional chaos to purely stochastic behaviour.

10:53AM H8.00002 Lagrangian chaos in three- dimensional steady buoyancy-driven flows¹, SEBASTIAN CONTRERAS, MICHEL SPEETJENS, HERMAN CLERCX, Eindhoven University of Technology — Natural convection plays a key role in fluid dynamics owing to its ubiquitous presence in nature and industry. Buoyancy-driven flows are prototypical systems in the study of thermal instabilities and pattern formation. The differentially heated cavity problem has been widely studied for the investigation of buoyancy-induced oscillatory flow. However, far less attention has been devoted to the three-dimensional Lagrangian transport properties in such flows. This study seeks to address this by investigating Lagrangian transport in the steady flow inside a cubic cavity differentially-heated from the side. The theoretical and numerical analysis expands on previously reported similarities between the current flow and lid-driven flows. The Lagrangian dynamics are controlled by the Péclet number (Pe) and the Prandtl number (Pr). Pe controls the behaviour qualitatively in that growing Pe progressively perturbs the integrable state (Pe=0), thus paving the way to chaotic dynamics. Pr plays an entirely quantitative role in that Pr<1 and Pr>1 amplifies and diminishes, respectively, the perturbative effect of non-zero Pe.

¹S.C. acknowledges financial support from Consejo Nacional de Ciencia y Tecnología (CONACYT).

11:06AM H8.00003 Slip induced mixing in a model slug flow, S GOWTHAM SANKARANANTH, S PUSHPAVANAM, Indian Inst of Tech-Madras — Mixing of reactants in microfluidic slugs has a significant influence on the performance of processes. We discuss how mixing can be enhanced in slug flows by introducing periodic hydrophobicity on the confining walls. We consider a rectangular slug moving in a straight microchannel constructed by a shift-reflect transform of a unit cell with finite slip on one wall. This leads to alternating regions of slip and no-slip on each wall. The velocity field within the 2D slug is approximated as that in a driven cavity and computed by a Chebyshev spectral collocation. We go beyond a blinking flow model by capturing the velocity field under the discontinuous boundary conditions of inter-cell transit using domain decomposition. Thus, advection is described by a sequence of maps. It is seen that the hydrophobic sections reduce the size of the closer vortex and locally attract the separatrix. This permits "crossing" of streamlines in adjacent unit cells, opening up the possibility of chaotic mixing. "Good crossing", as quantified by an Eulerian indicator called "transversality", seems to occur in a larger area when slug length is comparable to unit cell length. We quantify mixing and the internal structures that result using different Lagrangian techniques to reach a holistic consensus.

11:19AM H8.00004 Nonlinear Dynamic Stability of the Viscoelastic Plate Considering Higher Order Modes, YUANXIANG SUN, CHENG WANG, State Key Laboratory of Explosion Science and Technology, Beijing Institute of Technology — The dynamic stability of viscoelastic plates is investigated in this paper by using chaotic and fractal theory. The nonlinear integro-differential dynamic equation is changed into an autonomic 4-dimensional dynamical system. The numerical time integrations of equations are obtained by using the fourth order Runge-Kutta method. And the Lyapunov exponent spectrum, the fractal dimension of strange attractors and the time evolution of deflection are obtained. The influence of viscoelastic parameter on dynamic buckling of viscoelastic plates is discussed. The effect of higher order modes on dynamic stability of viscoelastic plate is obtained, the necessity of considering higher order modes is discussed.

11:32AM H8.00005 Topology of azimuthally travelling waves in thermocapillary liquid bridges, FRANCESCO ROMANO, HENDRIK C. KUHLMANN, TU Wien — The topology of the laminar three-dimensional flow in a cylindrical liquid bridge driven by thermocapillary forces is investigated. Attention is focussed on travelling hydrothermal waves which are analysed in a co-rotating frame of reference in which the flow becomes steady. Chaotic and regular regions in form of KAM tori are found as well as closed streamlines. The flow features are discussed in terms of shape, location and period of closed orbits, KAM structures, their relation to the basic-state toroidal vortex flow and the dependence on the Marangoni number.

11:45AM H8.00006 Transitional behavior of convective patterns in porous media: Insights from basin stability analysis, HAMID KARANI, CHRISTIAN HUBER, Georgia Institute of Technology — The present study investigates the transitional behavior of convective modes in Horton-Rogers-Lapwood convection (HRLC). We first provide new pore-scale numerical and experimental evidences on the variation of the stability level of single-cell and double-cell convection modes in a 2D HRLC problem. In order to interpret this transitional behavior, we employ the concept of basin stability and develop a basin stability diagram of the first four convection modes in HRLC. This is in contrast to the standard bifurcation analysis of HRLC using linear stability analysis and continuation techniques, which only provides local information about the (range of) existence, and any possible co-existence of different convection modes. The present basin stability analysis of HRLC not only provides the local information about the (co-)existence of different patterns, but also, it determines their relative stability as well as how the basin of stability of these modes contract or expands as the Rayleigh number varies. The results of the present study show how establishing the dependence of basin stability on the Rayleigh number is essential to analyze the transition between different convection patterns observed experimentally and numerically.

11:58AM H8.00007 Forced Snaking, BENJAMIN PONEDEL, EDGAR KNOBLOCH, Univ of California - Berkeley — We study spatial localization in the real subcritical Ginzburg-Landau equation $u_t = m_0 u + m_1 \cos\left(\frac{2\pi}{\ell}x\right) u + u_{xx} + d|u|^2 u - |u|^4 u$ with spatially periodic forcing. When $d > 0$ and $m_1 = 0$ this equation exhibits bistability between the trivial state $u = 0$ and a homogeneous nontrivial state $u = u_0$ with stationary localized structures which accumulate at the Maxwell point $m_0 = -3d^2/16$. When spatial forcing is included its wavelength is imprinted on u_0 creating conditions favorable to front pinning and hence spatial localization. We use numerical continuation to show that under appropriate conditions such forcing generates a sequence of localized states organized within a snakes-and-ladders structure centered on the Maxwell point, and refer to this phenomenon as *forced snaking*. We determine the stability properties of these states and show that longer lengthscale forcing leads to stationary trains consisting of a finite number of strongly localized, weakly interacting pulses exhibiting *foliated snaking*.

12:11PM H8.00008 Noise induced transitions in rugged energy landscapes, MARC

PRADAS, Department of Mathematics and Statistics, The Open University, ANDREW DUNCAN, Department of Mathematics, Imperial College London, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, GREG PAVLIOTIS, Department of Mathematics, Imperial College London — External or internal random fluctuations are ubiquitous in many physical and technological systems and can play a key role in their dynamics often inducing a wide variety of complex spatiotemporal phenomena, including noise-induced spatial patterns and noise-induced phase transitions. Many of these phenomena can be modelled by noisy multiscale systems characterized by the presence of a wide range of different time- and lengthscales interacting nontrivially with each other. Here we analyse the effects of additive noise on systems that are described in terms of a rugged energy landscape, modelled as a slowly-varying multiscale potential perturbed by periodic multiscale fluctuations. Some examples of this problem include the dynamics of sessile droplets on heterogeneous substrates, crystallization and the evolution of protein folding. We demonstrate that the interplay between noise and the small scale fluctuations in the potential can give rise to a dramatically different bifurcation structure and dynamical behaviour compared to that of the original, unperturbed model. For instance, we observe several nontrivial and largely unexpected dynamic-state transitions controlled by the noise intensity. We characterize these transitions in terms of critical exponents.

12:24PM H8.00009 Onset of chaos in helical vortex breakdown at low Reynolds number, SIMON PASCHE, EPFL - LMH, FRANÇOIS GALLAIRE, EPFL - LFMI, FRANÇOIS AVELLAN, EPFL - LMH — Swirling jet flows are generally characterized by two non-dimensional parameters: the swirl and the Reynolds number. Bubble, spiral or double spiral vortex breakdown as well as columnar vortex are part of the observed dynamics when these two control parameters are varied. This rich dynamic produces strong mixing that is traditionally investigated in the framework of Lagrangian chaos, with typical applications to combustion chambers.

In contrast to chaotic advection, Eulerian chaos has not been reported for such open flows. Here, Eulerian chaos is studied through direct numerical flow simulations of an unconfined Grabowsky and Berger vortex using the incompressible Navier-Stokes solver NEK5000. At a fixed swirl number, a sequence of periodic, quasiperiodic, chaotic, quasiperiodic and periodic states is observed as the Reynolds number increases from 200 to 300. Therefore, Fourier spectrum, Poincaré section map, sensitivity to initial condition and largest Lyapunov exponent are computed to identify the chaotic window which results from the nonlinear interaction between a self-sustained single helical mode, triggered by an upstream bubble breakdown, and other helical modes. Finally, a route to chaos in the incompressible Navier-Stokes equations is sketched.

12:37PM H8.00010 Large-scale columnar vortices in rotating turbulence¹, NAOTO

YOKOYAMA, Kyoto University, MASANORI TAKAOKA, Doshisha University — In the rotating turbulence, flow structures are affected by the angular velocity of the system's rotation. When the angular velocity is small, three-dimensional statistically-isotropic flow, which has the Kolmogorov spectrum all over the inertial subrange, is formed. When the angular velocity increases, the flow becomes two-dimensional anisotropic, and the energy spectrum has a power law k^{-2} in the small wavenumbers in addition to the Kolmogorov spectrum in the large wavenumbers. When the angular velocity decreases, the flow returns to the isotropic one. It is numerically found that the transition between the isotropic and anisotropic flows is hysteretic; the critical angular velocity at which the flow transitions from the anisotropic one to the isotropic one, and that of the reverse transition are different. It is also observed that the large-scale columnar structures in the anisotropic flow depends on the external force which maintains a statistically-steady state. In some cases, small-scale anticyclonic structures are aligned in a columnar structure apart from the cyclonic Taylor column. The formation mechanism of the large-scale columnar structures will be discussed.

¹This work was partially supported by JSPS KAKENHI.

Monday, November 21, 2016 10:40AM - 12:50PM –
Session H9 General Fluid Dynamics B117 - Thomas Corke, University of Notre Dame**10:40AM H9.00001 Biofuels spills in surface waters a laboratory investigation of mixing and interfacial dynamics¹**, XIAOXIANG WANG, ALINE COTEL, University of Michigan — There are increasing risks of spills of ethanol-based biofuels in aquatic environments, however the environmental impact of such accidents is poorly understood and no adequate mitigation strategies are in place today. The interaction of water and biofuels is a complex dynamical problem and we aim to quantify the physical processes involved in such dynamics. A solution of ethanol and glycol is used to represent a typical ethanol-based fuel. A small-scale Plexiglas tank has been designed to investigate the effect of natural conditions on the mixing of water and biofuels, e.g. slope angle, flow rate, wave amplitude and frequency in wind driven conditions. Our previous work showed that the existence of two distinct mixing regimes; a first turbulence-driven fast mixing regime and a second regime driven by interface instabilities. We investigate these mixing regimes under an extended range of physical parameters representing more natural configurations.

¹Funded by NSF

10:53AM H9.00002 Hydrodynamical force on a solid sphere in an incompressible inviscid fluid¹, RABAB ALARKI, Department of Mathematics & Statistics, Texas AM University, Corpus Christi, D. PALANIAPPAN, Texas AM University, Corpus Christi — Simple analytic results for the hydrodynamical force exerted on a rigid sphere of radius a placed in singularity driven potential flows are determined. The motion induced singularities considered are (i) a source; (ii) a dipole; and (iii) a vortex ring, located at $(0, 0, c)$, where $c > a$. The calculation is based on the exact solutions of the classical Neumann boundary value problem for a spherical boundary in inviscid hydrodynamics. The expressions for the force due to source and dipole are found to be algebraic in a/c , the radius-location ratio, while the result for a vortex ring is expressed in an integral form. Our analysis shows that the force due to a tangentially oriented initial dipole is less than that of a dipole in the radial direction. Graphical illustration are presented demonstrating the variation of the force with respect to a/c . The results may also be of interest in the study of superfluids - treated as incompressible fluids - such as liquid helium or the interior of a neutron star.

¹Supported by King Abdullah Foreign Scholarship Program, Saudi Ministry of Education

11:06AM H9.00003 Numerical investigation on fluid flow past transversely oscillating vertical rectangular cylinder¹ , JEEVANANTHAN KANNAN², ARUL PRAKASH K³, Indian Institute of Technology Madras

— In the present study, the rectangular cylinder was forced to vibrate for various flow configurations such as the AR (Aspect Ratio) ranging from 0.2 to 1 and Reynolds number based on (depth of the cylinder) as 100, 150, 200. The frequency ratio (excitation frequency, f_e / natural shedding frequency, f_{ns}) chosen for the study was 0.5, 0.75, 1.0, 1.5 and 2.0. The vibrating amplitude 0.1, 0.2 and 0.3 of cylinder depth were also considered. For the slender aspect ratios (AR<1), the flow phenomena becomes more complex, due to the short vortex formation length. The separated shear layers were incessantly swiveling behind the cylinder dispense the vortices in the downstream of the wake as inline shedding packets. Three dimensional Studies are also established for the selected cases. The influence of the cylinder vibration on the wake patterns, phase plane, lift, drag force etc. are presented and discussed.

¹This Investigation has been sponsored by ARDB-Aerodynamics Panel, Grant No: (DARO /08/ 1031663/M/I Dated 08/08/2012), India.

²M.S Student

³Associate Professor

11:19AM H9.00004 A Numerical Simulation of the Density Oscilator¹ , SERGIO HERNANDEZ

ZAPATA, ERICK JAVIER LOPEZ SANCHEZ, GERARDO RUIZ CHAVARRIA, Facultad de Ciencias, UNAM — In this work we carry out a numerical simulation for the dynamics that originates when a fluid (salty water) is located on top of another less dense fluid (pure water) in the presence of gravity. This is an unstable situation that leads to the development of intercalating lines of descending salty water and ascending pure water. Another situation is studied where the fluids are in two containers joined by a small hole. In this case a time pattern of alternating flows develops leading to an oscillator. The study of the velocity field around the hole shows than in a certain interval of time it develops intercalating lines like in the former situation. An interesting result is the fact that when a given fluid is flowing in one direction a vorticity pattern develops in the other fluid. The Navier-Stokes, continuity and salt diffusion equations, are solved numerically in cylindrical coordinates, using a finite difference scheme in the axial and radial directions and a Fourier spectral method for the angular coordinate. On the other hand, the second order Adams-Bashforth method is used for the time evolution. The results are compared to a numerical simulation of a pedestrian oscillator we developed based on the Hebling and Molnar social force model.

¹The authors want to acknowledge support by DGAPA-UNAM (Project PAPIIT IN-115315 "Ondas y estructuras coherentes en dinmica de fluidos"

11:32AM H9.00005 Decalcomania , BERNARDO PALACIOS, SANDRA ZETINA, ROBERTO ZENIT, Universidad

Nacional Autonoma de Mexico — Decalcomania is a painting technique, used by many abstract painters such as Max Ernst and Remedios Varo, which is used to create striated textures of unique aesthetic appeal. A decalcomania consists of separating two flexible sheets, between which fluid paint had been placed. As the flexible sheets are detached and the paint retracts from its original edge, finger-like striae are formed. The technique holds a certain similarity to that used to test adhesive materials and to the Saffman-Taylor instability. To fully understand the process, we recreate it in a controlled manner. We used an experimental setup consisting of two transparency sheets, which separate angularly from each at a controlled rate. The process is filmed with a high-speed camera. The separation speed and fluid properties are varied, to determine their effect on the pattern formation and the produced texture. We present and discuss preliminary results of this study.

11:45AM H9.00006 The Effects of Outer Flow Conditions on the Emergence and Evolution of Geometrical Self Similarity of a Bluff Body during Ablation¹ , MICHAEL ALLARD,

CHRISTOPHER M WHITE, University of New Hampshire — The ablation process (i.e., erosion) of a bluff body, low-temperature ablator is investigated. Two experimental configurations in a heated open-circuit thermal boundary layer wind tunnel are considered: (a) the bluff body is supported in the free stream or (b) placed within the boundary layer growing on the bottom wall of the tunnel. These two configurations were chosen to investigate the effects of outer flow conditions (i.e. uniform in the free stream and varying with the boundary layer) on the emergence and evolution of geometrical self similarity during ablation. A time sequence of streamwise-transverse and streamwise-wall normal images were recorded. The images were analyzed to investigate the temporal evolution of the bluff body's projected area, perimeter, and curvature. The results were compared to similar studies where the erosion was caused from fluid shear force and chemical dissolution both of which scaled-similarly. The insights gained from this study can be used to progress towards physics-based models of bluff body ablation.

¹This work is supported by the NSF (CBET-0967224)

11:58AM H9.00007 Latent heat of vehicular motion , FARZAD AHMADI, AUSTIN BERRIER, MOHAMMAD HABIBI, JONATHAN BOREYKO, Virginia Tech —

We have used the thermodynamic concept of latent heat, where a system loses energy due to a solid-to-liquid phase transition, to study the flow of a group of vehicles moving from rest. During traffic flow, drivers keep a large distance from the car in front of them to ensure safe driving. When a group of cars comes to a stop, for example at a red light, drivers voluntarily induce a "phase transition" from this "liquid phase" to a close-packed "solid phase." This phase transition is motivated by the intuition that maximizing displacement before stopping will minimize the overall travel time. To test the effects of latent heat on flow efficiency, a drone captured the dynamics of cars flowing through an intersection on a Smart Road where the initial spacing between cars at the red light was systematically varied. By correlating the experimental results with the Optimal Velocity Model (OVM), we find that the convention of inducing phase transitions at intersections offers no benefit, as the lag time (latent heat) of resumed flow offsets the initial increase in displacement. These findings suggest that in situations where gridlock is not an issue, drivers should not decrease their spacing during stoppages in order to maximize safety with no loss in flow efficiency.

12:11PM H9.00008 Large-aperture Tunable Plasma Meta-material to Interact with Electromagnetic Waves¹, THOMAS CORKE², ERIC MATLIS³, University of Notre Dame — The formation of spatially periodic arrangements of glow discharge plasma resulting from charge instabilities were investigated as a tuneable plasma meta-material. The plasma was formed between two 2-D parallel dielectric covered electrodes: one consisting of an Indium-Tin-Oxide coated glass sheet, and the other consisting of a glass-covered circular electrode. The dielectric covered electrodes were separated by a gap that formed a 2-D channel. The gap spacing was adjustable. The electrodes were powered by a variable amplitude AC generator. The parallel electrode arrangement was placed in a variable pressure vacuum chamber. Various combinations of gap spacing, pressure and voltage resulted in the formation of spatially periodic arrangements (lattice) of glow discharge plasma. The lattice spacing perfectly followed 2-D packing theory, and was fully adjustable through the three governing parameters. Lattice arrangements were designed to interact with electromagnetic (EM) waves in the frequency range between 10GHz-80GHz. Its feasibility was investigate through an EM wave simulation that we adapted to allow for plasma permittivity. The results showed a clear suppression of the EM wave amplitude through the plasma gratings.

¹Supported by AFOSR

²Fellow

³Member

12:24PM H9.00009 Fluid Shearing for Accelerated Chemical Reactions - Fluid Mechanics in the VFD, EVGENIA LEIVADAROU, STUART DALZIEL, Univ of Cambridge, G. K. BATCHELOR LABORATORY, DEPARTMENT OF APPLIED MATHEMATICS AND THEORETICAL PHYSICS TEAM — The Vortex Fluidic Device (VFD) is a rapidly rotating tube that can operate under continuous flow with a jet feeding liquid reactants to the tube's hemispherical base. It is a new 'green' approach to the organic synthesis with many industrial applications in cosmetics, protein folding and pharmaceutical production. The rate of reaction in the VFD is enhanced when the collision rate is increased. The aim of the project is to explain the fluid mechanics and optimize the performance of the device. One contribution to the increased yield is believed to be the high levels of shear stress. We attempt to enhance the shear stress by achieving high velocity gradients in the boundary layers. Another factor is the uncontrolled vibrations due to imperfections in the bearings and therefore it is important to assess their influence in the initial spreading. The surface area of the film should be maximized with respect to the rotation rate, geometry and orientation of the tube, flow rate, wettability and contact line dynamics. Experiments are presented for a flat disk and a curved bowl, establishing the optimum height of release, rotation rate and tube orientation. Vibrations were imposed to investigate the changes in the film formation. We discuss the implications of our results in the VFD.

12:37PM H9.00010 A Theoretical and Experimental Study for a Developing Flow in a Thin Fluid Gap¹, QIANHONG WU, JI LANG, KEI-PENG JEN, Villanova University, RUNGUN NATHAN, Penn State Berks, VUCBMSS TEAM — In this paper, we report a novel theoretical and experimental approach to examine a fast developing flow in a thin fluid gap. Although the phenomena are widely observed in industrial applications and biological systems, there is a lack of analytical approach that captures the instantaneous fluid response to a sudden impact. An experimental setup was developed that contains a piston instrumented with a laser displacement sensor and a pressure transducer. A sudden impact was imposed on the piston, creating a fast compaction on the thin fluid gap underneath. The motion of the piston was captured by the laser displacement sensor, and the fluid pressure build-up and relaxation was recorded by the pressure transducer. For this dynamic process, a novel analytical approach was developed. It starts with the inviscid limit when the viscous fluid effect has no time to appear. This short process is followed by a developing flow, in which the inviscid core flow region decreases and the viscous wall region increases until the entire fluid gap is filled with viscous fluid flow. A boundary layer integral method is used during the process. Lastly, the flow is completely viscous dominant featured by a typical squeeze flow in a thin gap. Excellent agreement between the theory and the experiment was achieved. The study presented herein, filling the gap in the literature, will have broad impact in industrial and biomedical applications.

¹This research was supported by the National Science Foundation under Award 1511096

Monday, November 21, 2016 10:40AM - 12:50PM –
Session H10 DFD GPC: Convection and Buoyancy Driven Flows: Theory I B118-119 -
 Olga Shishkina, Max Planck Institute for Dynamics and Self-Organization

10:40AM H10.00001 Hydrodynamic stability in the presence of a stochastic source: convection as a case study, JARED WHITEHEAD, Brigham Young University, JURAJ FOLDES, University of Virginia, NATHAN GLATT-HOLTZ, Tulane University, GEORDIE RICHARDS, Utah State University — We quantify the stability of a conductive state in Rayleigh-Bénard convection when the fluid is driven not only by an enforced temperature gradient, but also by a mean zero stochastic (in time) internal heat source, a modeled system applicable to situations such as convection in stars, nuclear reactors, and the earth's mantle. We explore the effects of such a mean zero forcing on the onset of convection. The methods applied to the convection problem here, are applicable to any other question of hydrodynamic stability where a stochastic forcing is present.

10:53AM H10.00002 Temperature variance profiles of turbulent thermal convection at high Rayleigh numbers¹, XIAOZHOU HE, Harbin Institute of Technology Shenzhen Graduate School, EBERHARD BODENSCHATZ, Max Planck Institute for Dynamics and Self-Organization, GUENTER AHLERS, University of California Santa Barbara — We present measurements of the Nusselt number Nu , and of the temperature variance σ^2 as a function of vertical position z , in turbulent Rayleigh-Bénard convection of two cylindrical samples with aspect ratios (diameter D /height L) $\Gamma = 0.50$ and 0.33 . Both samples had $D = 1.12$ m but different L . We used compressed SF₆ gas at pressures up to 19 bars as the fluid. The measurements covered the Rayleigh-number range $10^{13} < Ra < 5 \times 10^{15}$ at a Prandtl number $Pr \simeq 0.80$. Near the side wall we found that σ^2 is independent of Ra when plotted as a function of z/λ where $\lambda \equiv L/(2Nu)$ is a thermal boundary-layer thickness. The profiles $\sigma^2(z/\lambda)$ for the two Γ values overlapped and followed a logarithmic function for $20 \lesssim z/\lambda \lesssim 120$. With the observed "-1"-scaling of the temperature power spectra and on the basis of the Perry-Townsend similarity hypothesis, we derived a fitting function $\sigma^2 = p_1 \ln(z/\lambda) + p_2 + p_3(z/\lambda)^{-0.5}$ which describes the σ^2 data up to $z/\lambda \simeq 1500$.

¹Supported by the Max Planck Society, the Volkswagenstiftung, the DFD Sonderforschungsbereich SFB963, and NSF Grant DMR11-58514.

11:06AM H10.00003 Measured temperature fluctuations and Reynolds number in turbulent Rayleigh-Bénard convection with varying roughness size¹, YICHAO XIE, KEQING XIA,

Department of Physics, The Chinese University of Hong Kong — We present measurements of the temperature fluctuations σ_T and of the Reynolds number Re in turbulent Rayleigh-Bénard convection in cylindrical cell with pyramid-shaped rough top and bottom plates. To study the effects of roughness size, we varied a roughness parameter λ , defined as a single roughness height h (kept at a constant of 8 mm) over its base width d , from 0.5 to 4.0. Fluorinert Liquid FC-770 was used as the working fluid with the Rayleigh number Ra varying from 4.49×10^9 to 9.94×10^{10} and Prandtl number Pr kept at 23.34. It is found that σ_T in both cell center and sidewall increases dramatically with λ . The scaling exponent of the normalized σ_T with respect to Ra increases from -0.16 to -0.09 at cell center and -0.23 to -0.08 near sidewall when λ is increased from 0.5 to 4.0. The Reynolds number Re based on the circulation time of the large-scale circulation (LSC) also increases with λ , suggesting a faster LSC. The scaling exponent of Re with respect to Ra increases from 0.47 to 0.55 with λ increased from 0.5 to 4.0. The study reveals that the flow and temperature fluctuations are very sensitive to the perturbation induced by rough plate with vary λ .

¹This work is supported by the Hong Kong Research Grant Council under grant number N_CUHK437/15.

11:19AM H10.00004 Scaling Analysis of Temperature Variability Between a Rotating Cylinder and a Turbulent Buoyant Jet, CAELAN LAPOINTE, NICHOLAS T. WIMER, TORREY R.S. HAYDEN,

JASON D. CHRISTOPHER, GREGORY B. RIEKER, PETER E. HAMILTON, University of Colorado, Boulder — Vortex shedding from a cylinder is a canonical problem in fluid dynamics and is a phenomenon whose behavior is well documented for a wide range of Reynolds numbers. Industrial processes, by contrast, often have many moving parts that may also be exposed to high temperatures, resulting in highly complex flow fields. This complexity can, in turn, introduce velocity and temperature variations that may be undesirable for a particular industrial process. In this study, we specifically seek to understand and parameterize temperature variability between a rotating cylinder and a high-temperature turbulent buoyant jet. The relevance of this configuration for industrial processing is outlined, and velocity and temperature fields between the jet and cylinder are obtained using large eddy simulations (LES). In the LES, key parameters such as the angular velocity and diameter of the cylinder, the dimensions, velocity, and temperature of the turbulent buoyant jet, and the distance between the cylinder and the jet are varied. The resulting LES results are then used to develop scaling relationships between temperature variance near the cylinder and other problem parameters. Such scaling relations will be highly beneficial for the estimation of temperature variations in industrial applications.

11:32AM H10.00005 Temperature fluctuation in Rayleigh-Bénard convection: Logarithmic vs power-law¹, YU-HAO HE, KE-QING XIA, The Chinese University of Hong Kong — We present an experimental

measurement of the rms temperature (σ_T) profile in two regions inside a large aspect ratio ($\Gamma = 4.2$) rectangular Rayleigh-Bénard convection cell. The Rayleigh number (Ra) is from 3.2×10^7 to 1.9×10^8 at fixed Prandtl number ($Pr = 4.34$). It is found that, in one region, where the boundary layer is sheared by a large-scale wind, σ_T versus the distance (z) above the bottom plate, obeys power law over one decade, whereas in another region, where plumes concentrate and move upward (plume-ejection region), the profile of σ_T has a logarithmic dependence on z . When normalized by a typical temperature scale θ_* , the profiles of σ_T at different Rayleigh numbers collapse onto a single curve, indicating a universality of σ_T profile with respect to Ra . 1. This work is supported by the Hong Kong Research Grant Council under grant number N_CUHK437/15.

11:45AM H10.00006 The scaling transition between Nu number and boundary thickness in RB convection, HONG-YUE ZOU, SKLTCS, COE, Peking Univ., XI CHEN, Department of Mechanical Engineering, Texas Tech Univ., ZHEN-SU SHE, SKLTCS, COE, Peking Univ. — A quantitative theory is developed for the vertical mean temperature profile (MTP) and mean velocity profile (MVP) in turbulent Rayleigh-Bénard convection(RBC), which explains the experimental and numerical observations of logarithmic law in MTP and the coefficient A varying along the Ra. Based on a new mean-field approach via symmetry analysis to wall-bounded turbulent flow yields accurate scaling of the sub-layer buffer layer and log-layer over a wide range of Rayleigh number and gives an explanation of their physical mechanism. In particular, based on the scaling of multi-layer thickness for mean temperature and velocity, we first prove that the coefficient A follows a -0.121 scaling, which agrees well with the experimental data, and the scaling transition of Nu from 1/3 to 0.38 is due to the thickness variation of the multi-layer. The new explanation of mean temperature logarithmic law is that the effect of inverse pressure gradient (LSC) driving the plume to side wall, which yields the similarity between vertical temperature transport and vertical momentum.

11:58AM H10.00007 Adjoint analyses of enhanced solidification for shape optimization in conjugate heat transfer problem, KENICHI MORIMOTO, HIDENORI KINOSHITA, YUJI SUZUKI, The University of Tokyo — In the present study, an adjoint-based shape-optimization method has been developed for designing extended heat transfer surfaces in conjugate heat transfer problems. Here we specifically consider heat conduction-dominated solidification problem under different thermal boundary conditions: (i) the isothermal condition, and (ii) the conjugate condition with thermal coupling between the solidified liquid and the solid wall inside the domain bounded by the extended heat transfer surface. In the present shape-optimization scheme, extended heat transfer surfaces are successively refined in a local way based on the variational information of a cost functional with respect to the shape modification. In the computation of the developed scheme, a meshless method is employed for dealing with the complex boundary shape. For high-resolution analyses with boundary-fitted node arrangement, we have introduced a bubble-mesh method combined with a high-efficiency algorithm for searching neighboring bubbles within a cut-off distance. The present technique can be easily applied to convection problems including high Reynolds number flow. We demonstrate, for the isothermal boundary condition, that the present optimization leads to tree-like fin shapes, which achieve the temperature field with global similarity for different initial fin shapes. We will also show the computational results for the conjugate condition, which would regularize the present optimization due to the fin-efficiency effect.

12:11PM H10.00008 Boundary layer fluctuations and their effects on mean and variance temperature profiles in turbulent Rayleigh-Bénard convection, YIN WANG, Department of Physics, Hong Kong University of Science and Technology, Hong Kong, XIAOZHOU HE, Shenzhen Graduate School, Harbin Institute of Technology, Shenzhen, China, PENG TONG, Department of Physics, Hong Kong University of Science and Technology, Hong Kong — We report simultaneous measurements of the mean temperature profile $\theta(z)$ and temperature variance profile $\eta(z)$ near the lower conducting plate of a specially designed quasi-two-dimensional cell for turbulent Rayleigh-Bénard convection. The measured $\theta(z)$ is found to have a universal scaling form $\theta(z/\delta)$ with varying thermal boundary layer (BL) thickness δ , and its functional form agrees well with the recently derived BL equation by Shishkina et al. The measured $\eta(z)$, on the other hand, is found to have a scaling form $\eta(z/\delta)$ only in the near-wall region with $z/\delta < 2$. Based on the experimental findings, we derive a new BL equation for $\eta(z/\delta)$, which is in good agreement with the experimental results. The new BL equations thus provide a common framework for understanding the effect of BL fluctuations. *This work was supported by the Research Grants Council of Hong Kong SAR and by the China Thousand Young Talents Program.

12:24PM H10.00009 Heat and momentum transport scalings in vertical convection¹

, OLGA SHISHKINA, Max Planck Institute for Dynamics and Self-Organization — For vertical convection, where a fluid is confined between two differently heated isothermal vertical walls, we investigate the heat and momentum transport, which are measured, respectively, by the Nusselt number Nu and the Reynolds number Re . For laminar vertical convection we derive analytically the dependence of Re and Nu on the Rayleigh number Ra and the Prandtl number Pr from our boundary layer equations and find two different scaling regimes: $Nu \sim Pr^{1/4} Ra^{1/4}$, $Re \sim Pr^{-1/2} Ra^{1/2}$ for $Pr \ll 1$ and $Nu \sim Pr^0 Ra^{1/4}$, $Re \sim Pr^{-1} Ra^{1/2}$ for $Pr \gg 1$. Direct numerical simulations for Ra from 10^5 to 10^{10} and Pr from 0.01 to 30 are in excellent agreement with our theoretical findings and show that the transition between the regimes takes place for Pr around 0.1. We summarize the results from Shishkina, Phys. Rev. E 93 (2016) 051102R and present new theoretical and numerical results for transitional and turbulent vertical convection.

¹The work is supported by the Deutsche Forschungsgemeinschaft (DFG) under the grant Sh 405/4 - Heisenberg fellowship.

12:37PM H10.00010 Gravity current jump conditions, revisited , MARIUS UNGARISH, Technion, IIT, Haifa, Israel, ANDREW J HOGG, Un. Bristol UK

— Consider the flow of a high-Reynolds-number gravity current of density ρ_c in an ambient fluid of density ρ_a in a horizontal channel $z \in [0, H]$, with gravity in $-z$ direction. The motion is often modeled by a two-layer formulation which displays jumps (shocks) in the height of the interface, in particular at the leading front of the dense layer. Various theoretical models have been advanced to predict the dimensionless speed of the jump, $Fr = U/\sqrt{g'h}$; g', h are reduced gravity and jump height. We revisit this problem and using the Navier-Stokes equations, integrated over a control volume embedding the jump, derive balances of mass and momentum fluxes. We focus on understanding the closures needed to complete this model and we show the vital need to understand the pressure head losses over the jump, which we show can be related to the vorticity fluxes at the boundaries of the control volume. Our formulation leads to two governing equations for three dimensionless quantities. Closure requires one further assumption, depending on which we demonstrate that previous models for gravity current fronts and internal bores can be recovered. This analysis yield new insights into existing results, and also provides constraints for potential new formulae.

Monday, November 21, 2016 10:40AM - 12:50PM –

Session H11 Jets: Impingement on Surfaces and Crossflows C120-121-122 - Constantine Megaridis, University of Illinois at Chicago

10:40AM H11.00001 Liquid jet impinging orthogonally on a wettability-patterned surface , THEODORE KOUKORAVAS, ARITRA GHOSH, PALLAB SINHA MAHAPATRA, University of Illinois at Chicago, RANJAN GANGULY, Jadavpur University, India, CONSTANTINE MEGARIDIS, University of Illinois at Chicago — Jet impingement has many technological applications because of its numerous merits, especially those related to the ability of liquids to carry away heat very efficiently. The present study introduces a new configuration* employing a wettability-patterning approach to divert an orthogonally-impinging laminar water jet onto a predetermined portion of the target surface. Diverging wettable tracks on a superhydrophobic background provide the means to re-direct the impinging jet along paths determined by the shape of these tracks on the solid surface. In a heat transfer example of this method, an open-surface heat exchanger is constructed and its heat transfer performance is characterized. Since this approach facilitates prolonged liquid contact with the underlying heated surface through thin-film spreading, evaporative cooling is also promoted. We demonstrate flow cases extracting 100 W/cm² at water flow rates of O(10 mL/min). By comparing with other jet-impingement cooling approaches, the present method provides roughly four times more efficient cooling by using less amount of coolant. The reduced coolant use, combined with the gravity-independent character of this technique, offer a new paradigm for compact heat transfer devices designed to operate in reduced- or zero-gravity environments. * T. P. Koukoravas, A. Ghosh, P. Sinha Mahapatra, R. Ganguly and C. M. Megaridis, Intl J. Heat Mass Transfer 95, 142-152, 2016.

10:53AM H11.00002 Spreading Characteristics of Newtonian Free Surface Liquid Jets Impacting a Moving Substrate , HATEF RAHMANI, YUCHEN GUO, SHELDON GREEN, Department of Mechanical Engineering, University of British Columbia, ALI VAKIL, Department of Mechanical Engineering, University of British Columbia and Coanda Research and Development Corp — The impingement of high-speed liquid jets on a solid substrate is salient to a number of industrial processes, including surface coating in the railroad industry. The impingement of Newtonian liquid jets is studied both experimentally and through simulation. On impingement the liquid jet spreads laterally from the impingement location to form a lamella that is then convected downstream, producing an overall U-shaped liquid surface. A variety of jet and substrate velocities, liquid viscosities, and jet diameters were studied. It is found that the lamella dimensions (width (W), radius (R), thickness (h)) vary with the jet Reynolds number (Re_{jet}) and vary inversely with the substrate Reynolds number (Re_{sub}). Interestingly, the ratio W/R is almost constant, independent of the jet viscosity, diameter, and speed, and also independent of the substrate speed. Furthermore, the lamella radius and width scale as $Re_{jet}/\sqrt{Re_{sub}}$ and the lamella thickness scales as $1/\sqrt{Re_{sub}}$. The experimental results were in good agreement with volume-of-fluid (VOF) CFD simulations, which implies that the simulations may be used to probe the physics of impingement.

11:06AM H11.00003 Numerical study of an impinging jet to a turbulent channel flow in a T-Junction configuration , MICHAIL GEORGIOU, MILTIADIS PAPALEXANDRIS, Université catholique de Louvain — In this talk we report on Large Eddy Simulations of an impinging planar jet to a turbulent channel flow in a T-Junction configuration. Due to its capacity for mixing and heat transfer enhancement, this type of flow is encountered in various industrial applications. In particular, our work is related to the emergency cooling systems of pressurized water reactors. As is well known, this type of flow is dominated by a large separation bubble downstream the jet impingement location. Secondary regions of flow separation are predicted both upstream and downstream the impinging jet. We describe how these separation regions interact with the shear layer that is formed by the injection of the jet to the crossflow, and how they affect the mixing process. In our talk we further examine the influence of the jet's velocity to characteristic quantities of the jet, such as penetration length and expansion angle, as well as to the first and second-order statistics of the flow.

11:19AM H11.00004 ABSTRACT WITHDRAWN –

11:32AM H11.00005 Magnetic Resonance Velocimetry analysis of an angled impinging jet, ALEXANDRE IRHOUD, French Military Academy of Saint-Cyr, MICHAEL BENSON, CLAIRE VERHULST, BRET VAN POPPEL, United States Military Academy, CHRIS ELKINS, Stanford University, DAVID HELMER, United States Military Academy — Impinging jets are used to achieve high heat transfer rates in applications ranging from gas turbine engines to electronics. Despite the importance and relative simplicity of the geometry, simulations historically fail to accurately predict the flow behavior in the vicinity of the flow impingement. In this work, we present results from a novel experimental technique, Magnetic Resonance Velocimetry (MRV), which measures three-dimensional time-averaged velocity without the need for optical access. The geometry considered in this study is a circular jet angled at 45 degrees and impinging on a flat plate, with a separation of approximately seven jet diameters between the jet exit and the impingement location. Two flow conditions are considered, with Reynolds numbers of roughly 800 and 14,000. Measurements from the MRV experiment are compared to predictions from Reynolds Averaged Navier Stokes (RANS) simulations, thus demonstrating the utility of MRV for validation of numerical analyses of impinging jet flow.

11:45AM H11.00006 The Time-Resolved Flow Field of a Spatially Oscillating Jet in Crossflow, F. OSTERMANN¹, R. WOSZIDLO², C.N. NAYERI³, C.O. PASCHEREIT⁴, Hermann-Föttinger-Institut, Technische Universität Berlin — Spatially oscillating jets in crossflow emitted by fluidic oscillators have been proven beneficial for flow control applications in recent studies. However, the driving mechanism behind the efficacy remains unknown. The presented study examines the fundamental, time-resolved flow field of a spatially oscillating jet in crossflow. The inclination angle between oscillation plane and crossflow is 90deg . The underlying experimental dataset is acquired plane-by-plane by a traversable stereoscopic particle image velocimetry system. Phase-averaging reduces stochastic noise, compensates low sampling rates, and allows combining the individual planes to a time-resolved three-dimensional flow field. The trajectory of the oscillating jet is much shallower than a steady jet. Two counter-rotating streamwise vortices are revealed. The sense of rotation is opposite to that of the counter-rotating vortex pair of steady jets in crossflow. This sense of rotation enables the vortices to prevail far downstream because they push each other toward the wall. The strength of the vortices is alternating. This vortex pair is a promising candidate to be the driving mechanism behind the high efficacy in separation control.

¹PhD Student

²Research Advisor

³Research Associate

⁴Professor, Chair of Fluid Dynamics

11:58AM H11.00007 Structural and mixing characteristics in actively controlled transverse jets¹, TAKESHI SHOJI, ANDREA BESNARD, ELIJAH HARRIS, ROBERT M'CLOSKEY, ANN KARAGOZIAN, UCLA, LUCA CORTELEZZI, Politecnico di Milano — These experiments explore the effect of external excitation on gaseous transverse jet (TJ) structural and mixing characteristics, emphasizing axisymmetric jet forcing. Sinusoidal as well as single and multiple square wave pulses, the latter with variable amplitudes, are explored for a range of jet-to-crossflow momentum flux ratios J , spanning regimes² of absolutely unstable upstream shear layers ($J < 10$) and convectively unstable shear layers ($J > 10$). The studies utilize acetone PLIF imaging of the jet, as done for unforced jets³. Axisymmetric forcing, irrespective of the waveform, can enhance cross-sectional symmetry of the TJ for convectively unstable conditions, but generally disrupts the usually symmetric counter-rotating vortex pair (CVP) observed for the absolutely unstable TJ. Conditions producing deeply penetrating, periodic vortical structures, such as square wave forcing at critical stroke ratios, increase jet spread, but do not always optimize molecular mixing. Creating multiple vortex structures of different strengths via multiple square pulses leads to enhanced interactions and accelerated vortex breakdown, potentially increasing mixing.

¹Supported by NSF (CBET-1437014) & AFOSR (FA9550-15-1-0261)

²Megerian, et al, JFM **593**, pp. 93-129, 2007

³Gevorkyan, et al, JFM **790**, pp. 237-274, 2016

12:11PM H11.00008 Analysis of the stability and sensitivity of jets in crossflow, MARC REGAN, KRISHNAN MAHESH, Univ of Minnesota - Twin Cities — Jets in crossflow (transverse jets) are a canonical fluid flow in which a jet of fluid is injected normal to a crossflow. A high-fidelity, unstructured, incompressible, DNS solver is shown (Iyer & Mahesh 2016) to reproduce the complex shear layer instability seen in low-speed jets in crossflow experiments. Vertical velocity spectra taken along the shear layer show good agreement between simulation and experiment. An analogy to countercurrent mixing layers has been proposed to explain the transition from absolute to convective stability with increasing jet to crossflow ratios. Global linear stability and adjoint sensitivity techniques are developed within the unstructured DNS solver in an effort to further understand the stability and sensitivity of jets in crossflow. An Arnoldi iterative approach is used to solve for the most unstable eigenvalues and their associated eigenmodes for the direct and adjoint formulations. Frequencies from the direct and adjoint modal analyses show good agreement with simulation and experiment. Development, validation, and results for the transverse jet will be presented. Supported by AFOSR (Iyer, P. S. & Mahesh, K. 2016 A numerical study of shear layer characteristics of low-speed transverse jets. J. Fluid Mech. 790, 275-307)

12:24PM H11.00009 ABSTRACT WITHDRAWN —

12:37PM H11.00010 Large bouncing jets¹, KARL CARDIN, MARK WEISLOGEL, Portland State University — We experimentally investigate the phenomena of large jet rebound (bounce), a mode of fluid transfer following oblique jet impacts on hydrophobic surfaces. We initially seek to describe the regimes of such jet bounce in tests conducted in the weightless environment of a drop tower. A parametric study reveals the dependence of the rebound mode on the relevant dimensionless groups such as Weber number We_{\perp} defined on the velocity component perpendicular to the surface. We show that significantly larger diameter jets behave similarly as much smaller jets demonstrated during previous terrestrial investigations when $We_{\perp} \approx 1$. For $We_{\perp} > 1$, large jet impacts create fishbone-like structures. We also explore rebounds from nonplanar substrates. Improving our understanding of such jet rebound opens avenues for unique transport capabilities.

¹NASA Cooperative Agreement NNX12A047A

Monday, November 21, 2016 10:40AM - 12:50PM —
Session H12 Multiphase Flows: Bubbly Flow C123 - Thomas Shepard, University of St. Thomas

10:40AM H12.00001 Influence of bubble size on effervescent atomization. Part 1: bubble characterization and mean spray features, TAYLOR LEWIS, THOMAS SHEPARD, DAVID FORLITI, University of St. Thomas — In the effervescent atomization process a gas-liquid bubbly mixture is ejected from a nozzle with the goal of enhancing liquid break-up. In this work, high speed images are taken of the bubbly flow inside of an effervescent atomizer as well as downstream of the atomizer exit. The use of varying porous plate media grades and channel inserts at the air injection site of the atomizer permitted independent control of mean bubble size. Digital image analyses were used for bubble characterization and measuring mean spray features. The roles of air injection geometry on bubble population parameters inside of the effervescent atomizer are detailed. The effect of bubble size is examined at multiple gas to liquid flow rate ratios for which the bubbly flow regime was maintained. Results are presented demonstrating the influence of bubble size on the average jet width, jet dark core length, and liquid break-up.

10:53AM H12.00002 Influence of bubble size on effervescent atomization. Part 2: unsteady spatial and temporal features, THOMAS SHEPARD, DAVID FORLITI, TAYLOR LEWIS, University of St. Thomas — In this work, high-speed images of the near-nozzle exit of an effervescent atomizer operating at low gas to liquid ratios are examined using proper orthogonal decomposition and digital image analyses. These techniques allow for the extraction of coherent spatial and temporal patterns present in the high-speed image sets. The effervescent atomizer was operated in the bubbly regime and the experimental facility allowed independent control over bubble size. The impact of varying the mean bubble size on the atomizer near exit field is presented at multiple gas to liquid flow rate ratios. The results demonstrate an influence of mean bubble diameter on peak instability frequency, instability amplitude, axial convection velocities and dominant mode structure.

11:06AM H12.00003 Experimental Characterization of Interchannel Mixing of Multiphase Flow Through a Narrow Gap, SIMO A. MÄKIHARJU, Univ. of California, JAMES W. GOSE, Univ. of Michigan, JOHN R. BUCHANAN JR., ALEXANDER G. MYCHKOVSKY, KIRK T. LOWE, Naval Nuclear Laboratory, STEVEN L. CECCIO, Univ. of Michigan — Two-phase mass transfer through a gap connecting two adjacent channels was investigated as a function of gap geometry and flow conditions. An experiment with a simplified geometry was conducted to aid in the physical understanding and to provide data for validation of numerical computations. The flow loop consisted of two $(127 \text{ mm})^2$ channels connected by a $1,219 \text{ mm}$ (L) $\times 229 \text{ mm}$ (W) gap, the height of which could be adjusted from 0 to 50 mm. The inlet Reynolds number in each channel could be independently varied from $4 \times 10^4 - 1 \times 10^5$. During previous experiments, the single phase mixing was extensively investigated. The inlet void fraction was varied from 1 to 20%. Gas was injected as nominally monodisperse bubbles with diameter $O(5 \text{ mm})$. The mass transfer through the gap was determined from measurements of the flow rates of water and air, and tracer concentration taken at channel inlets/outlets. The void fraction, bubble diameter distribution and gas flux was determined at the inlets based on flow rate measurements prior to gas injection, optical probes and Wire Mesh Sensor (WMS) data. At the outlets the gas fluxes were based on WMS measurements and the liquid phase mixing was determined based on measurement of the tracer concentration and liquid flow rate after separation of gas. Imaging of fluorescent tracer dye was utilized for select conditions to examine the dynamics of the mixing.

11:19AM H12.00004 Numerical Simulation of Bubbly Flows in an Aeration Tank with Biochemical Reactions, KHATEEB NOOR UL HUDA, KAZUYA SHIMIZU, University of Tokyo, XIAOBO GONG, Shanghai Jiao Tong University, SHU TAKAGI, University of Tokyo — For bubbly flow with biochemical reactions, all the analyses including overall fluid flow, bubble motion, bubble dissolution at local level and bacterial reactions/consumption of substrates are important. The developed system is provided by mixed Eulerian-Lagrangian formulation in which liquid media is represented in Eulerian system and bubbles are tracked individually. Murai and Matsumoto [1] developed a model to track bubbles to predict plume structure in finely dispersed domain. Gong et al. [2] developed the model further to include mass transfer, gas dissolution and mixing phenomenon entrained in this model. In this research we are using the model to include simulation of bacterial biochemical reactions for the purification of water and make it resemble as the wastewater purification tank. The gas bubble dissolution and mass transfer from gas to liquid phase is linked with biochemical reactions for an overall comprehensive study. The main area associated with this research is to incorporate all biochemical reactions in this bubbly flow based on situation of water and demand. In this particular study, various kinds of biomass and substrates are considered. A detailed model for biological wastewater purification involving reactions using bacteria's is developed and primary validation has been carried out based on experimental study. Finally, we tried to achieve physical optimization for this biochemical reactions. [1] Murai, et al., ASME-Publications-FED, 236, (1996), pp. 67-74. [2] Gong, X., et al., 2009, Int J Multiphas Flow, 35, pp.155-162.

11:32AM H12.00005 Bubble dynamics and bubble-induced turbulence of a single-bubble chain¹, JOOHYOUNG LEE, HYUNGMIN PARK, Seoul National University — In the present study, the bubble dynamics and liquid-phase turbulence induced by a chain of bubbles injected from a single nozzle have been experimentally investigated. Using a high-speed two-phase particle image velocimetry, measurements on the bubbles and liquid-phase velocity field are conducted in a transparent tank filled with water, while varying the bubble release frequency from 0.1 to 35 Hz. The tested bubble size ranges between 2.0-3.2 mm, and the corresponding bubble Reynolds number is 590-1100, indicating that it belongs to the regime of path instability. As the release frequency increases, it is found that the global shape of bubble dispersion can be classified into two regimes: from asymmetric (regular) to axisymmetric (irregular). In particular, at higher frequency, the wake vortices of leading bubbles cause an irregular behaviour of the following bubble. For the liquid phase, it is found that a specific trend on the bubble-induced turbulence appears in a strong relation to the above bubble dynamics. Considering this, we try to provide a theoretical model to estimate the liquid-phase turbulence induced by a chain of bubbles.

¹Supported by a grant funded by Samsung Electronics, Korea.

11:45AM H12.00006 The budget of turbulent kinetic energy in bubble plumes by acoustic Doppler velocimetry¹, CHRIS LAI, SCOTT SOCOLOFSKY, Texas A&M Univ — We present an experimental investigation on the TKE budget of a two-phase air-water bubble plume in an otherwise quiescent ambient. The required three-dimensional turbulent velocity field was measured by a profiling acoustic Doppler velocimeter. Experiments were carried out in a square water tank of 1m^3 and covered both adjustment phase ($z/D < 5$) and asymptotic regime ($z/D \geq 5$) of the plume in which the latter is characterized by a constant local Fr_p . The dynamic length scale D has previously been derived from a two-fluid approach and delineates the two regimes. Data on the mean flow establish the existence of an asymptotic regime when $z/D > 8$ with an entrainment coefficient of 0.095 and a Fr_p of 1.63. The data also corroborate well with previous measurements of large-scale bubble plumes. A budget of TKE was performed using curve-fits derived from the radial profiles of second- and third-order moments of turbulent velocities. From the budget, TKE production by bubbles was found to be larger than that by fluid shear. Approximately 55-60% of the total work done by bubbles is used to create fluid turbulence.

¹This research was made possible by a grant from The Gulf of Mexico Research Initiative to the Gulf Integrated Spill Research (GISR) Consortium.

11:58AM H12.00007 DNS assisted modeling of bubbly flows in vertical channels¹, GRETAR TRYGGVASON, MING MA, JIACAI LU, University of Notre Dame — The transient motion of bubbly flows, in vertical channels, is studied, using direct numerical simulations (DNS), where every continuum length- and time-scale are resolved. The results of several simulations, starting with laminar or turbulent liquid flows, including one with several hundred bubbles of different sizes and a friction Reynolds number of 500, are reviewed. At statistically steady state, nearly spherical bubbles in upflow form distinct wall-layers, but sufficiently deformable bubbles, as well as bubbles in downflow, do not. The transient evolution, particularly for nearly spherical bubbles in upflow is more complex. The bubbles first move toward the walls and then the liquid slowly slows down, eventually allowing some bubbles to return to the center of the channel. The use of the DNS results to help with modeling the average flow are discussed, including simple analytical models for laminar upflow and downflow and the use of statistical learning and neural networks to relate closure terms to resolved quantities for more complex flows. The prospects of using results from simulations of large system with bubbles of different sizes in turbulent flows for LES-like simulations are explored, including the simplification of the interface structure by filtering.

¹Research supported by DOE (CASL) and NSF.

12:11PM H12.00008 Numerical Simulation of Turbulent Bubbly Flow in a Vertical Square Duct, PRATAP VANKA, PURUSHOTAM KUMAR, KAI JIN, University of Illinois at Urbana-Champaign — We numerically investigate the dynamics of a large number of gas bubbles in a turbulent liquid flow in a confined vertical square duct, a problem of interest to many industrial equipment. The fluid flow is simulated by Direct Numerical Simulations and the motions of the bubbles are resolved by an accurate Volume of Fluid (VOF) technique. The flow is considered periodic in the streamwise direction with an imposed pressure gradient. The surface tension force is incorporated through a Sharp Surface Force (SSF) method that is observed to generate only very small spurious velocities at the interface. The algorithm has been programmed on a multiple-GPU computer in a data parallel mode. The turbulence driven secondary flows are first ensured to agree with previous DNS/LES by other researchers. A very fine grid with $192 \times 192 \times 768$ control volumes is used to resolve the liquid flow as well as 864 bubbles using 12 grid points across each bubble in all directions. The computations are carried out to 1.5 million time steps. It is seen that the bubbles preferentially migrate to walls, starting from a uniform layout. We present instantaneous and time mean velocities, turbulence statistics and compare them with unladen flow as well as with a bubbly flow in a planar channel.

12:24PM H12.00009 Direct Numerical Simulation of Insoluble Surfactant Effect on Turbulent Channel Bubbly Flows¹, JIACAI LU, GRETAR TRYGGVASON, University of Notre Dame — Direct Numerical Simulations (DNS) have been successfully used to obtain detailed data for turbulent channel bubbly flows. However, most of DNS that have been done so far remain problematic in comparing to most experiments. One of the major reasons is that real bubbly flows contain surfactants. The surfactants adhere to the interface, and produce an uneven distribution of the surfactant concentration due to the moving of bubbles and result in uneven surface tension over bubble surfaces. In this project, the effect of surfactants on the flow of many bubbles in an upward turbulent channel flow is studied by using of Direct Numerical Simulation with 3D Front-tracking method. The surfactant mass and the interfacial area are directly tracked in the method, and the surfactant mass remains conserved during the evolution. By using of different elasticity numbers in the non-linear equation of state which relates the surface tension to the surfactant concentration, the simulations show that the evolution of the turbulent channel bubbly flow are much different among the cases with contaminated bubbles and clean bubbles. Profiles of many parameters, such as streamwise velocity, shear stress and etc., are also compared at the statistically steady state for these cases.

¹Research supported by DOE (CASL).

12:37PM H12.00010 Bubbly flows around a two-dimensional circular cylinder¹, JUBEOM LEE, HYUNGMIN PARK, Seoul National University — Two-phase cross flows around a bluff body occur in many thermal-fluid systems like steam generators, heat exchangers and nuclear reactors. However, our current knowledge on the interactions among bubbles, bubble-induced flows and the bluff body are limited. In the present study, the gas-liquid bubbly flows around a solid circular cylinder are experimentally investigated while varying the mean void fraction from 5 to 27 %. The surrounding liquid (water) is initially static and the liquid flow is only induced by the air bubbles. For the measurements, we use the high-speed two-phase particle image velocimetry techniques. First, depending on the mean void fraction, two regimes are classified with different preferential concentration of bubbles in the cylinder wake, which are explained in terms of hydrodynamic force balances acting on rising bubbles. Second, the differences between the two-phase and single-phase flows (while matching their Reynolds numbers) around a circular cylinder will be discussed in relation to effects of bubble dynamics and the bubble-induced turbulence on the cylinder wake.

¹Supported by a grant (MPSS-CG-2016-02) through the Disaster and Safety Management Institute funded by Ministry of Public Safety and Security of Korean government.

Monday, November 21, 2016 10:40AM - 12:50PM –
Session H13 DFD GPC: Geophysical Fluid Dynamics: Oceans II General C124 - Chris Vogl, University of Washington

10:40AM H13.00001 ABSTRACT WITHDRAWN –

10:53AM H13.00002 Simulation of Earthquake-Generated Sea-Surface Deformation, CHRIS VOGL, RANDY LEVEQUE, University of Washington — Earthquake-generated tsunamis can carry with them a powerful, destructive force. One of the most well-known, recent examples is the tsunami generated by the Tohoku earthquake, which was responsible for the nuclear disaster in Fukushima. Tsunami simulation and forecasting, a necessary element of emergency procedure planning and execution, is typically done using the shallow-water equations. A typical initial condition is that using the Okada solution for a homogeneous, elastic half-space. This work focuses on simulating earthquake-generated sea-surface deformations that are more true to the physics of the materials involved. In particular, a water layer is added on top of the half-space that models the seabed. Sea-surface deformations are then simulated using the Clawpack hyperbolic PDE package. Results from considering the water layer both as linearly elastic and as nearly incompressible are compared to that of the Okada solution.

11:06AM H13.00003 Airborne infrared remote sensing characterization of submesoscale eddies, GEOFFREY SMITH, GEORGE MARMORINO, W. DAVID MILLER, Naval Research Laboratory, RYAN NORTH, INGRID ANGEL-BENAVIDES, BURCKARD BASCHEK, Helmholtz-Zentrum Geesthacht Centre for Materials and Coastal Research — Airborne remote sensing surveys off Santa Catalina Island, CA ($33^{\circ}30'N$ $118^{\circ}31'W$) were conducted as part of a larger study of the occurrence and behavior of submesoscale phenomena. This builds upon previous work by DiGiacomo and Holt, who utilized SAR imagery to characterize the size and distribution of predominately cyclonic 'spiral eddies' in the Southern California Bight. In the present work the thermal surface expression of a single cyclonic eddy captured in February 2013 will be investigated. Advances made in methods to estimate eddy circulation and vorticity directly from the thermal imagery will be discussed and compared with *in situ* measurements. Inferences about localized mixing and flow instabilities can also be drawn from the imagery, and these too will be discussed in the context of *in situ* data. A simple model will be offered describing the three dimensional flow in the core of the eddy and how that can be used to explain the surface imagery. Connections between the signatures surrounding the eddy and the core itself will also be discussed in the context of the model.

11:19AM H13.00004 Predictability of the Dynamic Mode Decomposition in Coastal Processes, RUO-QIAN WANG, University of California, Berkeley, LIV HERDMAN, United States Geological Survey, MARK STACEY, University of California, Berkeley, PATRICK BARNARD, United States Geological Survey — Dynamic Mode Decomposition (DMD) is a model order reduction technique that helps reduce the complexity of computational models. DMD is frequently easier to interpret physically than the Proper Orthogonal Decomposition. The DMD can also produce the eigenvalues of each mode to show the trend of the mode, establishing the rate of growth or decay, but the original DMD cannot produce the contributing weights of the modes. The challenge is selecting the important modes to build a reduced order model. DMD variants have been developed to estimate the weights of each mode. One of the popular methods is called Optimal Mode Decomposition (OMD). This method decomposes the data matrix into a product of the DMD modes, the diagonal weight matrix, and the Vandermonde matrix. The weight matrix can be used to rank the importance of the mode contributions and ultimately leads to the reduced order model for prediction and controlling purpose. We are currently applying DMD to a numerical simulation of the San Francisco Bay, which features complicated coastal geometry, multiple frequency components, and high periodicity. Since DMD defines modes with specific frequencies, we expect DMD would produce a good approximation, but the preliminary results show that the predictability of the DMD is poor if unimportant modes are dropped according to the OMD. We are currently testing other DMD variants and will report our findings in the presentation.

11:32AM H13.00005 The response of the Ocean Surface Boundary Layer and Langmuir turbulence to tropical cyclones, DONG WANG, TOBIAS KUKULKA, Univ of Delaware, BRANDON REICHL¹, TETSU HARA, ISAAC GINIS, Univ of Rhode Island — The interaction of turbulent ocean surface boundary layer (OSBL) currents and the surface waves' Stokes drift generates Langmuir turbulence (LT), which enhances OSBL mixing. This study investigates the response of LT to extreme wind and complex wave forcing under tropical cyclones (TCs), using a large eddy simulation (LES) approach based on the wave-averaged Navier-Stokes equations. We simulate the OSBL response to TC systems by imposing the wind forcing of an idealized TC storm model, covering the entire horizontal extent of the storm systems. The Stokes drift vector that drives the wave forcing in the LES is determined from realistic spectral wave simulations forced by the same wind fields. We find that the orientations of Langmuir cells are vertically uniform and aligned with the wind in most regions despite substantial wind-wave misalignment in TC conditions. LT's penetration depth is related to Stokes drift depth and limited by OSBL depth. A wind-projected surface layer Langmuir number is proposed and successfully applied to scale turbulent vertical velocity variance in extreme TC conditions.

¹Current affiliation: Princeton University/NOAA GFDL

11:45AM H13.00006 Laboratory experiments with a buoyancy forced circulation in a rotating basin, CATHERINE VREUGDENHIL, ROSS GRIFFITHS, BISHAKHADATTA GAYEN, Australian National University — We consider the relative influence of buoyancy forcing and Coriolis effects on convection forced by a differential in heating at a horizontal surface in a rectangular basin. Laboratory experiments with water are reported for a rotating *f*-plane basin and a range of Ekman number $E = 2 \times 10^{-7} - 1 \times 10^{-5}$. Heating is applied over half of the base as a uniform flux and cooling applied over the other half as a uniform temperature, resulting in a flux Rayleigh number $Ra_F = O(10^{14})$ large enough to ensure turbulent convection, where Ra_F defined in terms of domain length L . Compared to the non-rotating circulation where Nusselt number (a measure of the convective to conductive heat transfer) scales as $Nu \sim Ra_F^{1/6}$, the strongly rotating regime is determined by a geostrophic balance of the larger scales of horizontal flow in the inviscid thermal boundary with $Nu \sim Ro^{1/6}$, where $Ro = B^{1/2}/(f^{3/2}L)$ is the natural Rossby number (B is buoyancy flux per unit area and f is Coriolis parameter). We also find evidence for a further transition into a regime where the circulation is dominated by deep 'chimney' convection in a field of small vertical plumes and Nu is more weakly dependent on rotation.

11:58AM H13.00007 Turbulent mixing at high Schmidt number: new results from a hybrid spectral compact finite difference and dual grid resolution approach¹, M.P. CLAY, P.K. YEUNG, Georgia Tech, T. GOTOH, Nagoya Inst Tech, Japan — Turbulent mixing at high Schmidt number (Sc) (low molecular diffusivity) is characterized by fluctuations that arise at sub-Kolmogorov scales and are hence difficult to resolve or measure. Simulations in the recent past have provided some basic results but were still limited in either the Reynolds number or the Schmidt number. We have developed a massively parallel implementation of a hybrid pseudo-spectral and combined compact finite difference technique [Gotoh et al. *J. Comput Phys.* 231, 7398-7414 (2012)] where the velocity and scalar fields are computed at different grid resolutions (the latter up to 8192^3). A specific target is the scalar field maintained by a uniform mean gradient at Taylor-scale Reynolds number 140 and $Sc = 512$, which is comparable to the value (700) for salinity in the ocean. Preliminary results at moderately high Sc are in support of Batchelor (k^{-1}) scaling for the spectrum in the viscous-convective range, followed by exponential fall-off in the viscous-diffusive range. Data over a wide range of Reynolds and Schmidt numbers are used to examine the approach to local isotropy and a saturation of intermittency suggested by previous work.

¹Supported by NSF Grant ACI-1036170 and a subaward via UIUC

12:11PM H13.00008 The role of turbulence driven by tidal and librational forcing in planetary fluid layers., ALEXANDER GRANNAN, University of California-Los Angeles, BENJAMIN FAVIER, Aix-Marseille Universite, CNRS, Ecole Centrale Marseille, IRPHE UMR 7342, BRUCE BILLS, Jet Propulsion Laboratory, Caltech, Pasadena, CA, USA, MICHAEL LE BARS, Aix-Marseille Universite, CNRS, Ecole Centrale Marseille, IRPHE UMR 7342, JONATHAN AURNOU, University of California-Los Angeles — The turbulence generated in the liquid metal cores and oceans of planetary bodies can have profound effects on energy dissipation and magnetic field generation. An important driver of such turbulence is mechanical forcing from precession, libration, and tidal forcing. On Earth, such forcing mechanisms in the oceans are crucial but the role that such forcings play for other planetary bodies also possessing oceans and liquid metal cores are not generally considered. Recent laboratory experimental and numerical studies of Grannan et al. Phys. Fluids 2014, Favier et al. Phys. Fluids 2015, and Grannan et al. Geophys. J. Int. 2016 have shown that turbulent flow is driven by an elliptic instability which is a triadic resonance between two inertial modes and the base flow. Based on the most recent work, a generalized scaling law for the saturated r.m.s. velocity is found, $U \sim \beta$, where β is the dimensionless equatorial ellipticity of the body. Using planetary values for tidal and librational forcing parameters, we argue that mechanically forced turbulent flows can play a significant role in dissipative processes, mixing, and magnetic field generation.

12:24PM H13.00009 Dimensions of continents and oceans – water has carved a perfect cistern, JOHN A WHITEHEAD, Woods Hole Ocean Inst — The ocean basins have almost exactly the correct surface area and average depth to hold Earth's water. Two processes are responsible for this. First, Earth's continental crust is thinned by erosion so that average elevation is a few hundred meters above sea level. Second, the crust is thickened by lateral compression from mountain formation and sediments and water lost in subduction is resupplied at least in part by volcanics. The resulting continents are approximately tabular in cross section, resulting in the well-known double hypsometric curve for Earth's elevation. Therefore, erosion and mountain building have enabled water to carve its own cistern in the form of all the ocean basins. A theoretical fluid model, suggested partly by laboratory experiments, produces such a tabular continent with a surface above sea level. A simple hydrostatic balance gives a first approximation for the average depth and area of oceans and continents for present Earth as a function of material volumes and densities. Using a wide range of possible crust volumes with the present water volume, the average continental crust thickness exceeds 22 km and ocean area exceeds 25% of the globe. Other volumes of water produce a wide range of areas and depths of oceans and crust.

12:37PM H13.00010 Ocean acidification: Towards a better understanding of calcite dissolution, MONICA M. WILHELMUS, University of California Riverside, JESS ADKINS, California Institute of Technology, DIMITRIS MENEMENLIS, Jet Propulsion Laboratory — The drastic increase of anthropogenic CO_2 emissions over the past two centuries has altered the chemical structure of the ocean, acidifying upper ocean waters. The net impact of this pH decrease on marine ecosystems is still unclear, given the unprecedented rate at which CO_2 is being released into the atmosphere. As part of the carbon cycle, calcium carbonate dissolution in sediments neutralizes CO_2 : phytoplankton at the surface produce carbonate minerals, which sink and reach the seafloor after the organisms die. On time scales of thousands of years, the calcium carbonate in these shells ultimately reacts with CO_2 in seawater. Research in this field has been extensive; nevertheless, the dissolution rate law, the impact of boundary layer transport, and the feedback with the global ocean carbon cycle remain controversial. Here, we (i) develop a comprehensive numerical framework via 1D modeling of carbonate dissolution in sediments, (ii) approximate its impact on water column properties by implementing a polynomial approximation to the system's response into a global ocean biogeochemistry general circulation model (OBGCM), and (iii) examine the OBGCM sensitivity response to different formulations of sediment boundary layer properties. We find that, even though the burial equilibration time scales of calcium carbonate are in the order of thousands of years, the formulation of a bottom sediment model along with an improved description of the dissolution rate law can have consequences on multi-year to decadal time scales.

Monday, November 21, 2016 10:40AM - 12:37PM –
Session H14 Free Surface Flows: Water Entrance and Splashes C125-126 - Christine Ikeda, Virginia Polytechnic Institute and State University

10:40AM H14.00001 On the Effect of Structural Response on the Hydrodynamic Loading of a Free-Falling Wedge¹, CHRISTINE IKEDA, Virginia Polytechnic Institute and State University, 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 theoretical prediction. The experimental program consisted of two 20° deadrise angle wedges dropped from a range of heights, $0.15 \leq H \leq 0.6$ m, while pressures and accelerations of the slam were measured. The first wedge had a rigid bottom, and the second wedge had a flexible bottom. Both experiments are compared with a non-linear boundary value flat cylinder theory in order to determine the effects of flexibility on the hydrodynamic pressure. 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.

¹This work is funded by the Office of Naval Research and the state of Louisiana Board of Regents Industrial Ties and Research Subprogram.

10:53AM H14.00002 Crossing the boundary: experimental investigation of water entry conditions of V-shaped wedges¹, TINGBEN XIAO, DANIEL YOHANN, LIONEL VINCENT, University of Southern California, SUNGHWAN JUNG, Virginia Tech, EVA KANSO, University of Southern California — Seabirds that plunge-dive at high speeds exhibit remarkable abilities to withstand and mitigate impact forces. To minimize these forces, diving birds streamline their shape at impact, entering water with their sharp beak first. Here, we investigate the impact forces on rigid V-shaped wedges crossing the air-water interface at high Weber numbers. We vary the impact velocity V by adjusting the height from which the wedge is dropped. Both a high-speed camera and a force transducer are used to characterize the impact. We found that the splash base and air cavity show little dependence on the impact velocity when rescaling by inertial time d/V , where d is the breadth of the wedge. The peak impact force occurs at time t_p smaller than the submersion time t_s such that the ratio t_p/t_s is almost constant for all wedges and impact velocities V . We also found that the maximum impact force, like drag force, scales as AV^2 , where A is the cross-sectional area of the wedge. We then propose analytical models of the impact force and splash dynamics. The theoretical predictions agree well with our experimental results. We conclude by commenting on the relevance of these results to understanding the mechanics of diving seabirds.

¹We acknowledge support from the National Science Foundation.

11:06AM H14.00003 Bubble baths: just splashing around?, WESLEY ROBINSON, NATHAN SPEIRS, SABERUL ISLAM SHARKER, RANDY HURD, BJ WILLIAMS, TADD TRUSCOTT, Utah State University — Soap Bubbles on the water surface would seem to be an intuitive means for splash suppression, but their presence appears to be a double edged sword. We present on the water entry of hydrophilic spheres where the liquid surface is augmented by the presence of a bubble layer, similar to a bubble bath. While the presence of a bubble layer can diminish splashing upon impact at low Weber numbers, it also induces cavity formation at speeds below the critical velocity. The formation of a cavity generally results in larger Worthington jets and thus, larger amounts of ejected liquid. Bubble layers induce cavity formation by wetting the sphere prior to liquid impact, causing them to form cavities similar to those created by hydrophobic spheres. Droplets present on a pre-wetted sphere disrupt the flow of the advancing liquid during entry, pushing it away from the impacting body to form an entrained air cavity. This phenomena was noted by Worthington with pre-wetted stone marbles, and suggests that the application of a bubble layer is generally ineffective as a means of splash suppression.

11:19AM H14.00004 Make water entry great again!¹, RANDY HURD, Utah State University, JESSE BELDEN, MICHAEL JANDRON, Naval Undersea Warfare Center, TATE FANNING, Brigham Young University, TADD TRUSCOTT, Utah State University — Upon free surface impact, silicone rubber spheres deform significantly and begin to vibrate producing unique nested cavities. We show that sphere deformation and cavity formation can be characterized by material shear modulus, density and impact velocity. Additionally, material vibration scales with sphere diameter and material wave speed. Applying a modified diameter, which reflects deformation, effectively collapses experimental pinch-off data with Froude number. A scaling argument shows that a deformable sphere loses energy proportional to the vibrational period of the sphere in the first stages of impact. The effective force coefficient of a deformable sphere through impact is nearly identical to a rigid sphere with the same solid-liquid density ratio. The scaling predicts how the cavity and projectile dynamics of a deformable sphere differs from a rigid counterpart.

¹Office of Naval Research, Navy Undersea Research Program (grant N0001414WX00811)

11:32AM H14.00005 Stable, streamlined and helical cavity formation by the impact of Leidenfrost spheres, MOHAMMAD MANSOOR, KAUST & USU, IVAN VAKARELSKI, KAUST, JEREMY MARSTON, Texas Tech University, TADD TRUSCOTT, Utah State University, SIGURDUR THORODDSEN, KAUST — This work reports results from an experimental study on the formation of stable-streamlined and helical cavity wakes following the free-surface impact of Leidenfrost spheres. The Leidenfrost effect encapsulates the sphere by a vapor layer to prevent any physical contact with the surrounding liquid. This phenomenon is essential for the pacification of acoustic rippling along the cavity interface to result in a stable-streamlined cavity wake. Such a streamlined configuration experiences drag coefficients an order of magnitude lower than those acting on room temperature spheres. A striking observation is the formation of helical cavities which occur for impact Reynolds numbers $Re_0 \geq 1.4 \times 10^5$ and are characterized by multiple interfacial ridges, stemming from and rotating synchronously about an evident contact line around the sphere equator. This helical configuration has 40 – 55 % smaller overall force coefficients than those obtained in the formation of stable cavity wakes.

11:45AM H14.00006 Aerodynamic shapes of two-dimensional splashes, LIONEL VINCENT, TINGBEN XIAO, DANIEL YOHANN, EVA KANSO, University of Southern California — We investigate experimentally the long-term evolution of a splash induced by the water entry of 90-degree wedge of breadth d and length L such that $L \gg d$. We find that for large speed entry speed V , the splash shows both concave and convex curvature, as opposed to single concave curvature observed for low speed. This peculiarity is found to be the result of a kink generated by the initial dynamics, the growth of which is favored by aerodynamics efforts ρV^2 , where ρ designates the density of air, and inhibited by surface tension effects scaling as σ/d . The transition between simply-curved splash and doubly-curved splash is found to happen for Weber number $\rho V^2 d / \sigma \simeq 1$. Doubly-curved splash sheets undergo significant stretch during their life span, altering the breaking up process.

11:58AM H14.00007 An Experimental Study of Droplets Produced by a Plunging Breakers¹, MARTIN ERININ, DAN WANG, DAVID TOWLE, XINAN LIU, JAMES DUNCAN, University of Maryland — In this study, the production of droplets by a mechanically generated plunging breaking water wave is investigated in a wave tank. The breaker, with an amplitude of 0.070 m, is generated repeatedly with a programmable wave maker by using a dispersively focused wave packet (average frequency 1.15 Hz). The profile histories of the breaking wave crests along the center plane of the tank are measured using cinematic laser-induced fluorescence. The droplets are measured using a cinematic digital in-line holographic system positioned at 30 locations along a horizontal plane that is 1 cm above the maximum wave crest height. This measurement plane covers the entire region in the tank where the wave breaks. The holographic system is used to obtain the droplet diameters (d , for $d > 100$ microns) and the three components of the droplet velocities. From these measurements and counting only the droplets that are moving up, the spatio-temporal distribution of droplet generation by the breaking wave is obtained. The main features of the droplet generation are correlated with the features and phases of the breaking process.

¹The support of the National Science Foundation under grant OCE0751853 from the Division of Ocean Sciences is gratefully acknowledged.

12:11PM H14.00008 Flow-induced oscillations of a floating moored cylinder¹, DANIEL CARLSON, YAHYA MODARRES-SADEGHI, Univ of Mass - Amherst — An experimental study of flow-induced oscillations of a floating model spar buoy was conducted. The model spar consisted of a floating uniform cylinder moored in a water tunnel test section, and free to oscillate about its mooring attachment point near the center of mass. For the bare cylinder, counter-clockwise (CCW) figure-eight trajectories approaching $A^*=1$ in amplitude were observed at the lower part of the spar for a reduced velocity range of $U^*=4\text{--}11$, while its upper part experienced clockwise (CW) orbits. It was hypothesized that the portion of the spar undergoing CCW figure eights is the portion within which the flow excites the structure. By adding helical strakes to the portion of the cylinder with CCW figure eights, the response amplitude was significantly reduced, while adding strakes to portions with clockwise orbital motion had a minimal influence on the amplitude of response.

¹This work is partially supported by the NSF-sponsored IGERT: Offshore Wind Energy Engineering, Environmental Science, and Policy (Grant Number 1068864).

12:24PM H14.00009 Critical Steady Surface Waves of Idea Fluid over a Bump with Surface Tension, JEONGWHAN CHOI, SANGWON LEE, JOONKYOUNG KIM, Korea University, SUNGIM WHANG, Ajou University — The paper deals with steady forced surface waves propagating on a two-dimensional incompressible and inviscid fluid with a small bump placed on a rigid flat bottom. If the surface tension coefficient T on the free surface is not zero and the wave is moving with a constant speed C , the wave motion is determined by two non-dimensional constants, $F = gh$ and $\lambda = T/(gh^2)$, where g is the gravity constant and h is the height of the fluid at infinity. It has been known that $F = 1$ and $\lambda = 1/3$ are the critical values of F and λ , respectively. In the critical case $F = 1 + \lambda$ and $\lambda = 1/3 + \lambda$ with $\lambda > 0$ a small parameter, a time-dependent forced Kawahara (F-Kawahara) equation is derived to model the wave propagation on the free surface and the steady F-Kawahara equation is studied both theoretically and numerically. It is shown that the steady F-Kawahara equation has many different kinds of one and multi-hump solutions when F and λ vary. In particular, for a fixed F , there is a λ_0 such that if $\lambda > \lambda_0$, two one-hump steady solutions can be obtained, one with small amplitude and the other with large amplitude. By using the unsteady F-Kawahara equation, it appears that the small one-hump solution is stable while the large one is unstable. In addition, two-hump solutions are unstable.

Monday, November 21, 2016 10:40AM - 12:50PM – Session H15 Bio: Cardiovascular Flow II E143/144 - Erica Cherry Kemmerling, Tufts University

10:40AM H15.00001 The effect of inlet boundary conditions in image-based CFD modeling of aortic flow, SUDHARSHAN MADHAVAN, ERICA CHERRY KEMMERLING, Department of Mechanical Engineering, Tufts University — CFD of cardiovascular flow is a growing and useful field, but simulations are subject to a number of sources of uncertainty which must be quantified. Our work focuses on the uncertainty introduced by the selection of inlet boundary conditions in an image-based, patient-specific model of the aorta. Specifically, we examined the differences between plug flow, fully developed parabolic flow, linear shear flows, skewed parabolic flow profiles, and Womersley flow. Only the shape of the inlet velocity profile was varied—all other parameters were held constant between simulations, including the physiologically realistic inlet flow rate waveform and outlet flow resistance. We found that flow solutions with different inlet conditions did not exhibit significant differences beyond 1.75 inlet diameters from the aortic root. Time averaged wall shear stress (TAWSS) was also calculated. The linear shear velocity boundary condition solution exhibited the highest spatially averaged TAWSS, about 2.5% higher than the fully developed parabolic velocity boundary condition, which had the lowest spatially averaged TAWSS.

10:53AM H15.00002 Hemodynamics of the Aortic Jet and Implications for Detection of Aortic Stenosis Murmurs¹, CHI ZHU, JUNGHEE SEO, HANI BAKHSHAEE, RAJAT MITTAL, Johns Hopkins University — Cardiac auscultation with a stethoscope has served as the primary method for qualitative screening of cardiovascular conditions for over a hundred years. However, a lack of quantitative understanding of the flow mechanism(s) responsible for the generation of the murmurs, as well as the effect of intervening tissue on the propagation of these murmurs has been a significant limiting factor in the advancement of automated cardiac auscultation. In this study, a multiphysics computational modeling approach is used to investigate these issues. Direct numerical simulation (DNS) is used to explore the fluid dynamics of the jets formed at the aortic valve and the pressure fluctuations generated by the interaction of this jet with the aortic wall. Subsequently, structural wave propagation in the tissue is resolved by a high-order, linear viscoelastic wave solver in order to explore the propagation of the murmurs through a tissue-like material. The implications of these results for cardiac auscultation are discussed.

¹The authors would like to acknowledge the financial support from NSF Grants IIS-1344772, CBET-1511200, and computational resource by XSEDE NSF Grant TG-CTS100002.

11:06AM H15.00003 Hemodynamics of physiological blood flow in the aorta with nonlinear anisotropic heart valve¹, FOTIS SOTIROPOULOS, College of Engineering and Applied Sciences, Stony Brook University, ANVAR GILMANOV, HENRYK STOLARSKI, Univ of Minn - Minneapolis — The hemodynamic blood flow in cardiovascular system is one of the most important factor, which causing several vascular diseases. We developed a new Curvilinear Immersed Boundary – Finite Element – Fluid Structure Interaction (CURVIB-FE-FSI) method to analyze hemodynamic of pulsatile blood flow in a real aorta with nonlinear anisotropic aortic valve at physiological conditions. Hyperelastic material model, which is more realistic for describing heart valve have been incorporated in the CURVIB-FE-FSI code to simulate interaction of aortic heart valve with pulsatile blood flow. Comparative studies of hemodynamics for linear and nonlinear models of heart valve show drastic differences in blood flow patterns and hence differences of stresses causing impact at leaflets and aortic wall.

¹This work is supported by the Lillehei Heart Institute at the University of Minnesota.

11:19AM H15.00004 Flow Dynamics of Contrast Dispersion in the Aorta, PARASTOU ESLAMI¹, JUNG-HEE SEO, Johns Hopkins University, MARCUS CHEN, National Institute of Health, RAJAT MITTAL, Johns Hopkins University — The time profile of the contrast concentration or arterial input function (AIF) has many fundamental clinical implications and is of importance for many imaging modalities and diagnosis such as MR perfusion, CT perfusion and CT angiography (CTA). Contrast dispersion in CTA has been utilized to develop a novel method- Transluminal Attenuation Flow Encoding (TAFE)- to estimate coronary blood flow (CBF). However, in clinical practice, AIF is only available in the descending aorta and is used as a surrogate of the AIF at the coronary ostium. In this work we use patient specific computational models of the complete aorta to investigate the fluid dynamics of contrast dispersion in the aorta. The simulation employs a realistic kinematic model of the aortic valve and the dispersion patterns are correlated with the complex dynamics of the pulsatile flow in the curved aorta. The simulations allow us to determine the implications of using the descending aorta AIF as a surrogate for the AIF at the coronary ostium. PE is supported by the NIH Individual Partnership Program. -/abstract- Category: 4.7.1: Biological fluid dynamics: Physiological - Cardiovasc

¹This work was done at Johns Hopkins University.

11:32AM H15.00005 Single camera volumetric velocimetry in aortic sinus with a percutaneous valve, CHRIS CLIFFORD, BRIAN THUROW, Auburn University, PREM MIDHA, IKECHUKWU OKAFOR, VRISHANK RAGHAV, AJIT YOGANATHAN, Georgia Institute of Technology — Cardiac flows have long been understood to be highly three dimensional, yet traditional in vitro techniques used to capture these complexities are costly and cumbersome. Thus, two dimensional techniques are primarily used for heart valve flow diagnostics. The recent introduction of plenoptic camera technology allows for traditional cameras to capture both spatial and angular information from a light field through the addition of a microlens array in front of the image sensor. When combined with traditional particle image velocimetry (PIV) techniques, volumetric velocity data may be acquired with a single camera using off-the-shelf optics. Particle volume pairs are reconstructed from raw plenoptic images using a filtered refocusing scheme, followed by three-dimensional cross-correlation. This technique was applied to the sinus region (known for having highly three-dimensional flow structures) of an in vitro aortic model with a percutaneous valve. Phase-locked plenoptic PIV data was acquired at two cardiac outputs (2 and 5 L/min) and 7 phases of the cardiac cycle. The volumetric PIV data was compared to standard 2D-2C PIV. Flow features such as recirculation and stagnation were observed in the sinus region in both cases.

11:45AM H15.00006 Simulations of blood flow in patient-specific aortic dissections with a deformable wall model, KATHRIN BAEUMLER, VIJAY VEDULA, ANNA SAILER KARMANN, ALISON MARSDEN, DOMINIK FLEISCHMANN, Stanford University — Aortic dissection is a life-threatening condition in which blood penetrates into the vessel wall, creating a second flow channel, often requiring emergency surgical repair. Up to 50% of patients who survive the acute event face late complications like aortic dilatation and eventual rupture. Prediction of late complications, however, remains challenging. We therefore aim to perform accurate and reliable patient-specific simulations of blood flow in aortic dissections, validated by 4D-Flow MRI. Among other factors, this is a computational challenge due to the compliance of the vessel walls and the large degree of membrane deformation between the two flow channels. We construct an anatomic patient-specific model from CT data including both flow channels and the membrane between them. We then run fluid structure interaction simulations using an arbitrary Lagrangian-Eulerian (ALE) formulation within a multiscale variational framework, employing stabilized finite element methods. We compare hemodynamics between a rigid and a deformable wall model and examine membrane dynamics and pressure differences between the two flow channels. The study focuses on the computational and modeling challenges emphasizing the importance of employing a deformable wall model for aortic dissections.

11:58AM H15.00007 Uncertainty Quantification in Multi-Scale Coronary Simulations Using Multi-resolution Expansion, JUSTIN TRAN, DANIELE SCHIAVAZZI, ABHAY RAMACHANDRA, Stanford University, ANDREW KAHN, University of California, San Diego, ALISON MARSDEN, Stanford University — Computational simulations of coronary flow can provide non-invasive information on hemodynamics that can aid in surgical planning and research on disease propagation. In this study, patient-specific geometries of the aorta and coronary arteries are constructed from CT imaging data and finite element flow simulations are carried out using the open source software SimVascular. Lumped parameter networks (LPN), consisting of circuit representations of vascular hemodynamics and coronary physiology, are used as coupled boundary conditions for the solver. The outputs of these simulations depend on a set of clinically-derived input parameters that define the geometry and boundary conditions, however their values are subjected to uncertainty. We quantify the effects of uncertainty from two sources: uncertainty in the material properties of the vessel wall and uncertainty in the lumped parameter models whose values are estimated by assimilating patient-specific clinical and literature data. We use a generalized multi-resolution chaos approach to propagate the uncertainty. The advantages of this approach lies in its ability to support inputs sampled from arbitrary distributions and its built-in adaptivity that efficiently approximates stochastic responses characterized by steep gradients.

12:11PM H15.00008 Computational fluid dynamics comparisons of wall shear stress in patient-specific coronary artery bifurcation using coronary angiography and optical coherence tomography.¹, ERIC POON, VIKAS THONDAPU, CHENG CHIN, CEDRIC SCHEERLINCK, TONY ZAHTILA, CHRIS MAMON, WILSON NGUYEN, ANDREW OOI, Univ of Melbourne, PETER BARLIS, Northern Health, Australia — Blood flow dynamics directly influence biology of the arterial wall, and are closely linked with the development of coronary artery disease. Computational fluid dynamics (CFD) solvers may be employed to analyze the hemodynamic environment in patient-specific reconstructions of coronary arteries. Although coronary X-ray angiography (CA) is the most common medical imaging modality for 3D arterial reconstruction, models reconstructed from CA assume a circular or elliptical cross-sectional area. This limitation can be overcome with a reconstruction technique fusing CA with intravascular optical coherence tomography (OCT). OCT scans the interior of an artery using near-infrared light, achieving a 10-micron resolution and providing unprecedented detail of vessel geometry. We compared 3D coronary artery bifurcation models generated using CA alone versus OCT-angiography fusion. The model reconstructed from CA alone is unable to identify the detailed geometrical variations of diseased arteries, and also under-estimates the cross-sectional vessel area compared to OCT-angiography fusion. CFD was performed in both models under pulsatile flow in order to identify and compare regions of low wall shear stress, a hemodynamic parameter directly linked with progression of atherosclerosis.

¹Supported by ARC LP150100233 and VLSCI VR0210

12:24PM H15.00009 Effects of vascular structures on the pressure drop in stenotic coronary arteries¹, JAERIM KIM, HAECHON CHOI, Seoul National University, JIHOON KWEON, YOUNG-HAK KIM, DONG HYUN YANG, NAMKUG KIM, University of Ulsan College of Medicine, Asan Medical Center — A stenosis, which is a narrowing of a blood vessel, of the coronary arteries restricts the flow to the heart and it may lead to sudden cardiac death. Therefore, the accurate determination of the severity of a stenosis is a critical issue. Due to the convenience of visual assessments, geometric parameters such as the diameter stenosis and area stenosis have been used, but the decision based on them sometimes under- or overestimates the functional severity of a stenosis, i.e., pressure drop. In this study, patient-specific models that have similar area stenosis but different pressure drops are considered, and their geometries are reconstructed from the coronary computed tomography angiography (CCTA). Both steady and pulsatile inflows are considered for the simulations. Comparison between two models that have a bifurcation right after a stenosis shows that the parent to daughter vessel angle results in different secondary flow patterns and wall shear stress distributions which affect the pressure downstream. Thus, the structural features of the lower and upper parts of a stenosis significantly affect the pressure drop.

¹Supported by 20152020105600

12:37PM H15.00010 Morphology of drying blood pools.¹, NICK LAAN, Institut de Recherche Criminelle de la Gendarmerie Nationale, FIONA SMITH, Aix-Marseille University, CELINE NICLOUX, Institut de Recherche Criminelle de la Gendarmerie Nationale, DAVID BRUTIN, Aix-Marseille University, D-BLOOD PROJECT COLLABORATION — Often blood pools are found on crime scenes providing information concerning the events and sequence of events that took place on the scene. However, there is a lack of knowledge concerning the drying dynamics of blood pools. This study focuses on the drying process of blood pools to determine what relevant information can be obtained for the forensic application. We recorded the drying process of blood pools with a camera and measured the weight. We found that the drying process can be separated into five different: coagulation, gelation, rim desiccation, centre desiccation, and final desiccation. Moreover, we found that the weight of the blood pool diminishes similarly and in a reproducible way for blood pools created in various conditions. In addition, we verify that the size of the blood pools is directly related to its volume and the wettability of the surface. Our study clearly shows that blood pools dry in a reproducible fashion. This preliminary work highlights the difficult task that represents blood pool analysis in forensic investigations, and how internal and external parameters influence its dynamics. We conclude that understanding the drying process dynamics would be advancement in timeline reconstitution of events.

¹ANR funded project: D-Blood Project

Monday, November 21, 2016 10:40AM - 12:50PM –
Session H16 Drops: Splash and Shatter D133/134 - Sigurdur Thoroddsen, King Abdullah University of Science and Technology

10:40AM H16.00001 Experiments on the breakup of drop-impact crowns by Marangoni holes, ABDULRAHMAN ALJEDAANI, C. L. WANG, A. JETLY, E. Q. LI, S. T. THORODDSEN, King Abdullah University of Science and Technology, Thuwal, Saudi Arabia — High-speed video experiments investigate the crown break up due to Marangoni instability when a highly viscous drop impacts on a thin layer of lower-viscosity liquid, which also has lower surface tension than the drop liquid. The presence of this low-viscosity film modifies the boundary conditions, giving effective slip to the drop, which forms a regular bowl-shaped crown, which rises vertically away from the solid and subsequently breaks up through the formation of a multitude of holes. Previous experiments [1] have proposed that the breakup of the crown results from a spray of fine droplets ejected from the thin film. These droplets can hit the inner side of the crown forming spots with lower surface tension, which drive the hole formation. We test the validity of this assumption by doing close-up imaging to identify individual spray droplets, to show how they hit the crown and influence the hole formation. For all the impact experiments, the release height was kept constant at $H=5.4$ m, leading to an impact velocity of $U=9.5$ m/s on the thin liquid film. [1] Thoroddsen, S. T., Etoh, T. G. & Takehara, K., Crown-breakup by Marangoni instability. *J. Fluid Mech.*, **557**, pp. 63-72 (2006).

10:53AM H16.00002 Binary drop interaction on surfaces: onset and bounding ligaments of Crescent-Moon fragmentation, LYDIA BOUROUIBA, YONGJI WANG, The Fluid Dynamics of Disease Transmission Laboratory, Massachusetts Institute of Technology — Drop impacts on surfaces can splash and create secondary droplets. These have important implications for industrial, environmental, and health processes such as air contamination by secondary pathogen-bearing droplets shaping disease transmission. Most studies of splash on surfaces have focused on the impact of one drop on a dry surface. Nevertheless, the outcome of impacts by spray or rain are shaped by the presence of adjacent sessile drops on the surface. Recently, in the context of rain and spray-induced disease transmission in crops, one particular binary drop interaction, the crescent-moon splash, was identified as a frequent and efficient source of secondary droplets (Gilet and Bourouiba ICB 2014 and JRSI 2015). The crescent-moon results from the interaction of an impacting drop with a sessile drop in the neighborhood of the impact point. Here, we report and rationalize the existence of a critical transition of impact parameters that enables the crescent-moon fragmentation to emerge. We also report and rationalize the peculiar, yet universal emergence of two bounding ligaments that are important in shaping the crescent-moon sheet.

11:06AM H16.00003 Numerical simulation of droplet splashing over varying thin liquid film.¹, AMARESH DALAL, JAI MANIK, GANESH NATARAJAN, Indian Inst of Tech-Guwahati — Droplet impact on wet surfaces is observed in various industrial processes and natural phenomenon. Behavior of droplet impact over thin liquid film is a complex phenomenon involving strong interface deformations. In the past, various studies have been performed to investigate the dynamic behavior of droplets using different geometries and physical conditions. But all the studies were primarily with constant film thickness. The present work is focused on the deformation of single and multiple droplets falling over thin liquid film with variable film thickness. The varying thicknesses of the film may be achieved by considering a sinusoidal varying bottom wall of two different amplitudes. It has been observed that the velocity with which the crown is spreading actually get decreased with the increase in the amplitude of the sinusoidally varying film. Similar behavior has been observed irrespective of the location of drop fall i.e. either falling over crest or over the trough. Also it has been noted that, in the case when droplet is falling over crest, the thickness of the lower portion of the crown rim also gets increased with the increase in amplitude of the film.

¹This Study is funded by a grant from BRNS, DAE, Government of India.

11:19AM H16.00004 Liquid dynamics and surface wettability in splashing, ANDRZEJ LATKA, ARNOOT BOELENS, JUAN DE PABLO, SIDNEY NAGEL, University of Chicago — The impact of a drop results in a beautiful splash that depends on the properties of the liquid, the substrate, and the surrounding air. The interactions of the three phases, particularly those of the ambient gas, have proven difficult to understand. Here, we focus on two aspects of splashing that make it exceptionally challenging: the surprising role of substrate wetting and the complicated hydrodynamics of drop impact. Splashing theories, by analogy to forced wetting, have predicted a strong dependence on wetting properties. By using high-speed interference imaging and simulations, we find that the dynamics of the air-liquid interface at the contact line are independent of substrate wetting properties. We also investigate the effect of the drops evolving shape. When the drop first contacts the surface, it initially exhibits a region of high negative curvature that later disappears after the drop has spread sufficiently. We find that the effect of air on splashing is significantly stronger in the initial regime and demonstrate that this difference leads to a high and low impact velocity regime.

11:32AM H16.00005 Unified thickness profile of radially expanding sheets in the air

, YONGJI WANG, LYDIA BOUROUIBA, The Fluid Dynamics of Disease Transmission Laboratory, Massachusetts Institute of Technology — The impact of a drop on a small solid surface or an edge results in a sheet expansion in the air. The sheet can then fragment into droplets. Understanding the dynamics of expansion and fragmentation in the air is important for a range of applications, including the transport of pathogen-bearing droplets created from contaminated leaves or surfaces. Here, we revisit the axisymmetric case of a radially expanding sheet formed from the impact of a drop on a small target of comparable size to that of the drop. We show that the temporal and spatial evolution of the sheet thickness profile is governed by a self-similar solution derived from first principles. The derived profile allows to collapse on a single curve the direct experimental measurements of sheet thickness profile for impacts on targets reported in the literature to date. A unified functional form governing the sheet thickness profile is proposed and reconciles the two conflicting theoretical profiles proposed in the literature thus-far. Finally, we show that the surface-to-drop size ratio plays an important role in affecting the thickness profile of the sheet in the air and rationalize the effects involved. Our findings allow to unify the thickness profile of unsteady expanding sheets in the air.

11:45AM H16.00006 How a laser impact fragments a liquid drop , HANNEKE GELDERBLOM,

ALEXANDER L. KLEIN, Physics of Fluids, Faculty of Science & Technology, University of Twente, The Netherlands, HENRI LHUISSIER, IUSTI, Aix-Marseille Université, France, DETLEF LOHSE, Physics of Fluids, Faculty of Science & Technology, University of Twente, The Netherlands, EMMANUEL VILLERMAUX, IRPHE, Aix-Marseille Université, France — The deposition of laser energy in a superficial layer of an unconfined liquid drop leads to propulsion, strong deformation of the drop into a thin sheet, and eventually fragmentation. Here we study the mechanisms leading to drop fragmentation by combining high-speed and stroboscopic imaging with analytical modelling. We investigate how ligaments and holes develop on the deforming drop, which eventually cause the drop to break up, and identify the dependence of this fragmentation process on the laser-pulse properties. We demonstrate that a Rayleigh-Taylor instability of the decelerating sheet rim leads to the formation of radial ligaments. The holes nucleating in the sheet result from a subtle interaction between the kinematic amplification of initial disturbances in the laser-beam profile and an intrinsic Rayleigh-Taylor instability caused by the rapid forward acceleration of the drop.

11:58AM H16.00007 Shock wave-droplet interaction , HAMED HABIBI KHOSHMEHR, ROUSLAN

KRECHETNIKOV, University of Alberta — Disintegration of a liquid droplet under the action of a shock wave is experimentally investigated. The shock wave-pulse is electromagnetically generated by discharging a high voltage capacitor into a flat spiral coil, above which an isolated circular metal membrane is placed in a close proximity. The Lorentz force arising due to the eddy current induced in the membrane abruptly accelerates it away from the spiral coil thus generating a shock wave. The liquid droplet placed at the center of the membrane, where the maximum deflection occurs, is disintegrated in the process of interaction with the shock wave. The effects of droplet viscosity and surface tension on the droplet destruction are studied with high-speed photography. Water-glycerol solution at different concentrations is used for investigating the effect of viscosity and various concentrations of water-sugar and water-ethanol solution are used for studying the effect of surface tension. Here we report on how the metamorphoses, which a liquid drop undergoes in the process of interaction with a shock wave, are affected by varied viscosity and surface tension.

12:11PM H16.00008 Its harder to splash on soft solids¹ , SAM HOWISON, Mathematical Institute,

Oxford University, CHRISTOPHER HOWLAND, Trinity College, Oxford University, ARNAUD ANTKOWIAK, Institut Jean Le Rond d'Alembert, Sorbonne University, RAFAEL CASTREJON-PITA, School of Engineering and Materials Science, Queen Mary, University of London, JAMES OLIVER, ROBERT STYLE, Mathematical Institute, Oxford University, ALFONSO CASTREJON-PITA, Department of Engineering Science, University of Oxford — Droplets splash when they impact dry, flat substrates above a critical velocity that depends on parameters such as droplet size, viscosity and air pressure. By imaging ethanol drops impacting silicone gels of different stiffnesses we show that substrate stiffness also affects the splashing threshold. Splashing is reduced or even eliminated: droplets on the softest substrates need over 70 more kinetic energy to splash than they do on rigid substrates. We show that this is due to energy losses caused by deformations of soft substrates during the first few microseconds of impact. We find that solids with Young's moduli $\geq 100\text{ kPa}$ reduce splashing, in agreement with simple scaling arguments. Thus materials like soft gels and elastomers can be used as simple coatings for effective splash prevention. Soft substrates also serve as a useful system for testing splash-formation theories and sheet-ejection mechanisms, as they allow the characteristics of ejection sheets to be controlled independently of the bulk impact dynamics of droplets.

¹EPSRC (CJH), John Fell Oxford University Press (OUP) Research Fund (AACP and RWS), The Royal Society (AAC-P)

12:24PM H16.00009 How a drop splashes upon impact onto a moving surface. , HAMED

ALMOHAMMADI, ALIDAD AMIRFAZLI, Department of Mechanical Engineering, York University, Toronto, ON, Canada — Understanding whether drop spreads or splashes upon impact onto a moving surface is important due to its applications in printing, spraying, and icing. A systematic study was performed to understand how drop splashes upon impact onto moving hydrophilic and hydrophobic surfaces. High speed imaging from top and side views was used to capture the impact of drops ($D_0 = 2.5\text{ mm}$) of liquids with three different viscosities ($\mu = 1\text{-}4.1\text{ mPa.s}$). Wide range of normal drop velocity ($V_n = 0.5\text{-}3.4\text{ m/s}$) and surface velocity ($V_s = 0\text{-}17\text{ m/s}$) were studied; such normal and tangential velocity ranges are not available in systems where a drop impacts at an angle relative to a surface. It was found that the splashing behavior of the drop upon impact onto a moving surface, unlike the understanding in the literature, is azimuthally different along the lamella contact line. Splashing probability decreases along the lamella contact line as velocity difference between the surface and the lamella decreases. A new model was developed to describe such azimuthally different behavior for splashing which is function of normal Reynolds and Weber numbers, V_s , and surface wettability. It is also found that the increase of the viscosity decreases the splashing threshold.

12:37PM H16.00010 An octahedron model for oscillating, bouncing drops , FRANCOIS

BLANCHETTE, Applied Math., UC Merced — We present a model for oscillating and bouncing liquid drops. The model uses 6 point masses distributed as the vertices of an octahedron, connected by linear springs. We derive the physically relevant choice of parameters and use this model to study drops bouncing on solid surfaces, as well as drops bouncing on a nearly inviscid liquid surfaces. The surfaces may be stationary or subject to forced oscillations.

Monday, November 21, 2016 10:40AM - 12:50PM –

Session H17 Reacting Flows: LES D131 - J.M. McDonough, University of Kentucky

10:40AM H17.00001 Wildfire simulation using LES with synthetic-velocity SGS models, J. M. McDONOUGH¹, TINGTING TANG², University of Kentucky — Wildland fires are becoming more prevalent and intense worldwide as climate change leads to warmer, drier conditions; and large-eddy simulation (LES) is receiving increasing attention for fire spread predictions as computing power continues to improve (see, e.g., Coen et al., *J. Appl. Meteor. Climatol.*, 2013; McGrattan, *NIST*, 2008). We report results from wildfire simulations over general terrain employing implicit LES for solution of the incompressible Navier–Stokes (N.–S.) and thermal energy equations with Boussinesq approximation, altered with Darcy, Forchheimer and Brinkman extensions, to represent forested regions as porous media with varying (in both space and time) porosity and permeability. We focus on subgrid-scale (SGS) behaviors computed with a synthetic-velocity model, a discrete dynamical system, based on the poor man's N.–S. equations (Tang et al., *Int. J. Bifur. Chaos*, 2016) and investigate the ability of this model to produce fire whirls (tornadoes of fire) at the (unresolved) SGS level.

¹Professor, Mechanical Engineering and Mathematics

²Graduate Student, Department of Mechanical Engineering

10:53AM H17.00002 LES Modeling of Supersonic Combustion at SCRAMJET Conditions¹, ZACHARY VANE, GUILHEM LACAZE, JOSEPH OEFELIN, Sandia Natl Labs — Results from a series of large-eddy simulations (LES) of the Hypersonic International Flight Research Experiment (HIFIRE) are examined with emphasis placed on the coupled performance of the wall and combustion models. The test case of interest corresponds to the geometry and conditions found in the ground based experiments performed in the HIFIRE Direct Connect Rig (HDCR) in dual-mode operation. In these calculations, the turbulence and mixing characteristics of the high Reynolds number turbulent boundary layer with multi-species fuel injection are analyzed using a simplified chemical model and combustion closure to predict the heat release measured experimentally. These simulations are then used to identify different flame regimes in the combustor section. Concurrently, the performance of an equilibrium wall-model is evaluated in the vicinity of the fuel injectors and in the flame-holding cavity where regions of boundary layer and thermochemical non-equilibrium are present.

¹Support for this research was provided by the Defense Advanced Research Projects Agency (DARPA)

11:06AM H17.00003 Hybrid Eulerian-Lagrangian Vortex Model for Turbulent Reacting Flows, JOHN ROYERO, KAREEM AHMED, University of Central Florida — A hybrid Eulerian-Lagrangian model for three dimensional large eddy simulations of turbulent reacting flows is presented. The method utilizes a Eulerian grid to resolve large scale flow features and the Lagrangian vortex element method to capture smaller subgrid scale effects and carry out reactions which are then communicated back to the Eulerian grid after a set number of Lagrangian time steps. Lagrangian influences are localized in order to reduce computational cost. The Lagrangian vortex method which utilizes the Helmholtz decomposition of the velocity into potential, expansive, and solenoidal components allows the separation of the various mechanisms contributing to vorticity including gas expansion, diffusion, external body forces and baroclinic torque and is coupled with the Eulerian solver allowing easier implementation in arbitrary reacting flows at a reduced computational cost compared to a pure Lagrangian solver.

11:19AM H17.00004 A Flamelet Modeling Approach for Multi-Modal Combustion with Inhomogeneous Inlets¹, BRUCE A. PERRY, Princeton Univ, MICHAEL E. MUELLER, Princeton University — Large eddy simulations (LES) of turbulent combustion often employ models that make assumptions about the underlying flame structure. For example, flamelet models based on both premixed and nonpremixed flame structures have been implemented successfully in a variety of contexts. While previous flamelet models have been developed to account for multi-modal combustion or complex inlet conditions, none have been developed that can account for both effects simultaneously. Here, a new approach is presented that extends a nonpremixed, two-mixture fraction approach for compositionally inhomogeneous inlet conditions to partially premixed combustion. The approach uses the second mixture fraction to indicate the locally dominant combustion mode based on flammability considerations and switch between premixed and nonpremixed combustion models as appropriate. To assess this approach, LES predictions for this and other flamelet-based models are compared to data from a turbulent piloted jet burner with compositionally inhomogeneous inlets, which has been shown experimentally to exhibit multi-modal combustion.

¹This work was supported by the NSF Graduate Research Fellowship Program under grant DGE 1148900.

11:32AM H17.00005 Three-dimensional CLEM-LES of irregular detonation propagation¹, BRIAN MAXWELL, MATEI RADULESCU, University of Ottawa — Recently, thin-channel experiments and 2D simulations have been conducted in order to investigate the effect of turbulent mixing rates on the structure of irregular detonation wave propagation. Furthermore, the dependence of the observed cell pattern, and also the reaction zone thickness, on the mixing of burned products with pockets of unburned gases, was investigated. The current work now includes 3D simulations, which are conducted to provide further validation of, and insight into, the 2D results. All simulations have been conducted using the Compressible Linear Eddy Model for Large Eddy Simulation (CLEM-LES). To date, the 3D results are found to match closely the previous 2D results. The agreement is partly due to sufficient resolution of the large scale fluid motions, which are observed experimentally to be predominant in only two directions. Furthermore, the CLEM-LES methodology incorporates 3D mixing effects at the subgrid level. Finally, it was found that turbulent fluctuations on the subgrid were found to give rise to statistically lower than average propagation velocities on the wave front. This lead to longer ignition delays for large amounts of gas passing through the wave, giving rise to the unburned pockets of gas observed experimentally.

¹Authors would like to acknowledge NSERC and Shell for project funding

11:45AM H17.00006 Comparison of Turbulence–Chemistry Interaction Models in the Large Eddy Simulation of High-Speed Combustion., WENHAI LI, KEN ALABI, TTC Technologies, Inc. Centereach, NY, FOLUSO LADEINDE, ZHIPENG LOU, State Univ of NY- Stony Brook — In this study, three turbulence-chemistry interaction models: the flamelet, eddy-breakup (EBU), and laminar chemistry models, are compared in the large-eddy simulation (LES) of high speed combustion. It is the case that the simple models still find extensive applications, with fairly acceptable results in many instances. The standard flamelet model developed for low Mach number flows has been modified to account for compressibility effects in supersonic combustion. The comparison exercise has been based on the bluff-body flames that occur under high-speed conditions.

11:58AM H17.00007 Evaluation of subgrid dispersion models for LES of spray flames

, QING WANG, Stanford University, XINYU ZHAO, University of Connecticut, LUCAS ESCLAPEZ, PAVAN GOVINDARAJU, MATTHIAS IHME, Stanford University — Turbulent dispersion models for particle-laden turbulent flows have been studied extensively over the past few decades, and different modeling approaches have been proposed and tested. However, the significance of the subgrid dispersion model and its influence on the flame dynamics for spray combustion have not been examined. To evaluate the performance of dispersion models for spray combustion, direct numerical simulations (DNS) of three-dimensional counterflow spray flames are studied. The DNS configuration features a series of different droplet sizes to study effects of different Stokes numbers. An *a priori* comparison of the statistics generated from three subgrid dispersion models is made, for both non-reacting and reacting conditions. Improved agreement with DNS is shown for the stochastic model and the regularized deconvolution model than a closure-free model. The effect of filter sizes in relation to droplet sizes are investigated for all models. Subsequently, a *posteriori* modeling of the same configuration with different resolutions is performed to compare these models in the presence of other subgrid models. Finally, models for the subgrid closure of scalar transport for multiphase droplet combustion are proposed and evaluated.

12:11PM H17.00008 Performance assessment of a pre-partitioned adaptive chemistry approach in large-eddy simulation of turbulent flames¹

, PERRINE PEPIOT, YOUWEN LIANG, ASHISH NEWALE, STEPHEN POPE, Cornell Univ — A pre-partitioned adaptive chemistry (PPAC) approach recently developed and validated in the simplified framework of a partially-stirred reactor is applied to the simulation of turbulent flames using a LES/particle PDF framework. The PPAC approach was shown to simultaneously provide significant savings in CPU and memory requirements, two major limiting factors in LES/particle PDF. The savings are achieved by providing each particle in the PDF method with a specialized reduced representation and kinetic model adjusted to its changing composition. Both representation and model are identified efficiently from a pre-determined list using a low-dimensional binary-tree search algorithm, thereby keeping the run-time overhead associated with the adaptive strategy to a minimum. The Sandia D flame is used as benchmark to quantify the performance of the PPAC algorithm in a turbulent combustion setting. In particular, the CPU and memory benefits, the distribution of the various representations throughout the computational domain, and the relationship between the user-defined error tolerances used to derive the reduced representations and models and the actual errors observed in LES/PDF are characterized.

¹This material is based upon work supported by the U.S. Department of Energy Office of Science, Office of Basic Energy Sciences under Award Number DE-FG02-90ER14128.

12:24PM H17.00009 LES/PDF studies of joint statistics of mixture fraction and progress variable in piloted methane jet flames with inhomogeneous inlet flows

, PEI ZHANG, Purdue University, ROBERT BARLOW, Sandia National Laboratories, ASSAAD MASRI, The University of Sydney, HAIFENG WANG, Purdue University — The mixture fraction and progress variable are often used as independent variables for describing turbulent premixed and non-premixed flames. There is a growing interest in using these two variables for describing partially premixed flames. The joint statistical distribution of the mixture fraction and progress variable is of great interest in developing models for partially premixed flames. In this work, we conduct predictive studies of the joint statistics of mixture fraction and progress variable in a series of piloted methane jet flames with inhomogeneous inlet flows. The employed models combine large eddy simulations with the Monte Carlo probability density function (PDF) method. The joint PDFs and marginal PDFs are examined in detail by comparing the model predictions and the measurements. Different presumed shapes of the joint PDFs are also evaluated.

12:37PM H17.00010 Scalar mixing in LES/PDF of a high-Ka premixed turbulent jet flame¹

, JIAPING YOUNG YANG, College of Engineering, Peking University — We report a large-eddy simulation (LES)/probability density function (PDF) study of a high-Ka premixed turbulent flame in the Lund University Piloted Jet (LUPJ) flame series, which has been investigated using direct numerical simulation (DNS) and experiments. The target flame, featuring broadened preheat and reaction zones, is categorized into the broken reaction zone regime. In the present study, three widely used mixing modes, namely the Interaction by Exchange with the Mean (IEM), Modified Curl (MC), and Euclidean Minimum Spanning Tree (EMST) models are applied to assess their performance through detailed *a posteriori* comparisons with DNS. A dynamic model for the time scale of scalar mixing is formulated to describe the turbulent mixing of scalars at small scales. Better quantitative agreement for the mean temperature and mean mass fractions of major and minor species are obtained with the MC and EMST models than with the IEM model. The multi-scalar mixing in composition space with the three models are analyzed to assess the modeling of the conditional molecular diffusion term. In addition, we demonstrate that the product of OH and CH₂O concentrations can be a good surrogate of the local heat release rate in this flame.

¹This work is supported by the National Natural Science Foundation of China (Grant Nos. 11521091 and 91541204)

Monday, November 21, 2016 10:40AM - 12:50PM –

Session H18 Flow Instability: Vortex and Wakes D135 - Adam Edstrand, Florida State University

10:40AM H18.00001 DNS of two-phase flow in an inclined pipe

, FANGFANG XIE, Zhejiang University, XIAONING ZHENG, Brown university, MICHAEL TRIANTAFYLLOU, Massachusetts Institute Technology, YIANNIS CONSTANTINIDES, Chevron Energy Technology Company, GEORGE KARNIADAKIS, Brown university — We study the de-stabilization mechanisms of two-phase flow in an inclined pipe subject to gravity with a phase-field approach. At the inlet, a stratified flow is imposed with a parabolic velocity profile. We found that due to gravity, the stratified flow will become unstable, causing a complex transitional flow inside the pipe. Firstly, a 2D channel geometry is considered. When the heavy fluid is injected in the top layer, inverted vortex shedding emerges, interacting periodically with the bottom wall as it develops further downstream. The accumulation of heavy fluid in the bottom wall causes a backflow, which interacts with the previous jet. On the other hand, when the heavy fluid is placed in the bottom layer, a big slug is formed and subsequently breaks into small pieces, some of which will be shed along the pipe. To describe the generation of vorticity from the two-phase interface and pipe walls, we analyze the circulation dynamics and connect it to the two-phase flow pattern. Moreover, we analyze the two-phase flow induced forces along the pipe, which is capable of producing unwanted and destructive vibrations. Finally, we conduct 3D simulations in the circular pipe and compared the differences of flow dynamics against the 2D simulation results.

10:53AM H18.00002 A parabolized stability analysis of a trailing vortex wake¹ , ADAM EDSTRAND, Florida State University, PETER SCHMID, Imperial College London, KUNIHIKO TAIRA, LOUIS CATTAFESTA, Florida State University

To aid in understanding how best to control a trailing vortex, we perform a parabolized stability analysis on a flow past a wing at a chord-based Reynolds number of 1000. At the upstream position, the wake instability branch dominates, with only a single vortex instability present in the spectrum. With downstream progression, the growth rate of the wake instability decays, but remains unstable 10 chords downstream. With the wake mode being unstable so far downstream, these results imply that the excitation of the wake instability, despite the varying base flow, will continue to see growth and potentially disrupt the trailing vortex. Conversely, the vortex instability in its formative region rapidly decays to the stable half-plane, then at 11 chords downstream becomes unstable again. We hypothesized the renewed instability growth far downstream is developing as a result of vortex instabilities, however the excitation of these instabilities proves to be challenging in the vortex far field. From these results, control near the two-dimensional wake behind the airfoil may better interfere with the trailing vortex formation; however, to determine the optimal disturbances, an adjoint analysis is required and is included in the future work of the project.

¹ONR Grants N00014-10-1-0832 and N00014-15-1-2403

11:06AM H18.00003 Flow Control Behind Bluff Bodies through the Interaction of an Attached Resonant Flexible Tail¹ , SAMUEL SHELLEY, University of Exeter, JOHN SMITH, DSTL, ALASTAIR HIBBINS, ROY SAMBLES, SIMON HORSLEY, University of Exeter

— Steady uniform flow, incident upon a bluff body can separate downstream causing a wake to form, this leads to the periodic shedding of vortices behind the body. By adding a thin flexible tail to the rear of the body one may reduce the drag as well as change the vortex shedding frequency (VSF). In this work we model the flow past a cylinder, in the Laminar flow regime, with an attached tail, varying the length and stiffness of the tail to couple the resonant frequencies of the tail to the natural VSF of the structure. We use this to explore how the drag and VSF of the system change as we couple to different vibrational modes of the tail. On increasing tail length, or decreasing tail stiffness progressively on passing where the natural VSF of the cylinder and tail resonances couple we see sharp increases in both the drag and VSF, which both gradually decrease again. The effect of changing the shape of the end of the tail is also investigated by exploring tails with square, rounded and triangular trailing edges. Experiments are being conducted in water at a higher Reynolds number using a tail made out of Neoprene to confirm these modelling results.

¹DSTL

11:19AM H18.00004 Sensitivity analysis of small circular cylinders as wake control , JULIO MENEGHINI, GUSTAVO PATINO, RAFAEL GIORIA, University of Sao Paulo

— We apply a sensitivity analysis to a steady external force regarding control vortex shedding from a circular cylinder using active and passive small control cylinders. We evaluate the changes on the flow produced by the device on the flow near the primary instability, transition to wake. We numerically predict by means of sensitivity analysis the effective regions to place the control devices. The quantitative effect of the hydrodynamic forces produced by the control devices is also obtained by a sensitivity analysis supporting the prediction of minimum rotation rate. These results are extrapolated for higher Reynolds. Also, the analysis provided the positions of combined passive control cylinders that suppress the wake. The latter shows that these particular positions for the devices are adequate to suppress the wake unsteadiness. In both cases the results agree very well with experimental cases of control devices previously published.

11:32AM H18.00005 The correlation between 2D-3D wake transition and propulsive efficiency of a flapping foil¹ , LIPING SUN, JIAN DENG, XUEMING SHAO, Department of Mechanics, Zhejiang University, Hangzhou 310027, People's Republic of China

— We study numerically the propulsive wakes produced by a flapping foil. As a major contribution of this report, we find an interesting coincidence that the efficiency maximum agrees well with the 2D-3D transition boundary. Although lack of direct 3D simulations, it is reasonable to conjecture that the propulsive efficiency increases with Strouhal number until the wake transits from a 2D state to a 3D state. By comparing between the pure pitching motion and the pure heaving motion, we find that the 2D-3D transition occurs earlier for the pure heaving foil than that of the pure pitching foil. Consequently, the efficiency for the pure heaving foil peaks more closely to the wake deflection boundary than that of the pure pitching foil. Furthermore, since we have drawn the maps on the same parametric space with the same Reynolds number, it is possible to make a direct comparison in the propulsive efficiency between a pure pitching foil and a pure heaving foil. We note that the maximum efficiency for a pure pitching foil is 15.6%, and that of a pure heaving foil is 17%, indicating that the pure heaving foil has a slightly better propulsive performance than that of the pure pitching foil for the currently studied Reynolds number of $Re = 1700$.

¹This research is supported by the National Natural Science Foundation of China (Grant No: 11272283) and the Public Projects of Zhejiang Province (Grant No: 2015C31126) to conduct this research.

11:45AM H18.00006 Mechanism of Secondary Instability of Flow around a Circular Cylinder , HUA-SHU DOU, Zhejiang Sci-Tech University, AN-QING BEN, Longshine Technology Com. Ltd., FLUID MECHANICS RESEARCH TEAM TEAM

— Flow around a circular cylinder in infinite domain is simulated with large eddy simulation at $Re=200$, and the mechanism of the origin of secondary vortex street is analyzed. The simulation results show that the vortex street generated in the cylinder near wake disappears as the flow moving downstream. Secondary instability occurs in far wake of the cylinder after the primary vortex street dying away. The processes of first instability and secondary instability in the cylinder wake are recorded in the simulation. The instability of the entire flow field is studied with the energy gradient theory. It is found that it is the high value of the energy gradient function generated by the zero velocity gradients that leads to the instability. As the vortex developing at rear of the cylinder, the value of the energy gradient function becomes low downstream, which leads to the vortex dying away. At further downstream, the value of the energy gradient function is enlarged again because of the role of perturbation, which leads to the secondary instability. It can be concluded that the interaction of the variation of the value of the energy gradient function and the perturbation leads to the occurrence of secondary instability.

11:58AM H18.00007 Unstable shear flows in two dimensional strongly correlated liquids – a hydrodynamic and molecular dynamics study , AKANKSHA GUPTA, RAJARAMAN GANESH, Institute for Plasma Research, India, ASHWIN JOY, Indian Institute of Technology Madras, India

— In Navier-Stokes fluids, shear flows are known to become unstable leading to instability and eventually to turbulence. A class of flow namely, Kolmogorov Flows (K-Flows) exhibit such transition at low Reynolds number. Using fluid and molecular dynamics, we address the physics of transition from laminar to turbulent regime in strongly correlated-liquids such as in multi-species plasmas and also in naturally occurring plasmas with K-Flows as initial condition. A 2D phenomenological generalized hydrodynamic model is invoked wherein the effect of strong correlations is incorporated via a viscoelastic memory. To study the stability of K-Flows or in general any shear flow, a generalized eigenvalue solver has been developed along with a spectral solver for the full nonlinear set of fluid equations. A study of the linear and nonlinear features of K-Flow in incompressible and compressible limit exhibits cyclicity and nonlinear pattern formation in vorticity. A first principles based molecular dynamics simulation of particles interacting via Yukawa potential is performed with features such as configurational and kinetic thermostats for K-Flows. This work reveals several interesting similarities and differences between hydrodynamics and molecular dynamics studies.

12:11PM H18.00008 Spiral vortex formation in cross-slot flow, SIMON HAWARD, NOA BURSHTEIN, Okinawa Inst of Sci & Tech, ROBERT POOLE, University of Liverpool, PAULO OLIVEIRA, Universidade de Beira Interior, MANUEL ALVES, Universidade do Porto, AMY SHEN, Okinawa Inst of Sci & Tech — Fluid flow through bisecting channels (cross-slots) results in the formation of a steady spiral vortex as the Reynolds number (Re) is increased above a modest critical value (Re_c). The value of Re_c is strongly dependent on the channel aspect ratio, $\alpha = d/w$, where d and w are the depth and width of the channel, respectively. Quasistatic experiments and numerical simulations over a range of Re show that for low α this symmetry-breaking bifurcation is supercritical, however subcritical behavior develops as α is increased. The system can be described by a Landau-type 6th-order polynomial potential and we identify a value of $\alpha \approx 0.55$ for which a tricritical point can be found. Dynamic experiments and simulations conducted across the transition indicate a plausible mechanism for the onset of the instability. Our analysis suggests that the transition results from the growth of center-point vorticity induced by random imbalances between two pairs of Dean vortices that form in the channel cross-section. Vorticity growth is governed by two distinct time scales. At short times, viscous diffusion dominates and vorticity grows slowly. Once the vorticity is sufficiently high, vortex stretching dominates and the vorticity grows rapidly until steady state is reached.

12:24PM H18.00009 Instability of a vortex sheet leaving a right-angled wedge, STEFAN LLEWELLYN SMITH, ANTHONY DAVIS, Univ of California - San Diego — We examine the dynamics of a semi-infinite vortex sheet attached not to a semi-infinite plate but instead to a rigid right-angled wedge, with the sheet aligned along one of its edges. The resulting linearised unsteady potential flow is forced by an oscillatory dipole in the uniform stream passing along the top of the wedge, while there is stagnant fluid in the remaining quadrant. The essentially quadrant-type geometry replaces the usual Wiener-Hopf technique by the Mellin transform. The core difficulty is that a first-order difference equation of period 4 requires a solution of period unity. As a result the complex fourth roots ($\pm 1 \pm i$) of -4 appear in the complementary function. The Helmholtz instability wave is excited and requires careful handling to obtain explicit results for the amplitude of the instability wave.

12:37PM H18.00010 Unstable flow of worm-like micelles in rectangular microfluidic channels, PAUL SALIPANTE, STEVEN HUDSON, NIST - Natl Inst of Stds & Tech — We investigate a jetting instability of shear banding worm-like micelle (WLM) solutions in microfluidic channels with rectangular cross-sections. The flow is tracked using both 3-D and 2-D particle tracking methods in channels of different aspect ratio, size, and wall materials. We observe that the instability forms in high aspect ratio channels within an intermediate range of volumetric flows. The location of the high velocity jet in the channel appears to be sensitive to stress localizations induced by channel defects and wall roughness. A lower concentration WLM solution, with a monotonic stress curve, does not show the banding instability but displays non-negligible velocity gradients across the channel width. The transient development of the instability at the entrance of the microfluidic channel is observed in various geometries. The experimental measurements are compared to finite volume simulations using the Johnson-Segelmann viscoelastic model. The simulations show a qualitatively similar behavior to our experimental observations and indicate that normal stresses in the cross stream directions lead to the development of the jetting flow.

Monday, November 21, 2016 10:40AM - 12:50PM –
Session H19 Bio: Flapping and Swimming II D136 - Leif Ristroph, The Courant Institute, New York University

10:40AM H19.00001 Dynamic Schooling of a Tandem Pair of Heaving Hydrofoils, JOEL NEWBOLT, LEIF RISTRÖPH, JUN ZHANG, New York University — The reverse von Kármán wake generated by a heaving hydrofoil has recently been shown to provide stable positions to a second hydrofoil heaving in the wake (Ramananarivo et al. at NYU). Because a similar wake structure is seen for many swimming and flying animals this fluid-mediated interaction is suspected to play a role in schooling and flocking. A newly designed experimental apparatus allows us to study this interaction in the case where the two foils are powered independently so that each foil may take on a different flapping amplitude, phase and frequency. Measurements show that the stable positions of the following foil can be shifted to any arbitrary downstream position by varying only the relative flapping phase between the foils. At different relative frequencies and amplitudes the following foil exhibits several distinct trajectories. When the following foil has a lower frequency and higher amplitude than the leader the spacing between the foils can undergo a periodic trajectory. When driven at a common frequency the follower occupies stable positions in the wake of the leader. When the follower has a higher frequency the spacing between the foils is unstable, either increasing or decreasing in time, depending on the relative amplitudes and initial conditions.

10:53AM H19.00002 Schooling behavior of heaving flexible airfoils¹, SUNGHYUK IM, HYUNG JIN SUNG, KAIST — The schooling behavior of rigid and flexible NACA0017 airfoils in the heaving motion is experimentally explored in a merry-go-round equipment. The airfoil was attached to the end of a horizontal support bar whose other end was connected to the freely rotating vertical axis. The axis was forced to undergo a sinusoidal motion in the vertical direction to make a pure heaving motion of the airfoils in the frequency range of 0.5 to 5 Hz. The propulsion due to the heaving airfoils is expressed by a horizontally rotating speed of the support bar. This experimental setup is simulating infinite schooling situations of airfoils in an in-phase heaving motion with the streamwise distance d . The ratio of the distance to the chord length d/c was determined by the number of airfoils ($1 \leq n \leq 8$). The rotational frequency F according to the heaving frequency f was measured with different experimental parameters. The schooling number $S = f/(nF)$, representing the number of heaving oscillations between each airfoil, was introduced to explain the schooling behavior of the airfoils. The effects of the flexibility, d/c and f on the propulsive performance were examined with the schooling behavior of the airfoils.

¹This work was supported by the Creative Research Initiatives (No. 2016-004749) program of the National Research Foundation of Korea (MSIP).

11:06AM H19.00003 Leveraging fluid-structure interaction for passive control of flapping locomotion, CHAN-YE OH, YANGYANG HUANG, ZITENG WEN, EVA KANSO, MITUL LUHAR, Univ of Southern California — While many living organisms employ active feedback control during flapping locomotion, there is increasing evidence to suggest that passive fluid-structure interactions also play a central role in dictating stability and efficiency (Liu et al. *Phys. Rev. Lett.* 108:06081-3, 2012). The current project seeks to experimentally evaluate and numerically verify the rotational stability and dynamics of a rigid Δ -flyer oscillating up-and-down in a rest fluid. We explore the dynamic behavior of the flyer in terms of three dimensionless parameters: opening angle, oscillation amplitude, and acceleration of the flyer. Within the parameter ranges tested, we identify four types of behavior: periodic rotation, chaotic dynamics, stable behavior (concave down position), and bistability (concave up and down position). The emergence of periodic and chaotic rotation depends primarily on the oscillation amplitude of the flyer, whereas transition from stability to bistability is dependent on both the amplitude and acceleration. The transition to bistability occurs at a constant ratio of drag to gravity, indicating that the stabilizing effect is hydrodynamic.

11:19AM H19.00004 Cyber-physical experiments on the efficiency of swimming protocols¹, NATHANIEL WEI, DANIEL FLORYAN, TYLER VAN BUREN, Princeton University, ALEXANDER SMITS, Princeton University and Monash University — We present results from experiments on a biologically inspired cyber-physical system, composed of a two-dimensional heaving and pitching rigid airfoil attached to a six component load cell, mounted to a traverse that can move along a water channel. A feedback controller, influenced by the apparatus of Mackowski and Williamson (J. Fluid Struct., 2011), introduces the effects of a fictional drag force specified by a virtual body profile and drives the traverse accordingly. Free-swimming protocols using the force-feedback system are compared with similar motions on a motionless traverse. The propulsive efficiency of burst-and-coast kinematics is also considered. Of particular interest are (1) the implementation of the cyber-physical control system with respect to the accessible experimental parameter space, (2) the impact of force-based streamwise actuation on experimental data, and (3) the effects of burst-and-coast motions on propulsive efficiency.

¹The work was supported by the Office of Naval Research (ONR) under MURI Grant N00014-14-1-0533.

11:32AM H19.00005 Airflow Actuation of Shortfin Mako Shark Denticles¹, SEAN DEVEY, PAUL HUBNER, AMY LANG, The University of Alabama — The shortfin mako shark is covered in microscopic scales called denticles, which may act as a mechanism for passive flow control. Recent research has investigated the theory that reversing flow could passively bristle these denticles, which could delay flow separation. Water tunnel studies have supported this theory, yet a wind tunnel study at a greater dynamic pressure found no significant differences between an airfoil covered with mako skin and a smooth airfoil. A likely cause is that surface tension between denticles, which must be wet to retain flexibility, prevented bristling. This would not be an issue in water. To determine what reverse airflow characteristics cause denticle bristling in air, a benchtop study was conducted in which a jet of air was impinged upon a sample of wet mako skin in the reverse flow direction. A microscope and camera captured video of the denticles under the air jet, and image analysis techniques were used to detect bristling. Analysis shows sporadic bristling around 16 m/s ($q = 150 \text{ Pa}$) but full bristling does not occur until above 35 m/s ($q = 740 \text{ Pa}$). The free stream velocities required to achieve such reversal speeds are much higher. For this reason, mechanical analogues will be used rather than real skin in future studies of this mechanism.

¹Funding from Boeing and NSF REU site grant EEC 1358991 is greatly appreciated

11:45AM H19.00006 Bistable flapping of flexible flyers in oscillatory flow, YANGYANG HUANG, EVA KANSO, University of Southern California — Biological and bio-inspired flyers move by shape actuation. The direct control of shape variables for locomotory purposes is well studied. Less is known about indirect shape actuation via the fluid medium. Here, we consider a flexible Δ -flyer in oscillatory flow that is free to flap and rotate around its fixed apex. We study its motion in the context of the inviscid vortex sheet model. We first analyze symmetric flapping about the vertical axis of gravity. We find that there is a finite value of the flexibility that maximizes both the flapping amplitude and elastic energy storage. Our results show that rather than resonance, the flyer relies on fluidic effects to optimize these two quantities. We then perturb the flyer away from the vertical and analyze its stability. Four distinct types of rolling behavior are identified: mono-stable, bistable, bistable oscillatory rotations and chaotic dynamics. We categorize these types of behavior in terms of the flyers and flow parameters. In particular, the transition from mono-stable to bistable behavior occurs at a constant value of the product of the flow amplitude and acceleration. This product can be interpreted as the ratio of fluidic drag to gravity, confirming the fluid role in this transition.

11:58AM H19.00007 Intermittent Swimming with a Flexible Propulsor¹, EMRE AKOZ, SAMANE ZEYGHAMI, KEITH MOORED, Lehigh University — Some animals propel themselves by using an intermittent swimming gait known as burst-and-glide or a burst-and-coast motion. These swimmers tend to have a more pronounced pitching of their caudal fins than heaving leading to low non-dimensional heave-to-pitch ratios. Recent work has shown that when this ratio is sufficiently low the efficiency of an intermittently heaving/pitching airfoil can be significantly improved over a continuously oscillating airfoil. However, fish that swim with an intermittent gait, such as cod and saithe, do not have rigid fins, but instead have highly flexible fins. To examine the performance and flow structures of an intermittent swimmer with a flexible propulsor, a fast boundary element method solver strongly coupled with a torsional-spring structural model was developed. A self-propelled virtual body combined with a flexible-hinged pitching airfoil is used to model a free-swimming animal and its flexible caudal fin. The duty cycle of the active to the coasting phase of motion, the torsional spring flexibility and the forcing frequency are all varied. The cost-of-transport and the swimming speed are measured and connected to the observed wake patterns.

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

12:11PM H19.00008 Unsteady propulsion in ground effects¹, SUNG GOON PARK, BOYOUNG KIM, HYUNG JIN SUNG, Department of Mechanical Engineering, KAIST — Many animals in nature experience hydrodynamic benefits by swimming or flying near the ground, and this phenomenon is commonly called ‘ground effect’. A flexible fin flapping near the ground was modelled, inspired by animals swimming. A transverse heaving motion was prescribed at the leading edge, and the posterior parts of the fin were passively fluttering by the fin-fluid interaction. The fin moved freely horizontally in a quiescent flow, by which the swimming speed was dynamically determined. The fin-fluid interaction was considered by using the penalty immersed boundary method. The kinematics of the flexible fin was altered by flapping near the ground, and the vortex structures generated in the wake were deflected upward, which was qualitatively analyzed by using the vortex dipole model. The swimming speed and the thrust force of the fin increased by the ground effects. The hydrodynamic changes from flapping near the ground affected the required power input in two opposite ways; the increased and decreased hydrodynamic pressures beneath the fin hindered the flapping motion, increasing the power input, while the transversely reduced flapping motion induced the decreased power input. The Froude propulsive efficiency was increased by swimming in the ground effects

¹Creative Research Initiatives (No. 2016-004749) program of the National Research Foundation of Korea (MSIP).

12:24PM H19.00009 Internally-actuated flexible fins swim faster and more efficiently with a passive attachment, PETER YEH, ALEXANDER ALEXEEV, Woodruff School of Mechanical Engineering, Georgia Institute of Technology — Using three dimensional computer simulations, we probe biomimetic free swimming of an internally-actuated flexible plate in the regime near the first natural frequency. The plate is driven by an oscillating internal moment approximating the actuation mechanism of a piezoelectric MFC bimorph. We show in our simulations that the addition of a passive attachment increases both swimming velocity and efficiency. Specifically, if the active and passive sections are of similar size, the overall performance is the best. We determine that this optimum is a result of two competing factors. If the passive section is too large, then the actuated portion is unable to generate substantial deflection to create sufficient thrust. On the other hand, a large actuated section leads to a bending pattern that is inefficient at generating thrust especially at higher frequencies.

12:37PM H19.00010 Enhancing propulsive efficiency through proper design of bending patterns of a flexible pitching foil¹ , SAMANE ZEYGHAMI, EMRE AKOZ, KEITH MOORED, Lehigh University

— Many aquatic animals propel themselves efficiently through water by oscillating flexible fins. These fins are, however, not homogeneously flexible, but instead their flexural rigidity varies along their chord and span. To detail the flow structures and propulsive performance of these functionally-graded propulsors a simple model of an unsteady pitching airfoil with a flexible hinge of varying location is examined. This acts as a first-order model of a functionally-graded fin by varying both the flexibility and bending pattern of the propulsor. Recent experiments have shown that adding a flexible 'tail' with the proper stiffness to a rigid pitching foil can effectively delay/suppress the formation of a deflected wake thereby enhancing the cycle-averaged wake momentum in the swimming direction. To extend these observations, we investigate the dependency of the wake pattern of a hinged pitching airfoil to the location and flexibility of the hinge by employing a fast boundary element method solver that is strongly coupled with a torsional spring structural model. The observed wake patterns are further connected to the thrust production and propulsive efficiency with the goal of determining the proper combinations of parameters that yields the maximum gain in efficiency.

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

Monday, November 21, 2016 10:40AM - 12:50PM –

Session H20 Bio:Biofilms and General Bioflows D137-138 - Jeff Eldredge, University of California, Los Angeles

10:40AM H20.00001 Fluid dynamic effects on staphylococci bacteria biofilms , ERICA SHERMAN, University of Nebraska - Lincoln, KENNETH BAYLES, JENNIFER ENDRES, University of Nebraska Medical Center, TIMOTHY WEI, University of Nebraska - Lincoln — *Staphylococcus aureus* bacteria are able to form biofilms and distinctive tower structures that facilitate their ability to tolerate treatment and to spread within the human body. The formation of towers, which break off, get carried downstream and serve to initiate biofilms in other parts of the body are of particular interest here. It is known that flow conditions play a role in the development, dispersion and propagation of biofilms in general. The influence of flow on tower formation, however, is not at all understood. This work is focused on the effect of applied shear on tower development. The hypothesis being examined is that tower structures form within a specific range of shear stresses and that there is an as yet ill defined fluid dynamic phenomenon that occurs hours before a tower forms. In this study, a range of shear stresses is examined that brackets 0.6 dynes/cm², the nominal shear stress where towers seem most likely to form. This talk will include PTV measurements and cell density data indicating variations in flow and biofilm evolution as a function of the applied shear. Causal relations between flow and biofilm development will be discussed.

10:53AM H20.00002 Effects of biofilm on flow over and through a permeable bed , FARZAN KAZEMIFAR, GIANLUCA BLOIS, MARCELO AYBAR, PATRICIA PEREZ-CALLEJA, ROBERT NERENBERG, Univ of Notre Dame, SUMIT SINHA, RICHARD HARDY, Durham Univ, JAMES BEST, Univ of Illinois, GREGORY SAMBROOK-SMITH, Univ of Birmingham, KENNETH CHRISTENSEN, Univ of Notre Dame — Biofilms constitute an important form of bacterial life in aquatic environments and are present at the interface of fluids and solids, such as riverbeds. Biofilms are permeable, heterogeneous, and deformable structures that can influence the flow and mass/momentun transport, yet their interaction with flow is not fully understood in part due to technical obstacles impeding quantitative experimental investigations. The porosity of river beds results in the generation of a diverse mosaic of 'suction' and 'ejection' events that are far removed from typical assumptions of turbulent flow structure over an impermeable bed. In this work, the effect of biofilm on bed permeability is studied. Experiments are conducted in a closed water channel equipped with 4-cm-deep permeable bed models consisting of horizontal cylinders normal to the bulk flow direction, forming an idealized two-dimensional permeable bed. Prior to conducting flow experiments, the models are placed within an independent biofilm reactor to initiate and control the biofilm growth. Once a targeted biofilm growth stage is achieved, the models are transferred to the water channel and subjected to transitional and turbulent flows. Long-distance microscopic particle image velocimetry measurements are performed to quantify the effect of biofilm on the turbulence structure of the free flow as well as the freestream-subsurface flow interaction.

11:06AM H20.00003 High resolution PIV of flow over biofilm covered walls , JOEL HARTENBERGER, MARC PERLIN, STEVEN CECCIO, University of Michigan — Microbial, 'slime' biofilms detrimentally affect the performance of engineered systems used every day from heat exchangers to large ocean-going vessels. The presence of a slime layer on a pipe wall or external boundary often leads to a significant increase in drag and may alter the nature of the turbulence in the adjacent flow. Despite these consequences, relatively few efforts have been undertaken to understand the underlying physical processes which couple biofilm characteristics with increased drag and other alterations to the flow. Experiments performed in a 1:14 scale replica of the US Navy's Large Cavitation Channel (LCC) at the University of Michigan investigate the effect of biofilm composition, coverage and thickness on the development of an external turbulent boundary layer (TBL) through the use of conventional and micro PIV. A range of fields of view (FOVs) were used to capture both the inner and outer regions of the boundary layer. The fine resolution of micro PIV gives an in-depth look at the near-wall region of the flow and may provide evidence linking specific biofilm features with flow characteristics while the less resolved, larger FOVs capture flow behavior to the freestream. Measurement techniques used to characterize the biofilm will be presented along with a description of the mean flow and turbulent fluctuations in the TBL.

11:19AM H20.00004 The effects of an algal biofilm on the turbulent boundary layer at high Reynolds number¹ , ELIZABETH MURPHY, University of Virginia, JULIO BARROS, MICHAEL SCHULTZ, KAREN FLACK, CECILY STEPPE, 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 increased drag. As with other types of roughness on aquatic surfaces, biofilms increase skin friction and thus induce severe drag penalties. In fact, slime layers appear to induce greater drag than would be predicted by the roughness height alone. Our work indicates that this is likely due to two characteristics of algal biofilms: i) flexible streamers that protrude into the flow, and ii) the compliant nature of a biofilm layer. High resolution PIV was used to measure the turbulent boundary layer flow over diatomaceous biofilm grown under dynamic conditions. Local mean streamwise velocity profiles were used to estimate the local wall shear stresses and to determine the similarity between the inner and outer layers of the boundary layer and those of a smooth wall. Spatially explicit turbulent kinetic energy (TKE), Reynolds shear stress (RSS), swirling strength and quadrant analyses over the biofilm were compared to those over a smooth wall and a rigid mesh roughness. We found that the combination of canopy flow due to streamers coupled with compliant wall-flow interactions result in large wall shear stresses and higher turbulence.

¹Funding provided by the ONR NURP program and the NSF GRIP program.

11:32AM H20.00005 The impact of shearing flows on electroactive biofilm formation, structure, and current generation¹, A-ANDREW JONES, CULLEN BUIE, Massachusetts Institute of Technology — A special class of bacteria exist that directly produce electricity. First explored in 1911, these electroactive bacteria catalyze hydrocarbons and transport electrons directly to a metallic electron acceptor forming thicker biofilms than other species. Electroactive bacteria biofilms are thicker because they are not limited by transport of oxygen or other terminal electron acceptors. Electroactive bacteria can produce power in fuel cells. Power production is limited in fuel cells by the bacteria's inability to eliminate protons near the insoluble electron acceptor not utilized in the wild. To date, they have not been successfully evolved or engineered to overcome this limit. This limitation may be overcome by enhancing convective mass transport while maintaining substantial biomass within the biofilm. Increasing convective mass transport increases shear stress. A biofilm may respond to increased shear by changing biomass, matrix, or current production. In this study, a rotating disk electrode is used to separate nutrient from physical stress. This phenomenon is investigated using the model electroactive bacterium *Geobacter sulfurreducens* at nutrient loads comparable to flow-through microbial fuel cells. We determine biofilm structure experimentally by measuring the porosity and calculating the tortuosity from confocal microscope images. Biofilm adaptation for electron transport is quantified using electrical impedance spectroscopy. Our ultimate objective is a framework relating biofilm thickness, porosity, shear stress and current generation for the optimization of bioelectrochemical systems

¹The Alfred P Sloan Foundation MPH Program

11:45AM H20.00006 LES of Laminar-to-Turbulent Particle-Fluid Dynamics in Human and Nonhuman Primate Airways: Applications to Aerosolized Drug Delivery Animal Testing, TAYLOR GEISLER, SOURAV PADHY, ERIC SHAQFEH, Stanford Dept. of Chemical Engineering, GIANLUCA IACCARINO, Stanford Dept. of Mechanical Engineering — Both the human health benefit and risk from the inhalation of aerosolized medications is often predicted by extrapolating experimental data taken using nonhuman primates to human inhalation. In this study, we employ Large Eddy Simulation to simulate particle-fluid dynamics in realistic upper airway models of both humans and rhesus monkeys. We report laminar-to-turbulent flow transitions triggered by constrictions in the upper trachea and the persistence of unsteadiness into the low Reynolds number bifurcating lower airway. Micro-particle deposition fraction and locations are shown to depend significantly on particle size. In particular, particle filtration in the nasal airways is shown to approach unity for large aerosols (8 microns) or high-rate breathing. We validate the accuracy of LES mean flow predictions using MRV imaging results. Additionally, particle deposition fractions are validated against experiments in 3 model airways.

11:58AM H20.00007 Using Computational Fluid Dynamics to examine airflow characteristics in Empty Nose Syndrome¹, TIM FLINT, University of Canterbury, MAHDI ESMAILY-MOGHADAM, ANDREW THAMBOO, NATHALIA VELASQUEZ, JAYAKAR V. NAYAK, Stanford University, MATHIEU SELLIER, University of Canterbury, PARVIZ MOIN, Stanford University — The enigmatic disorder, empty nose syndrome (ENS), presents with a complex subjective symptom profile despite objectively patent nasal airways, and recent reports suggest that surgical augmentation of the nasal airway can improve quality of life and ENS-related complaints. In this study, computational fluid dynamics (CFD) was performed both prior to, and following, inferior turbinate augmentation to model the resultant changes in airflow patterns and better understand the pathophysiology of ENS. An ENS patient with marked reduction in ENS symptoms following turbinate augmentation was identified, and pre- and post-operative CT imaging was collected. A Finite element framework with the variational multiscale method (Esmaily-Moghadam, Comput. Methods Appl. Mech. Engrg. 2015) was used to compute the airflow, temperature, and moisture transport through the nasal cavity. Comparison of the CFD results following corrective surgery showed higher levels of airflow turbulence. Augmentation produced 50%, 25%, and 25% increases in root mean square pressure, wall shear stress, and heat flux respectively. These results provide insight into the changes in nasal airflow characteristics attainable through surgical augmentation, and by extension, how nasal airflow patterns may be distorted in the 'overly patent' airway of ENS patients.

¹Supported by Stanford University CTR and Fulbright New Zealand

12:11PM H20.00008 Numerical tool development of fluid-structure interactions for investigation of obstructive sleep apnea, CHIEN-JUNG HUANG, University of California, SUSAN WHITE, UCLA School of Dentistry, SHAO-CHING HUANG, UCLA Institute for Digital Research and Education, SANJAY MALLYA, UCLA School of Dentistry, JEFF ELDREDGE, University of California — Obstructive sleep apnea (OSA) is a medical condition characterized by repetitive partial or complete occlusion of the airway during sleep. The soft tissues in the upper airway of OSA patients are prone to collapse under the low pressure loads incurred during breathing. The ultimate goal of this research is the development of a versatile numerical tool for simulation of air-tissue interactions in the patient specific upper airway geometry. This tool is expected to capture several phenomena, including flow-induced vibration (snoring) and large deformations during airway collapse of the complex airway geometry in respiratory flow conditions. Here, we present our ongoing progress toward this goal. To avoid mesh regeneration, for flow model, a sharp-interface embedded boundary method is used on Cartesian grids for resolving the fluid-structure interface, while for the structural model, a cut-cell finite element method is used. Also, to properly resolve large displacements, non-linear elasticity model is used. The fluid and structure solvers are connected with the strongly coupled iterative algorithm. The parallel computation is achieved with the numerical library PETSc. Some two- and three- dimensional preliminary results are shown to demonstrate the ability of this tool.

12:24PM H20.00009 Clinical questions and the role CFD can play¹, SAIKAT BASU, PhD, JULIA S. KIMBELL, PhD, ADAM M. ZANATION, MD, CHARLES S. EBERT, MD, BRENT A. SENIOR, MD, School of Medicine, UNC Chapel Hill — Use of computational fluid dynamics has revolutionized our perspectives on flow problems in engineering. These tools are however still underused in exploring clinical questions. Here we present some representative CFD-based findings that can improve current clinical practice. Chronic rhinosinusitis (CRS) is a complex inflammatory disease affecting over 11 million Americans yearly. It obstructs sinus pathways, thus hindering ventilation and clearance. Prescribed topical medications are often ineffective even after surgeries, partially owing to scanty drug delivery to the affected areas. We focus on improving the use of the most frequently used topical nasal sprays. From computed tomography (CT) scans, we develop 3D sinonasal airway models on the medical imaging software MimicsTM, which are then meshed using ICEM-CFDTM followed by airflow and particle simulations on FluentTM (v.14.5, ANSYS, Inc.). The results quantify aerosol particle delivery to target cavities before and after surgical alleviation. Various combinations of breathing techniques and head-nozzle orientations can increase target-site particle deposition over depositions using prevalent physician recommendations, and our findings facilitate identification of such optimal conditions.

¹Supported by the National Institutes of Health (NIH) grant R01 HL122154. The content is solely the responsibility of the authors and does not necessarily represent the official views of the NIH.

12:37PM H20.00010 Multi-scale analysis of active turbulence in living fluids, AMIN DOOSTMOHAMMADI, University of Oxford, JAVIER URZAY, Stanford University, JULIA YEOMANS, University of Oxford — Pattern formation in biological fluids manifests in the form of spatio-temporal chaos and is considered as a new class of turbulent flows. Here, we investigate the similarities and distinctions between turbulent-like flows in living fluids at low Reynolds numbers and classic high-Reynolds turbulence using multi-scale statistical tools. Turbulent characteristics of active flows are compared in two and three dimensions. In particular, we quantify the intermittency of meso-scale turbulence and explore energy cascades in two and three dimensions. Energy fluxes associated with viscous dissipation and local energy injection from active particles are quantified, shedding light on inter-scale phenomena in chaotic biological fluids.

Monday, November 21, 2016 10:40AM - 12:50PM –

Session H21 Bubbles: General D139-140 - James Bird, Boston University

10:40AM H21.00001 The equilibrium shape of bubbles on curved interfaces¹, JAMES BIRD, DANIEL POE, PETER WALLS, Boston University — The equilibrium shape for a bubble resting at a free surface depends on a balance of hydrostatic and capillary forces, with the smallest bubbles approximating a sphere and a hemisphere for the largest. This shape has been shown to be important to several processes ranging from gas transfer across the thin film cap to the production of jet droplets. Past works calculating the equilibrium shape assume that the interface is flat. However, there are instances where the curvature of the boundary may be comparable to the bubble itself. For example, a bubble bursting on the surface of a rain droplet. Here we relax the assumption of a flat interface and extend the classic bubble shape calculations to account for a curved interface boundary. An understanding of the extent of this deformation and the precise equilibrium bubble shape is important to applications in fields ranging from air-sea exchange to combustion dynamics.

¹We acknowledge financial support from NSF Grant No. 1351466

10:53AM H21.00002 History effects on the gas exchange between a bubble and a liquid¹, SHIGAN CHU, Johns Hopkins University, ANDREA PROSPERETTI², University of Houston — History effects are a distinctive feature of diffusive processes. For a diffusing gas bubble at rest in a liquid, such effects arise when the concentration of dissolved gas at the bubble surface, connected to the gas pressure by Henry's law, depends on time. This time dependence can be caused by several factors, such as varying ambient pressure, mole fraction in a multicomponent gas bubble, surface tension and others. In this study we consider history effects in the three situations mentioned above. More specifically, rectified diffusion in an oscillating ambient pressure field is explored under conditions when the diffusion length is larger than the bubble radius. History effects are found to be important in determining the threshold conditions for rectified diffusion. In contrast, history effects are small in the other two cases.

¹Supported by the BP/The Gulf of Mexico Research Initiative through the University of Texas Marine Science Institute (DROPPS II consortium: Dispersion Research on Oil: Physics and Plankton Studies).

²Also: Faculty of Science and Technology, University of Twente, Enschede, The Netherlands

11:06AM H21.00003 Effect of surfactants on bubble collisions with an air-water interface, SHIYAN WANG, TIANQI GUO, SADEGH DABIRI, PAVLOS P. VLACHOS, AREZOO M. ARDEKANI, School of Mechanical Engineering, Purdue University, West Lafayette, IN 47907, USA — Collisions of bubbles on an air-water interface are frequently observed in natural environments and industrial applications. We study the coefficient of restitution of a bubble colliding on an air-water interface in the presence of surfactants through a combination of experimental and numerical approaches. In a high concentration surfactant solution, bubbles experience perfectly inelastic collisions, and bubbles are arrested by the interface after the collision. As the surfactant concentration decreases, collisions are altered to partially inelastic, and eventually, elastic collisions occur in the pure water. In a high concentration surfactant solution, the reduced bouncing is attributed to the Marangoni stress. We identify the Langmuir number, the ratio between absorption and desorption rates, as the fundamental parameter to quantify the Marangoni effect on collision processes in surfactant solutions. The effect of Marangoni stress on the bubble's coefficient of restitution is non-monotonic, where the coefficient of restitution first decreases with Langmuir number, and then increases.

11:19AM H21.00004 Visualization of airflow growing soap bubbles, HAMOOD AL RAHBI, MATTHEW BOCK, SANGJIN RYU, University of Nebraska-Lincoln — Visualizing airflow inside growing soap bubbles can answer questions regarding the fluid dynamics of soap bubble blowing, which is a model system for flows with a gas-liquid-gas interface. Also, understanding the soap bubble blowing process is practical because it can contribute to controlling industrial processes similar to soap bubble blowing. In this study, we visualized airflow which grows soap bubbles using the smoke wire technique to understand how airflow blows soap bubbles. The soap bubble blower setup was built to mimic the human blowing process of soap bubbles, which consists of a blower, a nozzle and a bubble ring. The smoke wire was placed between the nozzle and the bubble ring, and smoke-visualized airflow was captured using a high speed camera. Our visualization shows how air jet flows into the growing soap bubble on the ring and how the airflow interacts with the soap film of growing bubble.

11:32AM H21.00005 How to make a giant bubble, JUSTIN BURTON, STEPHEN FRAZIER, Department of Physics, Emory University — Soap and water solutions can form massive, free floating films encompassing volumes in excess of 50 m^3 with thicknesses of only 1-10 microns when mixed with polymeric additives. These films are interesting from a physical standpoint due to their long lifetime and stability in ambient environments. We have investigated a variety of mixtures which are deemed "optimal" for making large bubbles, such as solutions made from guar seeds and polyethylene oxide (PEO). Making a giant bubble requires a balance between viscous and elastic forces. Drawing out a large soap film requires a low-viscosity solution, while elasticity enhances stability. Using a combination of shear rheology, drop-based extensional rheology, and time-dependent thickness measurements, we found that "optimal" solutions showed similar extensional properties even though their shear viscosity differed by more than an order of magnitude. Soap and water solutions with polymers lived 2-3 times longer and drained more slowly than typical soap and water solutions, even though their initial thicknesses were similar. In addition, polymeric bubbles showed increased stability to aging in dry environments. By varying the molecular weight and concentration of PEO in the solutions, we are able to optimize the lifetime of the film and determine the best way to make a giant bubble.

11:45AM H21.00006 Long-life of a bubble on the surface of a water-alcohol mixture, GIBRAN RAGE, Universidad Nacional Autonoma de Mexico, J. FEDERICO HERNANDEZ-SANCHEZ, King Abdullah University of Science and Technology, MONICA M. WILHELMUS, University of California - Riverside, ROBERTO ZENIT, Universidad Nacional Autonoma de Mexico — The lifetime of superficial bubbles has been used traditionally to determine the alcohol content in distilled beverages and spirits. With the proper alcohol content, the bubbles, known as pearls, have a particularly long life which is much longer than that in either pure water or pure ethanol. To understand this peculiar behavior, we conducted controlled experiments in water-ethanol mixtures and in samples of mezcal, an artisanal agave spirit. We assess the effect of the changes in viscosity, surface tension and density of the liquids. Also, we analyzed the effects of surfactants and evaporation rate differences, which lead to Marangoni convection in the draining film.

11:58AM H21.00007 Purging dissolved oxygen by nitrogen bubble aeration , TATSUYA

YAMASHITA, KEITA ANDO, Department of Mechanical Engineering, Keio University — We apply aeration with nitrogen microbubbles to water in order to see whether oxygen gas originally dissolved in the water at one atmosphere is purged by the aeration. The concentration of dissolved oxygen (DO) is detected by a commercial DO meter. To detect the dissolved nitrogen (DN) level, we observe the growth of millimetre-sized bubbles nucleated at glass surfaces in contact with the aerated water and compare it with the Epstein-Plesset theory that accounts for DO/DN diffusions and the presence of the glass surfaces. Comparisons between the experiment and the theory suggest that the DO in the water are effectively purged by the aeration.

12:11PM H21.00008 “Jumping” of Bubbles in Viscoelastic Fluids with Elasticity¹ ,

DIMITRIOS FRAGGEDAKIS², YANNIS DIMITAKOPOULOS, JOHN TSAMOPOULOS, University of Patras — Recently, it has been shown that phenomena known to be observed in viscoelastic liquids (e.g. the negative wake formation past deformable or rigid obstacles, cusp formation in bubbles), are present in elastoviscoelastic materials too, due to elasticity effects, Fragedakis et al., *Soft Matter* (2016). Based on the numerical results for bubbles in viscoelastic materials, Fragedakis et al. *J. Fluid Mech.* (2016), we focus our study on the rise of a confined air bubble in materials which exhibit elasto-viscoelastic behavior. Based on the rheological data by Mougin et al. (2012), the present study examines the conditions under which cusped bubble shapes are evident in yield stress materials. Moreover, the distance of the yield surface to the bubble is found to play a crucial role in both the rise velocity and the shape of the bubble. Additionally, if the yield stress effects are increased, the rise velocity is found to exhibit a discontinuous behavior. The mechanism which leads to such phenomena is found to be related with that discussed in Fragedakis et al. (2016). Ultimately, the existence of the yield surface near the bubble enhances the formation of the negative wake, irrespective of the position of the confinement in relation to the air bubble.

¹Bilateral Greece-Israel program of GSRT and LIMMAT Foundation

²Membership Pending

12:24PM H21.00009 Growth and collapse of laser-induced bubbles in gas-supersaturated gelatin gels.¹ , KEITA ANDO, NOBUYUKI NAKAMURA, Department of Mechanical Engineering, Keio University

— We study, with experiments and theory, the growth and collapse of laser-induced bubbles in a gelatin gel. The gel sample is prepared so as to obtain gas supersaturation, according to a difference between heat and gas diffusion rates. Spherical gas bubbles are created by focusing a nano-second laser pulse at 532 nm into the gas-supersaturated gel. The bubble dynamics are recorded by a high-speed camera. To explore effects of the gel elasticity on the bubble collapse, the experimental observations are compared to an extended Rayleigh-Plesset model that accounts for linear/nonlinear elasticity of the gel surrounding bubbles.

¹This work is supported by JSPS KAKENHI Grant No. 25709008.

12:37PM H21.00010 Boundary effects on streaming flow around a bubble located at the velocity antinode of a standing wave , MOHAMMAD ALHAMLI, College of technological studies

— A stable bubble trapped in a standing sound wave with frequency less than the resonance frequency of the bubble will be located at the velocity antinode. Steady streaming flow will develop around the bubble and is directly dependent on the bubbles boundary. Four boundary conditions are possible: 1) nonpulsating; no slip, 2) nonpulsating; free shear, 3) pulsating; no slip, and 4) pulsating; free shear. To solve for these conditions, we expanded the equations of motion with the dimensionless lateral oscillation amplitude, ϵ , using the singular perturbation method. The lateral oscillation amplitude is much smaller than the bubble radius $\epsilon \ll a$. Additionally, for the third and fourth cases, the dimensionless radial oscillation amplitude was assumed to be small, $\epsilon' \ll 1$. We also assumed a moderate value for the frequency parameter, $|M|$, which is the ratio of the bubble radius to the viscous length. For the nonpulsating cases the streaming flow patterns were quadrupole and symmetric across the quadrants and the intensity increases as we increase the frequency parameter. When we introduced the pulsation, we noticed that the streaming was symmetric across the polar plane and asymmetric below the equatorial plane for midrange values of the frequency parameter.

Monday, November 21, 2016 10:40AM - 12:50PM – Session H22 Microscale Flows: Particles-Orientation and Self-assembly E141/142 - Minami Yoda, Georgia Institute of Technology

10:40AM H22.00001 Effect of flow parameters on assembly of colloidal particle bands in Poiseuille and electroosmotic flow¹ , ANDREW YEE, MINAMI YODA, Georgia Institute of Technology

— Recent evanescent-wave visualizations (that only image the first $\sim 1 \mu\text{m}$ next to the wall) have shown that dielectric colloidal particles in combined Poiseuille and electroosmotic flow of dilute suspensions through fused-silica channels (with a depth of $34 \mu\text{m}$) assemble into streamwise bands. These bands have cross-sectional dimensions of a few μm and length comparable to that of the channel (*i.e.*, a few cm). They are roughly periodic along the cross-stream direction, even though there are no external forces in this direction. For moderate electric fields $|E| < 120 \text{ V/cm}$, the time scales for band formation at a given channel location appear to scale with the inverse of the shear rate (determined by Poiseuille flow), or $\dot{\gamma}^{-1}$. The results also suggest that the average number of bands N in steady-state (over a field of view of $200 \mu\text{m}$ square) decreases linearly with increasing $|E|$. These trends are not observed at higher $|E|$ and lower $\dot{\gamma}$, corresponding to cases where $N < 5$. In some cases, a large number of bands appear within a few seconds, then completely “disappear” from the near-wall region, and a much smaller number of bands then re-appear after several seconds.

¹Supported by ARO

10:53AM H22.00002 How particle properties affect the assembly and characteristics of colloidal particle bands¹, MINAMI YODA, ANDREW YEE, Georgia Institute of Technology — The interaction of suspended particles with a planar wall is a classic problem of colloid science. Particle-wall interactions in a flowing suspension are a newer area of interest, motivated by applications in microfluidics. Recent studies show that radius $a = 245$ nm particles in a dilute (volume fraction $\varphi = 0.17\%$) suspension are attracted to the wall, form 1D “pearl chains,” then assemble into concentrated streamwise bands with a roughly constant cross-stream spacing in combined Poiseuille and electroosmotic flow through fused-silica microchannels. The bands only exist within a few μm of the wall, and occur above a minimum shear rate $\dot{\gamma}$ and electric field magnitude $|E|$. Attracting (*i.e.*, concentrating) the particles to (near) the wall is a prerequisite for band formation; however, bands are not observed in all cases when particles are attracted to the wall. Particle properties appear to have a significant effect on these phenomena: decreasing φ , for example, appears to increase both the minimum $\dot{\gamma}$ and $|E|$ for band formation. Results are presented on how the assembly and characteristics of the bands are affected by properties such as φ , a (where $a < 1 \mu\text{m}$), and zeta-potential ζ_p .

¹Supported by ARO

11:06AM H22.00003 Mind the gap: a flow instability controlled by particle-surface distance, MICHELLE DRISCOLL, New York Univ NYU, BLAISE DELMOTTE, Courant Institute, New York University, MENA YOUSSEF, STEFANO SACANNA, New York University, Chemistry Department, ALEKSANDAR DONEV, Courant Institute, New York University, PAUL CHAIKIN, New York Univ NYU — Does a rotating particle always spin in place? Not if that particle is near a surface: rolling leads to translational motion, as well as very strong flows around the particle, even quite far away. These large advective flows strongly couple the motion of neighboring particles, giving rise to strong collective effects in groups of rolling particles. Using a model experimental system, weakly magnetic colloids driven by a rotating magnetic field, we observe that driving a compact group of microrollers leads to a new kind of flow instability. First, an initially uniformly-distributed strip of particles evolves into a shock structure, and then it becomes unstable, emitting fingers with a well-defined wavelength. Using 3D large-scale simulations in tandem with our experiments, we find that the instability wavelength is controlled not by the driving torque or the fluid viscosity, but a geometric parameter: the microrollers distance above the container floor. Furthermore, we find that the instability dynamics can be reproduced using only one ingredient: hydrodynamic interactions near a no-slip boundary.

11:19AM H22.00004 Long-lived ”critters” formed by hydrodynamic clustering, BLAISE DELMOTTE, Courant Institute, NYU, MICHELLE DRISCOLL, NYU Physics, MENA YOUSSEF, STEFANO SACANNA, NYU Chemistry, ALEKSANDAR DONEV, Courant Institute, NYU, PAUL CHAIKIN, NYU Physics — Self-assembly in colloidal systems often requires finely tuning the interactions between particles. When colloids are active, or moving due to an external drive, the assembly is even harder to achieve. Here we show that long-lived compact motile structures, called “critters”, can be formed just with hydrodynamic interactions. They naturally emerge from a recently discovered fingering instability in a system of microrollers near a floor. Our 3D large-scale simulations show that these critters are a stable state of the system, move much faster than individual rollers, and quickly respond to a changing drive. The formation of critters is robust to any initial condition and our experiments suggest that similar structures are formed even in a thermal colloidal system. We believe the critters are a promising tool for microscopic transport, flow, aggregation and mixing.

11:32AM H22.00005 Numerical simulations of electric field driven hierarchical self-assembly of monolayers of binary mixtures of particles¹, EDISON AMAH, NAGA MUSUNURI, SHAHADAT HOSSAIN, IAN FISCHER, PUSHPENDRA SINGH, New Jersey Institute of Technology — We numerically study the process of self-assembly of particle mixtures on fluid-liquid interfaces when an electric field is applied in the direction normal to the interface. Lateral forces cause particles to self-assemble into molecular-like hierarchical arrangements consisting of composite particles arranged in a pattern. As in experiments, if the particles sizes differ by a factor of two or more, the composite particle has a larger particle at its core with several smaller particles forming a ring around it. Approximately same sized particles form chains (analogous to polymeric molecules) in which positively and negatively polarized particles alternate when their concentrations are approximately equal, but when their concentrations differ substantially the particles whose concentration is larger form rings around the particles whose concentration is smaller. In some instances, particle chains with a positively polarized particle at one end and a negatively particle at the other folded to form circular chains. For submicron particles, only when the electric field intensity is larger than a critical value required for overcoming Brownian forces, a hierarchical pattern consisting of composite particles will form.

¹The work was supported by National Science Foundation

11:45AM H22.00006 Light-structured colloidal assemblies¹, ANTOINE AUBRET, UCSD, YOUSSEF MENA, NYU department of Chemistry, SOPHIE RAMANANARIVO, UCSD, STEFANO SACANNA, NYU department of Chemistry, JEREMIE PALACCI, UCSD, PALACCI LAB TEAM, SACANNA LAB TEAM — Self-propelled particles (SPP) are a key tool since they are of relative simplicity as compared to biological micro-entities and provide a higher level of control. They can convert an energy source into motion and work, and exhibit surprising non-equilibrium behavior. In our work, we focus on the manipulation of colloids using light. We exploit osmotic and phoretic effects to act on single and ensemble of colloids. The key mechanism relies on the photocatalytic decomposition of hydrogen peroxide using hematite, which triggers the motion of colloids around it when illuminated. We use hematite particles and particles with photocatalytic inclusions (*i.e.* SPP). We first show that the interactions between hematite and colloidal tracers can be tuned by adjusting the chemical environment. Furthermore, we report a phototoxic behavior (migration in light gradient) of the particles. From this, we explore the effect of spatio-temporal modulation of the light to control the motion of colloids at the single particle level, and to generate self-assembled colloidal structures through time and space. The so-formed structures are maintained by phoretic and hydrodynamic forces resulting from the motion of each particles. Ultimately, a dynamic light modulation may be a route for the creation of active colloidal motion on a collective scale through the synchronization of the individual motions of SPP.

¹This work is supported by NSF CAREER DMR 1554724

11:58AM H22.00007 Five Degree of Freedom Fluorescence Localization of Ellipsoidal Particles¹, CRAIG SNOEYINK, MD. ANISUL ISLAM, GORDON CHRISTOPHER, Texas Tech University — Symmetry breaking non-spherical particles can exhibit unique behavior when self-assembling due to increased degrees of freedom. For example, ellipsoidal particles on a fluid interface exhibit mesostructures that are dependent upon the both the contact angle of the ellipsoidal particle as well as the orientation. However, measuring the three dimensional position and orientation of these particles can be challenging. Here we present preliminary results on five degree of freedom fluorescence measurements of ellipsoidal particles on a fluid interface. Using the Bessel Beam Microscopy system and a novel compressed sensing based image analysis algorithm we will demonstrate 3D localization of ellipsoidal particles with 50 nm accuracy as well as pitch and yaw measurements with a resolution of 10 and 1 degrees respectively. We will discuss the technique as well as its implications for our understanding of non-spherical particle interactions and assembly at interfaces.

¹This material is based upon work supported by the National Science Foundation under Grant No. 1604398

12:11PM H22.00008 Flow-driven Assembly of Microcapsule Towers, HENRY SHUM, ANNA BALAZS, University of Pittsburgh — Large populations of the slime mold, *Dictyostelium discoideum*, are able to aggregate over a surface and collectively form a long, vertical stalk. Inspired by this biological behavior, we develop a synthetic mechanism for assembling tower-like structures using microcapsules as the building blocks. We accomplish this in simulations by generating a fluid flow field that draws microcapsules together along a surface and lifts them up at a central point. We considered a fluid flow generated by the local release of a chemical species from a patch on the surface. The concentration gradient of the diffusing chemical species causes radial diffusioosmotic flow along the solid surface toward the patch. Adhesive interactions keep the microcapsules attached to the surface as they are drawn together above the patch. To build a tower-like structure, some of the microcapsules must detach from the surface but remain attached to the rest of the cluster. The upward directed fluid flow above the patch then draws out the cluster into a tower shape. The final morphology of the aggregate structure depends on the flow field, the adhesive capsule-capsule and capsule-surface interaction strengths, and the sedimentation force on the capsules. Tuning these factors changes the structures that are produced.

12:24PM H22.00009 Interactions of inertially focused particles, KAITLYN HOOD, Massachusetts Institute of Technology, MARCUS ROPER, University of California Los Angeles — In inertial microfluidic devices, fluid inertia aligns submerged particles to a finite number of streamlines. Once particles are aligned on a streamline, particle interactions produce regularly spaced chains of particles. We demonstrate that viscous particle-particle and particle-wall interactions, combined with inertial focusing, give rise to a set spacing length for two particles. This model shows how the spacing length scale depends on particle size and Reynolds number. We also show that for two particles of different sizes, a range of spacing lengths can be achieved by tuning the Reynolds number.

12:37PM H22.00010 Elongational Flow Assists with the Assembly of Protein Nanofibrils¹, NITESH MITTAL, AYAKA KAMADA, CHRISTOFER LENDEL, FREDRIK LUNDELL, DANIEL SODERBERG, KTH Royal Institute of Technology — Controlling the aggregation process of protein-based macromolecular structures in a confined environment using small-scale flow devices and understanding their assembly mechanisms is essential to develop bio-based materials. Whey protein, a protein mixture with β -lactoglobulin as main component, is able to self-assemble into amyloid-like protein nanofibers which are stabilized by hydrogen bonds. The conditions at which the fibrillation process occurs can affect the properties and morphology of the fibrils. Here, we show that the morphology of protein nanofibers greatly affects their assembly. We used elongational flow based double flow-focusing device for this study. In-situ behavior of the straight and flexible fibrils in the flow channel is determined using small-angle X-ray scattering (SAXS) technique. Our process combines hydrodynamic alignment with dispersion to gel-transition that produces homogeneous and smooth fibers. Moreover, successful alignment before gelation demands a proper separation of the time-scales involved, which we tried to identify in the current study. The presented approach combining small scale flow devices with in-situ synchrotron X-ray studies and protein engineering is a promising route to design high performance protein-based materials with controlled physical and chemical properties.

¹We acknowledge the support from Wallenberg Wood Science Center

Monday, November 21, 2016 10:40AM - 12:50PM – Session H25 Microscale Flows: Drops and Bubbles-II E145 - Siva Vanapalli, Texas Tech University

10:40AM H25.00001 Dynamics of droplet entrapment in a constricted microchannel, MEHDJI NEKOUEI, SWASTIKA BITHI, SIVA VANAPALLI, Texas Tech University, Department of Chemical Engineering — Droplet migration and clogging in confined geometries is a problem of fundamental importance in oil recovery and droplet microfluidics. A confined droplet flowing through a conduit can either be arrested at the constriction or squeeze through it. The dynamics of the trapped and squeezed states are expected to depend on capillary number, drop size, viscosity ratio. Although there have been a number of studies on the dynamics of droplets passing through a constriction, investigations of dynamics of trapped droplets in constricted microchannels is lacking. In this work, we performed three-dimensional simulations of droplet trapping and squeezing process in a constricted microchannel. We also conducted experiments to validate the key results of the simulations. We investigated the impact of different system parameters on the onset of droplet immobilization at the constriction. We found that the continuous phase flows through the corners of the droplet, i.e. gutter flows to play an important role in determining the transition between trapping and squeezing. Therefore we evaluated the effect of different system parameters on gutter flows and found that the hydrodynamic resistance of gutters depends on the viscosity, size and confinement of the droplet.

10:53AM H25.00002 Bubble nucleation in superhydrophobic microchannels due to subcritical heating¹, ADAM COWLEY, DANIEL MAYNES, JULIE CROCKETT, BRIAN IVERSON, Brigham Young University — We report on experiments that investigate the effects of heating on laminar flow in superhydrophobic (SH) microchannels. The parallel plate microchannels (180 μm spacing) consist of two surfaces: a rib/cavity structured SH surface and a smooth glass surface. The back of the SH surface is in contact with an aluminum strip that is heated and a camera is used to image through the glass surface to visualize the flow. Thermocouples embedded in the aluminum obtain the temperature profile along the length of the channel. The friction factor-Reynolds product ($f\text{Re}$) is obtained via pressure drop and volumetric flow rate measurements. Five surface types/configurations are investigated: smooth hydrophilic, smooth hydrophobic, SH with ribs perpendicular to the flow, SH with ribs parallel to the flow, and SH with both ribs parallel to the flow and sparse ribs perpendicular to the flow. Both degassed and air-saturated water are used. When air-saturated water is used, the cavities of the SH surfaces act as nucleation sites and air is desorbed out of the water. Depending on the surface type/configuration, large bubbles can form and result in a large increase in $f\text{Re}$ and channel surface temperatures. When degassed water is used no bubble nucleation is observed, however, the air trapped in the cavities of the SH surfaces is quickly absorbed and the surfaces transition to a wetted state.

¹This research was supported by the National Science Foundation (NSF) (Grant No. CBET-1235881)

11:06AM H25.00003 Droplet breakup dynamics of weakly viscoelastic fluids, KRISTIN MARSHALL, TRAVIS WALKER, Oregon State Univ — The addition of macromolecules to solvent, even in dilute quantities, can alter a fluid's response in an extensional flow. For low-viscosity fluids, the presence of elasticity may not be apparent when measured using a standard rotational rheometer, yet it may still alter the response of a fluid when undergoing an extensional deformation, especially at small length scales where elastic effects are enhanced. Applications such as microfluidics necessitate investigating the dynamics of fluids with elastic properties that are not pronounced at large length scales. In the present work, a microfluidic cross-slot configuration is used to study the effects of elasticity on droplet breakup. Droplet breakup and the subsequent iterated-stretching, where beads form along a filament connecting two primary droplets were observed for a variety of material and flow conditions. We present a relationship on the modes of bead formation and how and when these modes will form based on key parameters such as the properties of the outer continuous-phase fluid. The results are vital not only for simulating the droplet breakup of weakly viscoelastic fluids but also for understanding how the droplet breakup event can be used for characterizing the extensional properties of weakly-viscoelastic fluids.

11:19AM H25.00004 In-air microfluidics: Drop and jet coalescence enables rapid multi-phase 3D printing, CLAAS WILLEM VISSER, University of Twente and Harvard University, TOM KAMPERMAN, DETLEF LOHSE, MARCEL KARPERIEN, University of Twente, UNIVERSITY OF TWENTE COLLABORATION — For the first time, we connect and integrate the fields of microfluidics and additive manufacturing, by presenting a unifying technology that we call In-air microfluidics (IAMF). We impact two liquid jets or a jet and a droplet train while flying in-air, and control their coalescence and solidification. This approach enables producing monodisperse emulsions, particles, and fibers with controlled shape and size (10 to 300 m) and production rates 100x higher than droplet microfluidics. A single device is sufficient to process a variety of materials, and to produce different particle or fiber shapes, in marked contrast to current microfluidic devices or printers. In-air microfluidics also enables rapid deposition onto substrates, for example to form 3D printed (bio)materials which are partly-liquid but still shape-stable.

11:32AM H25.00005 An Integrated microfluidic platform for liquid droplet in gas flow generation with in liquid flow collection and manipulation, POOYAN TIRANDAZI, CARLOS H. HIDROVO, Northeastern University — Discretization of biological samples and chemical reactions within digital droplets is a powerful technique which has rapidly emerged in many biochemical syntheses. The ability to generate, manipulate, and monitor millions of microdroplets in a short time provides great potential for high throughput screening and detection in microbiology. Here we report a microfluidic device for the formation of uniform microdroplets (50 μ m-100 μ m) using a high speed gas as the continuous phase. Gas-borne droplets are generated in a chip-based flow-focusing device fabricated in PDMS, and travel along the gaseous microchannel and are subsequently captured within a second liquid phase. The droplets are then transferred and collected in a minichamber and move into the manipulation section for further processing operations on the drops. All these steps are performed automatically in a single multilayer chip. This integrated microfluidic platform for generation, collection, and manipulation of the droplets provides great opportunities for monitoring and detection of gas-analytes. Utilizing the generated picoliter airborne droplets feature lower reaction times and higher transfer rates as compared to conventional air sampling techniques. Thus, it can greatly facilitate the investigation of airborne analytes by interrogation of the digital droplets using different analytical techniques. Furthermore, the presented liquid-in-gas generation method can be utilized for production of oil-free microparticles and microcapsules used in the food industry and for drug delivery.

11:45AM H25.00006 Confinement and viscosity ratio effect on droplet break-up in a concentrated emulsion flowing through a narrow constriction, JIAN WEI KHOR, Department of Mechanical Engineering, Stanford University, YA GAI, Department of Aeronautics and Astronautics, Stanford University, SINDY TANG, Department of Mechanical Engineering, Stanford University — We describe the dimensionless groups that determine the break-up probability of droplets in a concentrated emulsion during its flow in a tapered microchannel consisting of a narrow constriction. Such channel geometry is commonly used in droplet microfluidics to investigate the content of droplets from a concentrated emulsion. In contrast to solid wells in multi-well plates, drops are metastable, and are prone to break-up which compromises the accuracy and the throughput of the assay. Unlike single drops, the break-up process in a concentrated emulsion is stochastic. Analysis of the behavior of a large number of drops ($N > 5000$) shows that the probability of break-up increases with applied flow rate, the size of the drops relative to the size of the constriction, and the viscosity ratio of the emulsion. We found that the break-up probability collapses into a single curve when plotted as a function of the product of capillary number, viscosity ratio, and confinement factor defined as the un-deformed radius of the drop relative to the hydraulic radius of the constriction. The results represent a critical step towards the understanding of the physics governing instability in concentrated emulsions.

11:58AM H25.00007 Phase field investigation of droplet formation in flow focusing devices, FENG BAI, XIAOMING HE, Missouri Univ of Sci & Tech, XIAOFENG YANG, University of South Carolina, CHENG WANG, Missouri Univ of Sci & Tech — The phase field (P-F) method has been increasingly utilized to simulate multiphase fluids in microscale devices. However, most of the current models are based on mathematical analysis; and the physical meaning and validation of phase field method are still unclear. We demonstrate a phase field model in a flow focusing device to investigate the physical application of P-F method and clarify the role of diffusive term in Cahn-Hilliard equation. A characteristic mobility is defined as the product of the mobility tuning parameter and the square of interfacial thickness. The characteristic mobility reflects the correct relaxation time of the interface. Through systematic numerical investigations, we find that the characteristic mobility should be kept as a constant in order to correctly capture the physical process of droplet formation. This criterion has been verified with simulations by employing different interfacial thicknesses. In addition, this phase field model is validated by experiments with comparison of the size of droplet, the velocity of droplet and the period of droplet formation process.

12:11PM H25.00008 Flow-induced differential lateral migration of deformable particles by inner/outer viscosity ratio¹, YENG-LONG CHEN, SHIH-HAO WANG, WEI-TING YEH, Inst of Physics Academia Sinica — We investigate the practicality of flow-driven separation of deformable particles (DP) such as cells, droplets, and capsules in microfluidic flow. We use lattice Boltzmann-immersed boundary method to model the hydrodynamic coupling between DP and the fluid. We find that whether a DP migrates towards the wall or to the center at steady state depends strongly on particle Reynolds number Re , capillary numbers Ca , and viscosity ratio λ . The lateral steady state position d^* and velocity is determined by the competition between the inertia- and deformation-driven forces. In the deformation-dominated regime ($Ca >> Re$), DP migrates towards the channel centerline and flow faster for sufficiently small λ . In the inertia-dominated regime ($Ca << Re$), d^* is between the channel center and the wall and flow slower for small λ . For sufficiently large λ , DP migrates towards the wall as the inertia-driven lift effects increase and the particle velocity decreases. In the intermediate regime ($Ca \sim Re$), we find that d^* has non-monotonic dependence on λ , leading to complicated dependence of particle velocity. We find that the non-monotonic trend is a consequence of inertia-deformation coupling, and only occurs if the inertia- and deformation-driven lift effects are comparable. This result could provide be further utilized for separating soft particles with different internal fluid property.

¹MOST Taiwan, NCTS

12:24PM H25.00009 Micro-droplet formation via 3D printed micro channel , ZHEN

JIAN, JIAMING ZHANG, ERQIANG LI, SIGURDUR T. THORODDSEN, King Abdullah University of Science and Technology — Low cost, fast-designed and fast-fabricated 3D micro channel was used to create micro-droplets. Capillary with an outer diameter of 1.5 mm and an inner diameter of 150 μm was inserted into a 3D printed cylindrical channel with a diameter of 2 mm. Flow rate of the two inlets, insert depth, liquid (density, viscosity and surface tension) and solid (roughness, contact angle) properties all play a role in the droplet formation. Different regimes - dripping, jetting, unstable state - were observed in the micro-channel on varying these parameters. With certain parameter combinations, successive formation of micro-droplets with equal size was observed and its size can be much smaller than the smallest channel size. Based on our experimental results, the droplet formation via 3D printed micro T-junction was investigated through direct numerical simulations with a code called Gerris. Reynolds numbers $Re = \rho UL/\mu$ and Weber numbers $We = \rho U^2 L/\sigma$ of the two liquids were introduced to measure the liquid effect. The parameter regime where different physical dynamics occur was studied and the regime transition was observed with certain threshold values. Qualitative and quantitative analysis were performed as well between simulations and experiments.

12:37PM H25.00010 Droplets coalescence at microchannel intersection chambers with different shapes¹ , ZHAOMIAO LIU, XIANG WANG, RENTUO CAO, YAN PANG, College of Mechanical Engineering and Applied Electronics, Beijing University of Technology, China — The influence of microchannel intersection chamber shape on droplets coalescence process is investigated in this study. Three kinds of chamber shapes (half-round, triangle and camber) are designed to realize head-on droplets coalescence. The coalescence processes are visualized with high-speed camera system and the internal flow patterns are resolved with micro-PIV system. Experimental analyses on droplets coalescence position, coalescence time and the critical conditions are discussed. Both direct coalescence and late coalescence can be observed in the camber junction while only the late coalescence is present for the half-round and the triangle junction. The critical capillary number Ca^* varies for different working systems or intersection shapes. Ca^* in the camber junction is larger than that in the other two junctions for each working system and it decreases with the increase of the viscosity ratios for each intersection shape. Moreover, the characteristics of the velocity fields for different coalescence cases are analyzed for in-depth understanding of the process.

¹The authors do appreciate the financial support of No.11572013 of National Nature Scicence Funding of China.

Monday, November 21, 2016 10:40AM - 12:50PM – Session H26 Suspensions: Theory and Modeling E146 - James Swan, Assistant Professor, Dept. of Chemical Engineering, MIT

10:40AM H26.00001 Rapid Calculation of Thermal Forces in Coarse Grained Simulation of Colloidal Particles , JAMES SWAN, ANDREW FIORE, Massachusetts Inst of Tech-MIT, ALEKSANDER DONEV, FLORENCIO BALBOA, New York University Courant Institute — In the presented work, we will demonstrate a spectrally accurate method for calculation of thermal forces in implicit solvent simulations of soft materials such as colloidal dispersions. For implicit solvent models, the stochastic forces must be drawn from a normal distribution whose covariance is a complicated function of the particle configuration. For a system of interacting N particles, drawing a single sample requires $O(N^3)$ operations, if numerically exact techniques from linear algebra are employed. So-called fast methods can approximate the sampling with roughly $O(N^m \log N)$ computational complexity, where m is a coefficient greater than one which depends on the configuration of the particles. The computational complexity of the presented approach is $O(N(\log N)^{d/(d+3)})$, where d is the fractal dimension of the particulate structures being modeled. Remarkably, this new approach adapts to the structure of the material under study by leveraging the algebraic structure of Ewald summation and balancing the computational effort spent evaluating near-field and far-field contributions to the hydrodynamic interactions among the suspended particles. Applications of this approach to modeling colloidal gelation and particulate suspensions will be discussed.

10:53AM H26.00002 Effect of weak fluid inertia upon Jeffery orbits , JONAS EINARSSON, Stanford University — We consider the rotation of small neutrally buoyant axisymmetric particles in a viscous steady shear flow. When inertial effects are negligible the problem exhibits infinitely many periodic solutions, the Jeffery orbits. We compute how inertial effects lift their degeneracy by perturbatively solving the coupled particle-flow equations. We obtain an equation of motion valid at small shear Reynolds numbers, for spheroidal particles with arbitrary aspect ratios. At small shear Reynolds numbers the preferred Jeffery orbit is tumbling for prolate spheroids, and log-rolling for moderately oblate particles (aspect ratio $\lambda > 1/7.3$). For thinner oblate particles both log-rolling and tumbling are stable, separated by an unstable limit cycle. [Einarsson, J., Candelier, F., Lundell, F., Angilella, J. R. and Mehlig, B., PoF 27 063301 (2015)] We solved this long-standing problem by considering the symmetries that constrain the solution. In this case the symmetries reduced the problem to only four scalar integrals. Here I introduce an alternative method that accounts for the symmetries and tensorial nature of the governing equations, enables perturbative calculation of Stokes' equations, and is suitable for computer algebra. [Einarsson, J., and Mehlig, unpublished (2016)]

11:06AM H26.00003 A Generalized Frictional and Hydrodynamic Model of the Dynamics and Structure of Dense Colloidal Suspensions , JOAO MAIA, ARMAN BOROMAND, BRANDY GROVE, Case Western Reserve University, SAFA JAMALI, Massachusetts Institute of Technology — We perform mesoscopic DPD simulations incorporating both hydrodynamic and frictional interparticle interactions to study the effect of interaction potential on the rheology and structure of dense frictional colloidal suspensions. In particular, we performed a series of viscosity and normal stress measurements in suspensions with different volume fractions and obtained, for the first time, a complete picture of the dynamic state and of the microstructure. We confirmed that N_1 for semi-dense suspensions stays negative and grows with shear rate, which is consistent with hydrocluster-induced shear-thickening. We show that CST in colloidal suspensions can be explained solely via hydrodynamics, frictional bonds being transient and negligible to the rheological response. In dense suspensions and close to the jamming transition however, friction is required to obtain DST and replicate the recently experimental findings of a transition from negative to positive N_1 . We prove that hydroclusters form first at low stresses; this brings the particles together, thus allowing frictional contacts to develop, eventually leading to DST. In addition, when each particle is subject to an average of one frictional contact, N_1 reverses its increase but remains negative; at approximately two frictional contacts, a percolating network forms and N_1 becomes positive.

11:19AM H26.00004 A particle-particle collision strategy for arbitrarily shaped particles at low Stokes numbers¹, MOHSEN DAGHOOGHI, IMAN BORAZJANI, State Univ of NY - Buffalo — We present a collision strategy for particles with any general shape at low Stokes numbers. Conventional collision strategies rely upon a short-range repulsion force along particles centerline, which is a suitable choice for spherical particles and may not work for complex-shaped particles. In the present method, upon the collision of two particles, kinematics of particles are modified so that particles have zero relative velocity toward each other along the direction in which they have the minimum distance. The advantage of this novel technique is that it guarantees to prevent particles from overlapping without unrealistic bounce back at low Stokes numbers, which may occur if repulsive forces are used. This model is used to simulate sedimentation of many particles in a vertical channel and suspensions of non-spherical particles under simple shear flow.

¹This work was supported by the American Chemical Society (ACS) Petroleum Research Fund (PRF) grant number 53099-DNI9. The computational resources were partly provided by the Center for Computational Research (CCR) at the University at Buffalo.

11:32AM H26.00005 The Bretherton Problem for a Vesicle, JOSEPH BARAKAT, Stanford University, ANDREW SPANN, University of Texas at Austin, ERIC SHAQFEH, Stanford University — The motion of a lipid bilayer vesicle through a circular tube is investigated by singular perturbation theory in the limit of vanishing clearance. The vesicle is treated as a sac of fluid enclosed by a thin, elastic sheet that admits a bending stiffness. It is assumed that the vesicle is axisymmetric and swollen to a near-critical volume such that the clearance “e” between the membrane and the tube wall is very small. In this limit, bending resistance is of negligible importance compared to the isotropic tension, allowing the vesicle to be treated as a “no-slip bubble.” The effective membrane tension is found to scale inversely with “e” raised to the 3/2 power with a comparatively weak Marangoni gradient. The extra pressure drop is found to have a leading contribution due to the cylindrical midsection, which scales inversely with “e,” as well as a correction due to the end caps, which scales inversely with the square root of “e.” The apparent viscosity is predicted as a unique function of the geometry. The theory exhibits excellent agreement with a simplified, “quasi-parallel” theory and with direct numerical simulations using the boundary element method. The results of this work are compared to those for bubbles, rigid particles, and red blood cells in confined flows.

11:45AM H26.00006 Time evolution of shear-induced particle margination and migration in a cellular suspension, QIN M. QI, ERIC S.G. SHAQFEH, Stanford University — The inhomogeneous center-of-mass distributions of red blood cells and platelets normal to the flow direction in small vessels play a significant role in hemostasis and drug delivery. Under pressure-driven flow in channels, the migration of deformable red blood cells at steady state is characterized by a cell-free or Fahraeus-Lindqvist layer near the vessel wall. Rigid particles such as platelets, however, marginate and thus develop a near-wall excess concentration. In order to evaluate the role of branching and design suitable microfluidic devices, it is important to investigate the time evolution of particle margination and migration from a non-equilibrium state and determine the corresponding entrance lengths. From a mechanistic point of view, deformability-induced hydrodynamic lift and shear-induced diffusion are essential mechanisms for the cross-flow migration and margination. In this talk, we determine the concentration distribution of red blood cells and platelets by solving coupled Boltzmann advection-diffusion equations for both species and explore their time evolution. We verify our model by comparing with large-scale, multi-cell simulations and experiments. Our Boltzmann collision theory serves as a fast alternative to large-scale simulations.

11:58AM H26.00007 Dispersion of solids in fracturing flows of yield stress fluids¹, SARAH HORMOZI, Ohio University, IAN FRIGAARD, University of British Columbia — Solids dispersion is an important part of hydraulic fracturing. Whereas many frac fluids are low-viscous others transport solids through increased viscosity. In this context, one method for influencing both dispersion and solids carrying capacity is to use a yield stress fluid as the frac fluid. We propose a model framework for this scenario and analyse one of the simplifications. A key effect of including a yield stress is to focus high shear rates near the fracture walls. In typical fracturing flows this results in a large variation in shear rates across the fracture. In using shear-thinning viscous frac fluids, flows may vary significantly on the particle scale, from Stokesian behaviour to inertial behaviour across the width of the fracture. Equally, according to the flow rates, Hele-Shaw style models give way at higher Reynolds number to those in which inertia must be considered. We develop a model framework able to include this range of flows and make estimates of the streamwise dispersion in various relevant scenarios.

¹Schlumberger Oilfield Services, NSF and ACS PRF.

12:11PM H26.00008 Dynamics of magnetic particles suspended in Newtonian fluids under magnetic field, MINGYANG TAN, TRAVIS WALKER, Oregon State Univ — Anisotropic structures are commonly found in natural materials. Researchers are committed to developing meta-materials that mimic natural materials by introducing anisotropic filler particles. These materials can exhibit enhanced magnetic, mechanical, optical, and diffusive properties. In this study, a magnetic field is used to align magnetic oblate spheroids. We present an analytic solution based on a single-particle Stokes-flow model that describes the planar alignment of the particle in a rotating magnetic field. The analytic solution covers the full range of the magnetic field frequency agreeing well with our experimental results. Asymptotic solutions are also developed at both the high-frequency and the low-frequency limits of the field. The induced dipole of each particle can create its own magnetic field that can interact with neighboring particles, causing particles to aggregate. Different structures of particles are formed depending on the characteristics of the field, i.e., one-dimensional columns of particles in a constant field and two-dimensional sheets of particles in a rotating field. To simulate the realistic dynamics of the phenomena, we include hydrodynamic interactions between the particles via Stokesian dynamics.

12:24PM H26.00009 Particle cage dynamics in flowing colloidal dispersions, STEPHANIE MARENNE, JEFFREY F. MORRIS, Levich Institute / Chemical Engineering Department, City College of New York — The idea of the particle in a suspension at rest being trapped in a cage formed by its neighbors, widely used to understand glassy suspensions, has been applied to freely flowing suspensions. Stokesian Dynamics, a discrete particle simulation, is used to simulate the flow of monodisperse colloidal hard sphere suspensions. The cage analogy is useful to study the nonlinear stress in the material during start-up of shear flow, where the neighbor cage deforms and breaks, and during oscillatory shear flow where, depending on the amplitude of oscillation, the particle is trapped inside the cage or escapes during the oscillation cycle. A precise statistical definition of the cage in terms of the nearest neighbor ring in the pair distribution function is developed. We examine the dependence of the cage dynamics on the volume fraction of particles and the Peclet number P_e , the ratio between shear and Brownian forces. Under flow, the cage is found to break at quite definite positions, and the structural distortion is found to be clearly related to the shear and normal stress response. The shear strain needed to break the neighbor cage depends on P_e as Brownian motion enhances the total deformation. A simple model captures the strain at the stress overshoot for start-up of steady shear.

12:37PM H26.00010 Diffusion nearby elastic cell membranes, ABDALLAH DADDI-MOUSSA-IDER, ACHIM GUCKENBERGER, STEPHAN GEKLE, Biofluid Simulation and Modeling, Universität Bayreuth, Universitätsstraße 30, 95440, Germany — The physical approach of a small particle to the cell membrane represents the crucial step before active internalization and is governed by Brownian diffusion. Using a fully analytical theory, we show that the stretching and bending of cell membranes induces a long-lived subdiffusive behavior on the nearby particle (1,2). Such behavior is qualitatively different from the normal diffusion in a bulk fluid or near a hard-wall. the scaling exponent of the mean-square displacement can go as low as 0.87 in the perpendicular and 0.92 in the parallel direction. Moreover, we investigate the hydrodynamic interaction between two particles finding that the steady motion of two particles towards an elastic membrane possessing only shearing resistance leads to attractive interaction in contrast to the hard-wall case where the interaction is known to be repulsive. Our analytical predictions are compared with boundary-integral simulations where an excellent agreement is obtained (3).

References

- (1) Daddi-Moussa-Ider, A., Guckenberger, A. and Gekle, S., Phys. Rev. E 93, 012612 (2016)
- (2) Daddi-Moussa-Ider, A., Guckenberger, A. and Gekle, S., Phys. Fluids 28, 071903 (2016)
- (3) Daddi-Moussa-Ider, A. and Gekle, S., J. Chem. Phys., 145, 014905 (2016)

Monday, November 21, 2016 10:40AM - 12:37PM –
Session H27 Non-Newtonian Flows: Experimental Measurements and Numerical Studies E147-148 - Juan Carlos del Alamo, University of California, San Diego

10:40AM H27.00001 Two-Point Particle Tracking Microrheology of Nematic Lyotropic Liquid Crystals, MANUEL GOMEZ-GONZALEZ, Institute for Bioengineering in Catalonia, JUAN CARLOS DEL ALAMO, University of California at San Diego — Biological and technological complex fluids that are usually available in microscopic amounts (e.g. liquid crystals and biopolymer networks) can exhibit microstructural order leading to nematic rheological behavior. However, current microrheological methods cannot measure their directional viscoelastic coefficients. We recently introduced a directional two-point particle-tracking microrheology (D2PTM) technique to determine these coefficients (1). Here, we experimentally validate D2PTM by applying this method to disodium cromoglycate (DSCG), a lyotropic chromonic nematic liquid crystal that has recently sparked attention due to its biocompatibility and other interesting properties. We chose DSCG because its directional viscosity coefficients have been previously characterized by dynamic light scattering and are available in the literature. Our results suggest that D2PTM measurements agree well with measurements from previous methods. Furthermore, this new technique provides additional information about the microrheological response of nematic fluids that was not accessible via previous methods. (1) Gomez-Gonzalez, M and del Alamo, J C, "Two-point particle tracking microrheology of nematic complex fluids" Soft Matter 2016 12, 5758.

10:53AM H27.00002 Rate of chaotic mixing in localized flows¹, PIERRE JOP, JALILA BOUJEL, EMMANUELLE GOUILLART, FRANCK PIGEONNEAU, Surface du Verre et Interfaces, CNRS, Saint-Gobain, SURFACE DU VERRE ET INTERFACES TEAM — Most of the pastes in building materials are yield-stress fluids. Mixing them efficiently is required for industrial processes but linking the rate of the mixing to the fluid properties is a challenge. We study experimentally the rate of chaotic mixing in viscoplastic fluids by using a rod-stirring protocol with a rotating vessel. Only a limited zone localized around the stirring rods is highly sheared at a given time. Using a dyed spot as the initial condition, we measure the decay of concentration fluctuations of dye as mixing proceeds. Due to numerical simulations and experimental measurements, we relate the volume of highly sheared fluid to the parameters of the flow. We propose a quantitative two-zone model for the mixing rate, taking into account the geometry of the highly sheared zone as well as the rate at which fluid is renewed inside this zone. The model predicts correctly the scaling of the exponential mixing rates during a first rapid stage and a second slower one. Moreover we show that an optimal mixing exists when varying the ratio of the rotation rate of the vessel and the velocity of the rods.

¹French ANR (ANR-11-JS09-015)

11:06AM H27.00003 Spatial-temporal dynamics of Newtonian and viscoelastic turbulence in channel flow, SUNG-NING WANG, ASHWIN SHEKAR, MICHAEL GRAHAM, University of Wisconsin-Madison — Introducing a trace amount of polymer into liquid turbulent flows can result in substantial reduction of friction drag. This phenomenon has been widely used in fluid transport; however, the mechanism is not well understood. Past studies have found that in minimal domain turbulent simulations, there are occasional time periods when flow exhibits features such as weaker vortices, lower friction drag and larger log-law slope; these have been denoted as "hibernating turbulence". Here we address the question of whether similar behavior arises spatio-temporally in extended domains, focusing on turbulence at friction Reynolds numbers near transition and Weissenberg numbers resulting in low-medium drag reduction. By using image analysis and conditional sampling tools, we identify the hibernating states in extended domains and show that they display striking similarity as those in minimal domains. The hibernating states among different Weissenberg numbers exhibit similar flow statistics, suggesting they are unaltered by low to medium viscoelasticity. In addition, the polymer is much less stretched during hibernation. Finally, these hibernating states vanish as Reynolds number increases. However, they reoccur and gradually become dominant with increasing viscoelasticity.

11:19AM H27.00004 Extensional properties of mobile polymer solutions, CHRISTOPHE TIREL, CORIA, MARIE-CHARLOTTE RENOULT, LOMC, CHRISTOPHE DUMOUCHEL, CORIA, OLIVIER CRUMEYROLLE, LOMC, DENIS LISIECKI, CORIA, INNOCENT MUTABAZI, LOMC — A deep understanding of the influence of viscoelasticity on the dynamics of liquid flows remains a challenge in the non-Newtonian fluid mechanics field. Previous work has revealed that the addition of minute amount (2.5 part per million) of high molecular weight polymer to water, forming a viscoelastic solution with strong extensional properties, modifies the fission process during droplet snap off with spectacular effects: inhibition of the singularity observed in the reference Newtonian case and formation of a long-lived (milli-second) filament. The measurement of the extensional properties for such mobile polymer solutions is one of the most pressing problem. Here, a global measurement technique, based on the multi-scale analysis of the capillary instability of a free falling jet of a mobile polymer solution, is introduced. The method of analysis allows the characterisation of the jet breakup mechanism from which the relaxation time of the polymer solution can be extracted. One of the advantages of the technique is the simple experiment it requires.

11:32AM H27.00005 Yielding transition of Carbopol gel in a vertical pipe¹ , YANG LIU,

JOHN R.DE BRUYN, Department of Physics and Astronomy, The University of Western Ontario, 1151 Richmond Street London, Ontario, Canada, N6A 3K7, JOHN DE BRUYN TEAM — We have investigated the yielding transition of a simple yield-stress fluid (Carbopol 940) in a vertical pipe. The Carbopol gel was displaced by a Newtonian liquid injected at a constant, controlled rate at the bottom of the pipe. Rough-and smooth-walled pipes were used to study the effects of wall boundary conditions. The pressure in the Carbopol was measured by a pressure gauge fixed on the pipe wall, and the velocity profile in the Carbopol was measured by particle-image velocimetry (PIV). When the Newtonian liquid was injected, the rate of pressure increase was initially high, then decreased to a constant slow rate at later times. A time t_c was defined by the intersection of straight lines fit to the pressure-time data at early and late times. In the rough pipe, the wall shear stress at t_c is equal to the yield stress, suggesting that this time corresponds to yielding of the fluid. The velocity profiles were parabolic before yielding, and nearly a plug-like afterwards. In the smooth pipe, the pressure and velocity profiles appeared to show similar behavior to that in the rough pipe, but the wall shear stress at t_c is substantially smaller than the yield stress and fluid motion was due to wall slip.

¹NSERC

11:45AM H27.00006 Time-Resolved imaging Studies of Laser-Induced Jet Formation in Non-Newtonian Liquid Films , EMRE TURKOZ, CRAIG ARNOLD, Princeton University — Blister-actuated laser-induced

forward transfer (BA-LIFT) is a nozzle-less printing technique that offers an alternative to inkjet printing. The lack of a nozzle allows for a wider range of inks since clogging is not a concern. In this work, a focused laser pulse is absorbed within a polymer layer coated with a thin liquid film. The pulse causes a rapidly expanding blister to be formed that induces a liquid jet. Various well-studied non-Newtonian solutions are tested to examine how the shear-thinning and shear-thickening characteristics affect jet formation. The time delay between pulses is varied along with the energy, and different regimes of transfer are identified. We explore how Ohnesorge number, Weber number and spot size affect the jet formation and evaluate parameters that lead to breakup of jets into droplets.

11:58AM H27.00007 Influence of temperature on the drainage of thermoresponsive polymer thin film , ADRIEN BUSSONNIERE, MATTHEW JACKMAN, HIN LONG LEUNG, BO LIU, QINGXIA LIU, PEICHUN AMY TSAI, Univ of Alberta — Due to their switchable stability under external excitation, responsive aqueous foams have recently raised interests in various applications, such as washing, cleaning and mineral recovery, where stable foam and controlled destabilization are required. In this work, we investigate the influence of the temperature and polymer concentration on gravitational thin film drainage using a thermoresponsive polymer. The dynamics of film thinning was recorded on the thin film using a thickness measurement method. We successively illuminate the film with three LEDs of different wavelengths. The absolute thickness was accurately deduced using the three interference patterns. The results show an increase of drainage rates with increasing temperature but insignificant influence of polymer concentration (in the range between 50 and 300 mg/L). The thinning process was twice faster above the LCST (lower critical solution temperature) than that at room temperature. Our results of the temperate-dependent drainage show that the thermoresponsive solubility of polymer plays a key role in thin film stability.

12:11PM H27.00008 Dynamics of complex fluids in rotary atomization , BAVAND KE-SHAVARZ, GARETH MCKINLEY, MIT, MIT, MECHANICAL ENGINEERING DEPARTMENT TEAM — We study the dynamics of fragmentation for different Newtonian and viscoelastic liquids in rotary atomization. In this process, at the rim of a spinning cup, the centripetal acceleration destabilizes the formed liquid torus due to the Rayleigh-Taylor instability. The resulting ligaments leave the liquid torus with a remarkably repeatable spacing that scales linearly with the inverse of the rotation rate. Filaments then follow a well-defined geometrical path-line that is described by the involute of the circle. Knowing the geometry of this phenomenon we derive the detailed kinematics of this process and compare it with the experimental observations. We show that the ligaments elongate tangentially to the involute of the circle and thin radially as they separate from the cup. A theoretical form is derived for the spatial variation of the filament deformation rate. Once the ligaments are far from the cup they breakup into droplets since they are not stretched fast enough (compared to the critical rate of capillary thinning). We couple these derivations with the known properties of Newtonian and viscoelastic liquids to provide a physical analysis for this fragmentation process that is compared in detail with our experiments.

12:24PM H27.00009 Nonlinear equilibrium states in viscoelastic Taylor-Couette flow , LAURA NICOLAOU, JACOB PAGE, Imperial College London, TAMER ZAKI, Johns Hopkins University — Viscoelastic Taylor-Couette flow exhibits a variety of instabilities, some of which can arise in the absence of inertia altogether. Nonlinear solutions of the equilibrium states are sought, where the distorted mean flow is unchanged and the finite-amplitude instability waves are saturated. A key assumption in the theory is that nonlinearity is restricted to the action of the perturbations on the mean flow. Therefore, the perturbation shape is preserved throughout its growth and saturation, and is specified as a weighted sum of the unstable, linear eigenmodes. At low elasticity, a single unstable mode exists in the form of a stationary Taylor vortex, and the predicted saturated state compares favourably with direct numerical simulation (DNS). DNS reveals that the higher harmonics are an order of magnitude weaker than the fundamental frequency, which substantiates the assumption adopted in the theory. At moderate elasticity, multiple modes become unstable, including axially-travelling elastic rolls and stationary vortices. Equilibrium solutions built from superpositions of these modes explain observations from experiments in the literature over a range of shear rates.

Monday, November 21, 2016 10:40AM - 12:50PM –

Session H28 Particle-laden Flows: General II F149 - Andrea Prosperetti, Johns Hopkins University

10:40AM H28.00001 Closing the reduced position-space Fokker-Planck equation for shear-induced diffusion using the Physalis method¹ , ADAM J. SIERAKOWSKI, Johns Hopkins University, LAURA J. LUKASSEN, Max Planck Institute for Dynamics and Self-Organization — In the shear flow of non-Brownian particles, we describe the long-time diffusive processes stochastically using a Fokker-Planck equation. Previous work has indicated that a Fokker-Planck equation coupling the probability densities of position and velocity spaces may be appropriate for describing this phenomenon (Lukassen & Oberlack, Phys. Rev. E 89, 2014). The stochastic description, integrated over velocity space to obtain a reduced position-space Fokker-Planck equation, contains unknown space diffusion coefficients. In this work, we use the Physalis method for simulating disperse particle flows (Sierakowski & Prosperetti, J. Comp. Phys., 2016) to verify the colored-noise velocity space model (an Ornstein-Uhlenbeck process) by comparing the simulated long-time diffusion rate with the diffusion rate proposed by the theory. We then use the simulated data to calculate the unknown space diffusion coefficients that appear in the reduced position-space Fokker-Planck equation and summarize the results.

¹This study was partially supported by US NSF grant CBET1335965.

10:53AM H28.00002 Near-wall effects for momentum, heat and mass transport in gas-particle suspensions at moderate Reynolds numbers¹, STEFAN RADL, FEDERICO MUNICCHI, Graz University of Technology, CHRISTOPH GONIVA, DCS Computing GmbH, CFDEMresearch GmbH — Understanding transport phenomena in fluid-particle systems is of primary importance for the design of large-scale equipment, e.g., in the chemical industry. Typically, the analysis of such systems is performed by numerically solving a set of partial differential equations modeling the particle phase and the fluid phase as interpenetrating continua. Such models require a number of closure models that are often constructed via spatial filtering of data obtained from particle-resolved direct numerical simulations (PR-DNS). In the present work we make use of PR-DNS to evaluate corrections to existing closure models. Specifically, we aim on accounting for wall effects on the fluid-particle drag force and the particle-individual Nusselt number. We then propose an improved closure model to be used in particle-unresolved Euler-Lagrange (PU-EL) simulations. We demonstrate that such an advanced closure should account for a dimensionless filter size, as well as a normalized distance from the wall. In addition, we make an attempt to model the filtered fluid velocity profile in wall-bounded suspension flows.

¹The authors acknowledge funding from the European Commission through FP7 Grant agreement no. 604656, as well as VSC-3 and dcluster.tugraz.at

11:06AM H28.00003 Large scale simulation of particle laden flows using surface-resolved unstructured overset meshes¹, WYATT HORNE, KRISHNAN MAHESH, University of Minnesota — Particle-laden flows often involve a large range of length scales spanning from large convective scales down to scales near that of the length scale of individual particles. Resolving the fluid features at and below that of individual particles for cases with many moving particles presents difficult numerical challenges. We present a method that seeks to simulate such cases for many moving particles ($O(100,000)$ particles) that uses surface-resolved unstructured overset meshes. Details of the method are overviewed including communication strategies, mesh connectivity, particle movement and the base fluid solver. Simulation results are presented from simulations of single moving particles and from large scale particle-resolved direct numerical simulations of many moving particles.

¹Supported by DOE

11:19AM H28.00004 A numerical study of initial-stage interaction between shock and particle curtain¹, XIAOLONG DENG, LINGJIE JIANG, Beijing Computational Science Research Center — High speed particulate flow appears in many scientific and engineering problems. Wagner et al. 2012 studied the planar shock - particle curtain interaction experimentally, found the movement and expansion of the particle curtain, together with the movement of shock waves. Theofanous et al. 2016 did similar experiments, discovered a time scaling that reveals a universal regime for cloud expansion. In these experiments, both the particle-fluid interaction and the particle-particle collision are not negligible, which make it challenging to be dealt with. This work aims to numerically study and understand this problem. Applying the stratified multiphase model presented by Chang & Liou 2007 and regarding one phase as solid, following Regele et al. 2014, we study the initial stage of a planar shock impacting on a particle curtain in 2D, in which the particles can be regarded as static so that the collision between particles are not considered. The locations of reflected shock, transmitted shock, and contact discontinuity are examined. The turbulent energy generated in the interacting area is investigated. Keeping the total volume fraction of particles, and changing the particle number, good convergence results are obtained. Effective drag coefficient in 1D model is also calibrated.

¹The authors acknowledge the support from National Natural Science Foundation of China (Grant No. 91230203).

11:32AM H28.00005 Particle-laden turbulence under radiation: toward a novel small-particle solar receiver¹, ARI FRANKEL, ALI MANI, GIANLUCA IACCARINO, Stanford University — In particle-based solar receivers, an array of mirrors focuses sunlight onto a falling curtain of particles in a duct that absorb the light and warm up. The heated particles can be stored for later energy extraction. In this work we consider a design concept in which the particles and air are in a co-flowing configuration, and as the particles are heated they conduct the energy to the surrounding air. The air-particle mixture can then be separated and the heated air used for energy extraction. To assess the viability of this energy concept we have developed a simulation capability to analyze the flow of small particles in a turbulent flow with radiation. The code combines a point-particle direct numerical simulation of the particle-air flow in the low Mach number limit with the discrete ordinates solution of the gray, quasi-steady radiative transfer equation. We will describe the individual solution components and the coupling methodology. We will then demonstrate some results from the replication of a lab-scale experiment of a laser diode array irradiating a transparent channel with a flowing air-particle mixture.

¹This work was supported by the Predictive Science Academic Alliance Program 2 at Stanford

11:45AM H28.00006 Radiation Transport Through a Particle Laden Turbulent Flow, HILARIO TORRES, GIANLUCA IACCARINO, Stanford University — Direct numerical simulations of a turbulent duct flow laden with small particles was performed at several Reynolds and Stokes numbers. After the flow reached a statistically stationary state the instantaneous particle positions were saved at several time steps. Separate radiative heat transfer calculations were performed to study the amount of absorbed radiation and the local heat flux. The radiative source was considered outside of the duct, and one wall contained a window from which thermal radiation streamed through. The fluid was treated as transparent and the instantaneous particle positions obtained from the DNS were used to build Eulerian absorption and scattering fields. A Monte Carlo ray tracing code has been developed and used to solve for the radiative intensity, incident radiation, and heat flux inside of the domain. The qualitative behavior of the radiation fields as the particle positions change due to the turbulence are discussed. The effects of changes in the resolution of the Eulerian mesh used to convert the Lagrangian particles into absorption and scattering fields are also presented. The sensitivity of the amount of total absorbed radiation in the domain is also discussed in both of the previously mentioned cases.

11:58AM H28.00007 Laser Induced Dual Fluorescence Ratiometric Technique for Mixing Characterization in Microfluidic Systems¹, DAVID BEDDING, CARLSO HIDROVO, Northeastern University

Increasing the rate of mixing within microfluidic systems is vitally important in understanding biological and chemical reaction kinetics and mechanisms. The small length scales characteristic of these systems which translate into highly viscous, Stokes flows result in mixing that is primarily dominated by diffusion. In order to counteract this, an approach that utilizes inertial droplet collisions to promote chaotic advection between two mixing species has been developed. A Laser-Induced Dual Fluorescence (LIDF) system in conjunction with a high-speed camera and appropriate optics are used to capture two intensity fields providing information about the mixing process as well as the excitation intensity field over the volume of interest. The rate of mixing for the coalescing droplets was quantified by taking the standard deviation of the first intensity field over time, while the second intensity field provides information about the intensity field. A ratiometric imaging approach allows removal of mixing fluorescence signal noise in the form of variation in excitation intensity, primarily from the lasing patterns and lensing effects within the interrogation volume.

¹NSF CAREER Award Grant CBET 1151091

12:11PM H28.00008 Impact of nongray multiphase radiation in pulverized coal combustion¹, SOMESH ROY, Marquette University, BIFEN WU, University of Connecticut, MICHAEL MODEST, University of California, Merced, XINYU ZHAO, University of Connecticut — Detailed modeling of radiation is important for accurate modeling of pulverized coal combustion. Because of high temperature and optical properties, radiative heat transfer from coal particles is often more dominant than convective heat transfer. In this work a multiphase photon Monte Carlo radiation solver is used to investigate and to quantify the effect of nongray radiation in a laboratory-scale pulverized coal flame. The nongray radiative properties of carrier phase (gas) is modeled using HITEMP database. Three major species – CO, CO₂, and H₂O – are treated as participating gases. Two optical models are used to evaluate radiative properties of coal particles: a formulation based on the large particle limit and a size-dependent correlation. Effect of scattering due to coal particle is also investigated using both isotropic scattering and anisotropic scattering using a Henyey-Greenstein function. Lastly, since the optical properties of ash is very different from that of coal, the effect of ash content on the radiative properties of coal particle is examined.

¹This work used Extreme Science and Engineering Discovery Environment (XSEDE), which is supported by National Science Foundation grant number ACI-1053575

12:24PM H28.00009 Transport of inertial anisotropic particles under surface gravity waves, MICHELLE DIBENEDETTO, JEFFREY KOSEFF, NICHOLAS OUELLETTE, Stanford University — The motion of neutrally and almost-neutrally buoyant particles under surface gravity waves is relevant to the transport of microplastic debris and other small particulates in the ocean. Consequently, a number of studies have looked at the transport of spherical particles or mobile plankton in these conditions. However, the effects of particle-shape anisotropy on the trajectories and behavior of irregularly shaped particles in this type of oscillatory flow are still relatively unknown. To better understand these issues, we created an idealized numerical model which simulates the three-dimensional behavior of anisotropic spheroids in flow described by Airy wave theory. The particles response is calculated using a simplified Maxey-Riley equation coupled with Jefferys equation for particle rotation. We show that the particle dynamics are strongly dependent on their initial conditions and shape, with some additional dependence on Stokes number.

12:37PM H28.00010 Internal Combustion Engines as Fluidized Bed Reactors, ZOE LAVICH, ZACHARY TAIE, Oregon State University, SHYAM MENON, Louisiana State University, WALTER BECKWITH, SHANE DALY, Oregon State University, DEVIN HALLIDAY, Gas Technology Institute, CHRISTOPHER HAGEN¹, Oregon State University — Using an internal combustion engine as a chemical reactor could provide high throughput, high chemical conversion efficiency, and reactant/product handling benefits. For processes requiring a solid catalyst, the ability to develop a fluidized bed within the engine cylinder would allow efficient processing of large volumes of fluid. This work examines the fluidization behavior of particles in a cylinder of an internal combustion engine at various engine speeds. For 40 micron silica gel particles in a modified Megatech Mark III transparent combustion engine, calculations indicate that a maximum engine speed of about 60.8 RPM would result in fluidization. At higher speeds, the fluidization behavior is expected to deteriorate. Experiments gave qualitative confirmation of the analytical predictions, as a speed of 48 RPM resulted in fluidized behavior, while a speed of 171 RPM did not. The investigation shows that under certain conditions a fluidized bed can be obtained within an engine cylinder.

¹Corresponding Author

Monday, November 21, 2016 10:40AM - 12:50PM – Session H29 CFD: LES, DNS, Hybrid RANS/LES F150 - Inanc Senocak, Boise State University

10:40AM H29.00001 Evaluation of a general hybrid RANS/LES model in smooth wall reattachment, SIGFRIED HAERING, The University of Texas at Austin, ROBERT MOSER, Institute for Computational Engineering and Sciences, The University of Texas at Austin — Hybrid RANS/LES modeling approaches often exhibit deficiencies when used for common problems of engineering interest containing flow features such as unsteady smooth-wall separation and reattachment with non-trivial domains and discretization. Often, problem specific modifications and tuning must be employed rendering these models ineffective as generally predictive tools. A new broadly applicable hybrid RANS/LES modeling approach that is being developed to specifically address challenges associated with complex geometries and flows is presented. In general, the approach seeks to a balance between theoretical and actual modeled turbulent kinetic energy provided information from the underlying turbulence model, the resolved turbulence, and the available resolution. Anisotropy in the grid and resolved field are directly integrated into this balance. Here, we examine model performance with the case of a wall-mounted smooth hump of Greenblatt et.al. [1]. Excellent agreement with experimental results is attained while significantly outperforming delayed detached eddy simulation (DDES) for nearly the same computational expense and without any problem-specific modifications. [1] D. Greenblatt, K. B. et. al., "Experimental investigation of separation control part 1: Baseline and steady suction," AIAA Journal, vol. 44, no. 12, pp. 2820–2830, 2006.

10:53AM H29.00002 A new algebraic transition model based on stress length function, MENG-JUAN XIAO, ZHEN-SU SHE, SKLTCS, COE, Peking Univ. — Transition, as one of the two biggest challenges in turbulence research, is of critical importance for engineering application. For decades, the fundamental research seems to be unable to capture the quantitative details in real transition process. On the other hand, numerous empirical parameters in engineering transition models provide no unified description of the transition under varying physical conditions. Recently, we proposed a symmetry-based approach to canonical wall turbulence based on stress length function, which is here extended to describe the transition via a new algebraic transition model. With a multi-layer analytic form of the stress length function in both the streamwise and wall normal directions, the new model gives rise to accurate description of the mean field and friction coefficient, comparing with both the experimental and DNS results at different inlet conditions. Different types of transition process, such as the transition with varying incoming turbulence intensities or that with blow and suck disturbance, are described by only two or three model parameters, each of which has their own specific physical interpretation. Thus, the model enables one to extract physical information from both experimental and DNS data to reproduce the transition process, which may prelude to a new class of generalized transition model for engineering applications.

11:06AM H29.00003 Turbulent flow structure response to a varying wall-roughness arrangement: a modelling study, SUAD JAKIRLIC, BENJAMIN KRUMBEIN, Technical University of Darmstadt, Germany, POURYA FOOROGHI, FRANCO MAGAGNATO, BETTINA FROHNAPFEL, Karlsruhe Institute of Technology, Germany, DARMSTADT COLLABORATION, KARLSRUHE COLLABORATION — Presently adopted approach to the modelling of rough surfaces relies on introducing an additional drag term in the appropriately ‘filtered’ Navier-Stokes equations, accounting for the form drag and blockage effects, the roughness elements exert on the flow. A non-dimensional drag function $D(y)$ accounting for the shape of roughness elements is introduced. It is evaluated by applying a reference DNS of an open channel flow over a wall characterized by varying arrangement (aligned/staggered) of differently-shaped/sized roughness elements at a bulk Reynolds number $Re=6500$ by Fooroghi et al. (2016, 11th ETMM Symposium; an immersed boundary method is used to resolve the roughness geometry). The prime objective of the present work is to assess the roughness model capability to predict mean velocities and turbulent intensities in conjunction with a recently formulated hybrid LES/RANS (Reynolds-Averaged Navier-Stokes) model (Chang et al., 2014, IJHFF 49), based on the Very Large Eddy Simulation (VLES) concept of Speziale (1998, AIAA J. 36(2)). A seamless transition from RANS to LES is enabled depending on the ratio of the turbulent viscosities associated with the unresolved scales corresponding to the LES cut-off and those related to the turbulent properties of the VLES residual motion.

11:19AM H29.00004 Detached eddy simulation for turbulent fluid-structure interaction of moving bodies using the constraint-based immersed boundary method¹, NISHANT NANGIA, Department of Engineering Sciences and Applied Mathematics, Northwestern University, AMNEET P. S. BHALLA, BOYCE E. GRIFFITH, Department of Mathematics, University of North Carolina at Chapel Hill, NEELESH A. PATANKAR, Department of Mechanical Engineering, Northwestern University — Flows over bodies of industrial importance often contain both an attached boundary layer region near the structure and a region of massively separated flow near its trailing edge. When simulating these flows with turbulence modeling, the Reynolds-averaged Navier-Stokes (RANS) approach is more efficient in the former, whereas large-eddy simulation (LES) is more accurate in the latter. Detached-eddy simulation (DES), based on the Spalart-Allmaras model, is a hybrid method that switches from RANS mode of solution in attached boundary layers to LES in detached flow regions. Simulations of turbulent flows over moving structures on a body-fitted mesh incur an enormous remeshing cost every time step. The constraint-based immersed boundary (cIB) method eliminates this operation by placing the structure on a Cartesian mesh and enforcing a rigidity constraint as an additional forcing in the Navier-Stokes momentum equation. We outline the formulation and development of a parallel DES-cIB method using adaptive mesh refinement. We show preliminary validation results for flows past stationary bodies with both attached and separated boundary layers along with results for turbulent flows past moving bodies.

¹This work is supported by the National Science Foundation Graduate Research Fellowship under Grant No. DGE-1324585.

11:32AM H29.00005 A Split Forcing Technique to Reduce Log-layer Mismatch in Wall-modeled Turbulent Channel Flows¹, REY DELEON, University of Idaho, Boise State University, INANC SENOCAK, Boise State University — The conventional approach to sustain a flow field in a periodic channel flow seems to be the culprit behind the log-law mismatch problem that has been reported in many studies hybridizing Reynolds-averaged Navier-Stokes (RANS) and large-eddy simulation (LES) techniques, commonly referred to as hybrid RANS-LES. To address this issue, we propose a split-forcing approach that relies only on the conservation of mass principle. We adopt a basic hybrid RANS-LES technique on a coarse mesh with wall-stress boundary conditions to simulate turbulent channel flows at friction Reynolds numbers of 2000 and 5200 and demonstrate good agreement with benchmark data. We also report a duality in velocity scale that is a specific consequence of the split forcing framework applied to hybrid RANS-LES. The first scale is the friction velocity derived from the wall shear stress. The second scale arises in the core LES region, a value different than at the wall. Second-order turbulence statistics agree well with the benchmark data when normalized by the core friction velocity, whereas the friction velocity at the wall remains the appropriate scale for the mean velocity profile. Based on our findings, we suggest reevaluating more sophisticated hybrid RANS-LES approaches within the split-forcing framework.

¹Work funded by National Science Foundation under Grant No. 1056110 and 1229709. First author acknowledges the University of Idaho President’s Doctoral Scholars Award.

11:45AM H29.00006 A comparison study of convective schemes in hybrid RANS-LES calculations, BRANISLAV BASARA, AVL List GmbH, ZORAN PAVLOVIC, AVL-AST d.o.o. Slovenija — Nowadays it is commonly accepted to report on convection schemes in the case of Large Eddy Simulation (LES) calculations. However, in the case of hybrid RANS-LES calculations, the same discussion seems not to be relevant assuming that calculations are anyway performed on the coarser computational meshes and that the amount of unresolved and modelled turbulence impairs the calculation accuracy more than the error of convection schemes used in calculations. Therefore, we want to tackle this issue by using the Partially Averaged Navier-Stokes (PANS) model as the representative hybrid RANS-LES method but the conclusions derived in this work are equally applicable to other models. We will present results by using the central differencing (CD), MINMOD and SMART schemes but also using CD scheme only locally in the area of low unresolved-to-total ratios of kinetic energy (f_k). The paper will also show the performance of a step blending function, which depends on the prescribed constant value of the ratio f_k and the performance of a smooth function which directly uses the ratio f_k as the blending value. The results will be presented for the flow around the square cylinder.

11:58AM H29.00007 Performance investigation of multigrid optimization for DNS-based optimal control problems¹, CORNELIA NITA, KU Leuven, Mechanical Engineering, STEFAN VANDEWALLE, KU Leuven, Computer Science, JOHAN MEYERS, KU Leuven, Mechanical Engineering, Celestijnenlaan 300A, B3001 Leuven, Belgium — Optimal control theory in Direct Numerical Simulation (DNS) or Large-Eddy Simulation (LES) of turbulent flow involves large computational cost and memory overhead for the optimization of the controls. In this context, the minimization of the cost functional is typically achieved by employing gradient-based iterative methods such as quasi-Newton, truncated Newton or non-linear conjugate gradient. In the current work, we investigate the multigrid optimization strategy (MGOpt) in order to speed up the convergence of the damped L-BFGS algorithm for DNS-based optimal control problems. The method consists in a hierarchy of optimization problems defined on different representation levels aiming to reduce the computational resources associated with the cost functional improvement on the finest level. We examine the MGOpt efficiency for the optimization of an internal volume force distribution with the goal of reducing the turbulent kinetic energy or increasing the energy extraction in a turbulent wall-bounded flow; problems that are respectively related to drag reduction in boundary layers, or energy extraction in large wind farms. Results indicate that in some cases the multigrid optimization method requires up to a factor two less DNS and adjoint DNS than single-grid damped L-BFGS.

¹The authors acknowledge support from OPTEC (OPTimization in Engineering Center of Excellence, KU Leuven, grant no PFV/10/002).

12:11PM H29.00008 Machine Learning-Assisted Predictions of Turbulent Separated Flows over Airfoils¹, ANAND PRATAP SINGH, Univ of Michigan - Ann Arbor, SHIVAJI MEDIDA, Altair Engineering Inc., KARTHIK DURAISAMY, Univ of Michigan - Ann Arbor — RANS based models are typically found to be lacking in predictive accuracy when applied to complex flows, particularly those involving adverse pressure gradients and flow separation. A modeling paradigm is developed to effectively augment turbulence models by utilizing limited data (such as surface pressures and lift) from physical experiments. The key ingredients of our approach involve Inverse modeling to infer the spatial distribution of model discrepancies, and Neural networks to reconstruct discrepancy information from a large number of inverse problems into corrective model forms. Specifically, we apply the methodology to turbulent flows over airfoils involving flow separation. When the machine learning-generated model forms are embedded within a standard solver setting, we show that much improved predictions can be achieved, even in geometries and flow conditions that were not used in model training. The usage of very limited data (such as the measured lift coefficient) as an input to construct comprehensive model corrections provides a renewed perspective towards the use of vast, but sparse, amounts of available experimental datasets towards the end of developing predictive turbulence models.

¹This work was funded by the NASA Aeronautics Research Institute (NARI) under the Leading Edge Aeronautics Research for NASA (LEARN) program with Gary Coleman as the technical monitor.

12:24PM H29.00009 A Physics-Informed Machine Learning Framework for RANS-based Predictive Turbulence Modeling, HENG XIAO, JINLONG WU, JIANXUN WANG, Virginia Tech, JULIA LING, Sandia National Labs — Numerical models based on the Reynolds-averaged Navier-Stokes (RANS) equations are widely used in turbulent flow simulations in support of engineering design and optimization. In these models, turbulence modeling introduces significant uncertainties in the predictions. In light of the decades-long stagnation encountered by the traditional approach of turbulence model development, data-driven methods have been proposed as a promising alternative. We will present a data-driven, physics-informed machine-learning framework for predictive turbulence modeling based on RANS models. The framework consists of three components: (1) prediction of discrepancies in RANS modeled Reynolds stresses based on machine learning algorithms, (2) propagation of improved Reynolds stresses to quantities of interests with a modified RANS solver, and (3) quantitative, *a priori* assessment of predictive confidence based on distance metrics in the mean flow feature space. Merits of the proposed framework are demonstrated in a class of flows featuring massive separations. Significant improvements over the baseline RANS predictions are observed. The favorable results suggest that the proposed framework is a promising path toward RANS-based predictive turbulence in the era of big data. (SAND2016-7435 A)

12:37PM H29.00010 RANS simulations of variable density flows subject to a changing body forces and shocks, REBECCA BERTSCH, ROBERT GORE, 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. DNS and LES are ideal for studying these types of flows but are computationally expensive. RANS models have developed into accurate and efficient tools to investigate the evolution of turbulence in these complex flow problems and are well validated for prototypical variable density flows such as Rayleigh-Taylor and Richtmyer-Meshkov. However, most lack the ability to accurately capture mix features in VD flows subject to shocks and changing body forces. This talk will present results from a modified RANS model, which substitutes the molecular diffusion term in the species equation with a counter-gradient transport term that is dependent on the turbulent mass flux and species micro-densities. This modification better captures the mix physics across a range of Atwood numbers. Results from the new model will be presented for RM and RT and compared with DNS and experimental data.

Monday, November 21, 2016 10:40AM - 12:50PM –
Session H30 Granular Flows: Collision, Impact and Deformation F151 - Robert Behringer,
Duke University

10:40AM H30.00001 Pulling an intruder from a granular material¹, YUE ZHANG, R.P. BEHRINGER, Duke Univ — As a complement to 2D impact experiments, which involves a strongly fluctuating drag force involving collisional momentum transfer from intruder to grains, we consider a controlled 2D pull-out experiments, which is heuristically a reversed version of impact. During the pull-out experiment, a buried intruder is pulled out of a material, starting from rest. In the experiment, the intruder is subject to a gradually increasing upward vertical force, which we increase to the point where the intruder begins to accelerate upwards. To visualize this pulling process, we use 2D photoelastic disks from which circular intruders of different radii are pulled out. We will analyze the dynamics of the intruder and the structures of the force chains inside the granular system, which are captured by a high speed camera.

¹NSF-DMR-1206351, and the William M. Keck Foundation

10:53AM H30.00002 Dynamics of oblique impact in a 2D photoelastic granular medium¹, NOAH COX, Durham Academy, Durham, NC, XINYU WU, CACEY STEVENS BESTER, ROBERT BEHRINGER, Department of Physics, Duke University — Penetration of a solid projectile into dry granular matter is studied via two-dimensional impact experiments with bidisperse photoelastic grains. The drag force acting on the projectile is determined by high-speed imaging of the projectile's dynamics combined with the local granular response of the photoelastic grains. This force was previously shown using vertical impact to be the sum of a static, depth-dependent drag and a velocity-dependent inertial drag. Here the impact occurs obliquely, invoking a significant horizontal drag force that has not been fully explored. Accordingly we study the drag force model for oblique impact. We consider the influence of the projectile's impact speed and initial impact angle on its resultant trajectory. The path of the projectile changes drastically with impact angle. We therefore connect the effect of the impact angle to the nature of the drag force exerted on the projectile by the granular medium.

¹supported by Duke University Provost's Postdoctoral Program, NASA grant NNX15AD38G, and NSF-DMR-1206351

11:06AM H30.00003 Dynamics of drag force for projectile impact in granular media¹, LAUREN BEHRINGER, CACEY STEVENS BESTER, ROBERT BEHRINGER, Department of physics, Duke University — We study the way in which momentum is dissipated as a free-falling projectile impacts a dense granular target. An empirical force law has been widely accepted to describe this process, defining the stopping force as the sum of depth-dependent static force and velocity-dependent inertial drag. However, a complete understanding of the stopping force, incorporating grain-scale interactions during impact, remains unresolved. Using direct force measurements by way of a photoelastic imaging technique, we explore the complex fluctuating behavior of the forces acting on the projectile decelerating through a granular medium. Our results are used to study the static drag as the projectile comes to rest, as well as its connection to the effect of the container boundary of the granular target. We additionally vary the shape of the impeding object to infer intruder-grain interactions from force measurements.

¹Supported by Duke University Provosts Postdoctoral Program, NASA grant NNX15AD38G, NSF-DMR-1206351

11:19AM H30.00004 Optimize Operating Conditions on Fine Particle Grinding Process with Vertically Stirred Media Mill¹, YANG YANG, NEIL ROWSON, ANDY INGRAM, Univ of Birmingham — Stirred media mill recently is commonly utilized among mining process due to its high stressing intensity and efficiency. However, the relationship between size reduction and flow pattern within the mixing pot is still not fully understand. Thus, this work investigates fine particle grinding process within vertically stirred media mills by altering stirrer geometry, tip speed and solids loading. Positron Emitting Particle Tracking (PEPT) technology is utilized to plot routine of particles velocity map. By tacking trajectory of a single particle movement within the mixing vessel, the overall flow pattern is possible to be plotted. Ground calcium carbonate, a main product of Imerys, is chosen as feeding material (feed size D80 30um) mixed with water to form high viscous suspension. To obtain fine size product (normally D80 approximately 2um), large amount of energy is drawn by grinding mill to break particles through impact, shear attrition or compression or a combination of them. The results indicate higher energy efficient is obtained with more dilute suspension. The optimized stirrer proves more energy-saving performance by altering the slurry circulate.

¹Imerys Minerals Limited

11:32AM H30.00005 Biased Brownian motion in narrow channels with asymmetry and anisotropy, KIWING TO, Institute of Physics, Academia Sinica, ZHENG PENG, School of Physics and Electronics, Central South University — We study Brownian motion of a single millimeter size bead confined in a quasi-two-dimensional horizontal channel with built-in anisotropy and asymmetry. Channel asymmetry is implemented by ratchet walls while anisotropy is introduced using a channel base that is grooved along the channel axis so that a bead can acquire a horizontal impulse perpendicular to the longitudinal direction when it collides with the base. When energy is injected to the channel by vertical vibration, the combination of asymmetric walls and anisotropic base induces an effective force which drives the bead into biased diffusive motion along the channel axis with diffusivity and drift velocity increase with vibration strength. The magnitude of this driving force, which can be measured in experiments of tilted channel, is found to be consistent to those obtained from dynamic mobility and position probability distribution measurements. These results are explained by a simple collision model that suggests the random kinetic energies transfer between different translational degrees of freedom may be turned into useful work in the presence of asymmetry and anisotropy.

11:45AM H30.00006 Mechanisms of intruder motion in cyclically sheared granular media¹, HU ZHENG, Hohai University; Duke University, JONATHAN BARS , Duke University; Université de Montpellier, DONG WANG, ROBERT BEHRINGER, Duke University — We perform an experimental study showing how an intruder, a Teflon disk that experiences a moderate constant force, F , can advance through a granular material that is subject to quasi-static cyclic shear. The large Teflon disk is embedded in a layer of smaller bidisperse photoelastic disks. The granular medium and disk are contained in a horizontal cell, which is deformed from a square to a parallelogram and back again. The area of the cell remains constant throughout, and the protocol corresponds to cyclical simple shear. We find that the net intruder motion per cycle increases as a power law in N_c . The intruder motion relative to the granular background occurs primarily following strain reversals.

¹We acknowledge support from NSF Grant No. DMR1206351, NASA Grant No. NNX15AD38G and the W.M. Keck Foundation.

11:58AM H30.00007 Coefficient of restitution and surface deformation for inelastic particle collisions in a liquid, ANGEL RUIZ-ANGULO, Universidad Nacional Autonoma de Mexico, MELANY HUNT, California Institute of Technology — Granular flow simulations rely on models of the coefficient of restitution (ratio of rebound speed to impact speed). For collisions in a liquid, the rebound speed depends primarily on the impact Stokes number. In this study, we measure the coefficient of restitution for conditions in which two parameters are important: the Stokes number, St , and the ratio of impact velocity to yield velocity, $U^* = U_i/U_y$, where the yield velocity, U_y , is the maximum speed for an elastic collision. We also measure the surface deformation (crater depth and radius) for a range of St , U^* , and material properties. The results demonstrate that the surface deformation depends primarily on a corrected U^* , which incorporates the velocity decrease due to lubrication forces acting on the particle prior to collision. The particle rebound depends on the elastic strain energy stored during impact and on lubrication losses; these effects are incorporated in a model for the coefficient of restitution.

12:11PM H30.00008 Effect of bi-modality on sphere penetration into granular media

, NADIA KOURAYTEM, SIGURDUR THORODDSEN, King Abdullah University of Science and Technology, JEREMY MARSTON, Texas Tech University — We investigate the penetration of spheres impacting onto granular media, which are compositions of two discrete size ranges, thus creating a bi-modal material. We systematically vary the volume fraction of the two materials and measured the penetration depth over a wide range of impact speeds (0 to 5 m/s), as well as different sphere densities. We see maximum lubricating effects when we add 55% of the small 31-micrometer glass beads to 178-micrometer glass beads. Here the expected penetration depth increases by 60% when compared to a simple model, which is based on interpolating between the values obtained with the two pure monodisperse grains. Less effect is observed when 178-micrometer beads are mixed with a large 425-micrometer beads. We also study the lubrication effect of fine beads on larger rough Ottawa sand, seeing significant effects.

12:24PM H30.00009 Study of an athermal quasi static plastic deformation in a 2D granular material, JIE ZHANG, JIE ZHENG, shanghai jiao tong university — In crystalline materials, the plasticity has been well understood in terms of dynamics of dislocation, i.e. flow defects in the crystals where the flow defects can be directly visualized under a microscope. In a contrast, the plasticity in amorphous materials, i.e. glass, is still poorly understood due to the disordered nature of the materials. In this talk, I will discuss the recent results we have obtained in our ongoing research of the plasticity of a 2D glass in the athermal quasi static limit where the 2D glass is made of bi-disperse granular disks with very low friction. Starting from a densely packed homogeneous and isotropic initial state, we apply pure shear deformation to the system. For a sufficiently small strain, the response of the system is linear and elastic like; when the strain is large enough, the plasticity of the system gradually develops and eventually the shear bands are fully developed. In this study, we are particularly interested in how to relate the local plastic deformation to the macroscopic response of the system and also in the development of the shear bands.**12:37PM H30.00010 What causes the emergence of force chains in granular materials?**

, KRISHNARAJ KP, PRABHU R NOTT, Indian Institute of Science — A dense collection of grains may be viewed as a network of contacts which transmit forces. Force transmission in this network is influenced by constraints of geometry and packing, making it significantly different from information, transportation, or power networks. Experiments on two-dimensional disks report the presence of anisotropic quasi-linear structures called force chains, which are assumed to be the cause of their uncommon macroscopic behaviour. Studies have tried to quantify the properties of these structures, but the problem largely remains unresolved. We show using a simple discrete model, that force chain-like features are generic to any Euclidean packing of particles. The packing structure is sufficient to predict the essential features of the force network, regardless of the external forcing or boundary conditions. Using a novel method to study the structure of packing, we quantify the long range correlations in the system. The method reveals important, phase transition-like, properties in particle packings, the critical parameters and exponents of which characterize the geometry of the particle arrangements.

Monday, November 21, 2016 10:40AM - 12:24PM –

Session H31 Experimental Techniques - Laser/Wire Anemometry F152 - Laurent Mydlarski, McGill University

10:40AM H31.00001 High resolved velocity measurements using Laser Cantilever Anemometry, JAROSLAW PUCZYLOWSKI, MICHAEL HLLING, JOACHIM PEINKE, University of Oldenburg — We have developed

a new anemometer, namely the 2d-LCA (2d-Laser-Cantilever-Anemometer), that is capable of performing high resolved velocity measurements in fluids. The anemometer uses a microstructured cantilever made of silicon as a sensing element. The specific shape and the small dimensions (about 150m) of the cantilever allow for precise measurements of two velocity component at a temporal resolution of about 150kHz. The angular acceptance range is 180° in total. The 2d-LCA is a simple to use alternative to x-wires and can be used in many areas of operation including measurements in liquids or in 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. In the recent past new cantilever designs were implemented with the goal to further improve the angular resolution and increase the stability. In addition, we have designed more robust cantilevers for measurements in rough environments such as offshore areas. Successful comparative measurements with hot-wires have been carried out in order to assess the performance of the 2d-LCA.

10:53AM H31.00002 A nano cold-wire for velocity measurements, YI-CHUN HUANG, MATTHEW FU, YUYANG FAN, CLAYTON BYERS, MARCUS HULTMARK, Princeton University — We introduce a novel, strain-based sensor for both gaseous and liquid flows. The sensor consists of a free-standing, electrically conductive, nanoscale ribbon suspended between silicon supports. Due to its size, the nanoribbon deflects in flow under viscously dominated fluid forcing, which induces axial strain and a resistance change in the sensing element. The change in resistance can then be measured by a Wheatstone bridge, resulting in straightforward design and operation of the sensor. Since its operating principle is based on viscous fluid forcing, the sensor has high sensitivity especially in liquid or other highly viscous flows. A simple analytical model to understand the relation between forcing and strain is derived from the geometric and material constraints, and preliminary analysis using a low order model of the dynamic systems suggests that the sensor has a high frequency response. Lastly, a cylindrical structure to house the sensor with an axial and ventral channel to generate a pressure differential is being considered for typical velocimetry applications.**11:06AM H31.00003 ABSTRACT WITHDRAWN –****11:19AM H31.00004 Investigation of Constant Temperature Hot-wire System Response using Laser Pulse**, NICHOLAS JAFFA, Pennsylvania State Univ, SCOTT MORRIS, JOSHUA CAMERON, University of Notre Dame — Constant temperature hot-wire systems use a Wheatstone bridge and feedback amplifier circuit to maintain a constant average temperature across the wire yielding frequency responses of order 100 kHz. This high frequency response allows hot-wires to be used extensively for aerodynamic measurements in high speed flows and uncertainty at these high frequencies can be difficult to diagnose. The standard frequency response check for constant temperature hot-wires uses an electronic pulse across the circuit to check the electronic feedback circuit response time, but does not account for the impact of the heat transfer along the wire. In order to investigate the frequency response of the entire constant temperature hot-wire system, including the heat transfer along the wire, a novel method was developed using a pulsed PIV laser focused to illuminate only the hot-wire. The laser pulse duration was effectively an instantaneous change in wire surface temperature through radiation. A hot-wire was placed in a uniform open calibration jet for a range of flow conditions. The response of the entire hot-wire system was observed across a range of conditions including changes in flow, wire temperature, and thermal boundary conditions and compared with the electronic pulse test.

11:32AM H31.00005 Validation of a multi-sensor hotwire probe for boundary layer enstrophy measurements¹, SPENCER ZIMMERMAN, CALEB MORRILL-WINTER, University of Melbourne, JOSEPH KLEWICKI, University of Melbourne/University of New Hampshire — A multi-sensor hotwire probe capable of measuring the velocity and vorticity vectors has been designed and implemented in a turbulent boundary layer with the goal of elucidating the means by which the associated momentum transport is maintained under increasing scale separation between the velocity and vorticity fields with increasing Reynolds number. The capacity of this sensor to accurately measure each component of velocity and vorticity is first evaluated via synthetic experiment. The three-dimensional velocity field from the DNS of Sillero et al. (*Phys. Fluids* 25, 2013) is used to compute effective cooling for each sensor element, and the resulting signals are interpreted via two-dimensional calibration surfaces such as would be used to process physical experimental data. Results from this virtual validation experiment are presented and suggest the sensor is capable of resolving key features of the velocity and vorticity fields at physically achievable spatial resolutions. Results from measurements collected at the Flow Physics Facility (FPF) at the University of New Hampshire are presented alongside these predictions and exhibit very good agreement in trend, but with some differences in magnitude.

¹The support of the Australian Research Council and the National Science Foundation is gratefully acknowledged.

11:45AM H31.00006 Novel method and experimental validation of statistical calibration via Gaussianization in hot-wire anemometry, IGAL GLUZMAN, JACOB COHEN, YAAKOV OSHMAN, Technion Israel Institute of Technology — We introduce a statistical method based on Gaussianization to estimate the nonlinear calibration curve of a hot-wire probe, that relates the input flow velocity to the output (measured) voltage. The method uses as input a measured sequence of voltage samples, corresponding to different unknown flow velocities in the desired operational range, and only two measured voltages along with their known (calibrated) flow velocities. The novel method is validated against standard calibration methods using data acquired by hot-wire probes using wind-tunnel experiments. We demonstrate our new calibration technique by placing the hot-wire probe at certain region downstream of a cube-shaped body in a free stream of air flow. For testing our calibration method we rely on flow statistics that exist, among others, in a certain region of a turbulent wake formed downstream of the cube-shaped body. The specific properties are: first, the velocity signal in the wake should be as close to Gaussian as possible. Second, the signal should cover the desired velocity range that should be calibrated. The appropriate region to place our probe is determined via computation of the first four statistical moments of the measured signals in different regions of the wake.

11:58AM H31.00007 Recommendations for the design of interference probes for the simultaneous measurement of turbulent concentration and velocity¹, ALA'IS HEWES, LAURENT MYDLARSKI, McGill University — The present work focuses on the design and optimization of a thermal-anemometry-based interference probe used to simultaneously measure concentration and velocity at relatively high temporal and spatial resolutions in turbulent flows. Although a small number of similar measurements have been successfully performed, little work has been undertaken to investigate the design of such specialized probes, in which one hot-wire sensor is operated downstream of, and micrometers from, a second one. To this end, experiments performed in the non-buoyant region of a helium-air jet were undertaken to study the effects of overheat ratios, wire separation distances, wire diameters, and wire materials on the performance of interference probes. They revealed that accurate concentration and velocity measurements require that an interference probe have two wires of differing diameters with a small separation, of about $10 \mu\text{m}$, between the wires. Furthermore, the upstream wire should be operated at a high overheat ratio and the downstream wire at a low one. An optimal design for an interference probe is presented, and measurements made in a turbulent jet are used to benchmark its accuracy.

¹Supported by the Natural Sciences and Engineering Research Council of Canada (Grant 217184)

12:11PM H31.00008 Practical Considerations for Simultaneous LDV & PIV Measurements, STAMATIOS POTHOS, AARON BOOMSMA, DAN TROOLIN, TSI Inc — Simultaneous LDV and PIV measurements are useful for validation experiments and when correlating high temporal resolution measurements with large structures of the flow. Performing simultaneous LDV and PIV measurements can be a challenging task due to the differences in temporal and spatial resolution of each technique, as well as requirements for adequate signal. Even so, simultaneous hot-wire and PIV measurements is even more difficult. Unlike hot-wire, LDV is a non-intrusive technique that is unaffected by PIV laser light-sheet heating. Furthermore, hot-wire measurements are adversely affected by seeding particles in the flow required for PIV. In the present study, we discuss several practical considerations for performing simultaneous LDV and PIV measurements. We completed two separate experiments, each with different seeding densities, flow velocities, and working fluids. With these data sets, we studied the effects of temporal and spatial interpolation, up/down sampling, PIV window size and overlap on the simultaneous signals.

Monday, November 21, 2016 10:40AM - 12:50PM –
Session H32 Turbulent Boundary Layers: Spectral Analysis and Scaling Oregon Ballroom
201 - Nicholas Hutchins, University of Melbourne

10:40AM H32.00001 Comparison of spatial and temporal characteristics of a turbulent boundary layer in the presence of free-stream turbulence¹, EDA DOGAN, R. JASON HEARST, University of Southampton, RONALD E. HANSON, University of Toronto, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton — Free-stream turbulence (FST) has previously been shown to enhance the scale interactions occurring within a turbulent boundary layer (TBL). This is investigated further by generating FST with an active grid over a zero-pressure gradient TBL that developed on a smooth flat plate. Simultaneous measurements were performed using four hot-wires mounted to a rake that traversed the boundary layer height. Planar PIV measurements were also performed. Hot-wire measurements indicate that on average large-scale structures occurring in the free-stream penetrate the boundary layer and increase the streamwise velocity fluctuations throughout. Two-point correlations of the streamwise velocity fluctuations from the hot-wires enable determination of the inclination angle of the wall-structures in the boundary layer using Taylor's hypothesis. This angle is observed to be invariant around 11-15 degrees in the near-wall region in agreement with the literature for canonical TBLs. This presentation will compare the planar PIV data to these hot-wire measurements to determine if these phenomena that appear in the statistics using Taylor's hypothesis can be tracked to instantaneous spatial features in the TBL subjected to FST.

¹We acknowledge the financial support from the European Research Council (ERC grant agreement no. 277472), EPSRC (grant ref no: EP/I037717/1)

10:53AM H32.00002 Distance-from-the-wall scaling in turbulent boundary layers

RIO BAIDYA, JIMMY PHILIP, NICHOLAS HUTCHINS, JASON MONTY, IVAN MARUSIC, The University of Melbourne — An assessment of self-similarity in the inertial sublayer of turbulent boundary layers (TBL) is presented by simultaneously considering the streamwise and wall-normal velocities. Here, we utilise carefully conducted subminiature x -probe experiments at high Reynolds number. Moreover, the turbulent stresses are compared against results from a synthetic flow where the distance-from-the wall (z -) scaling is strictly enforced, following the Attached Eddy Hypothesis. We show that not all stresses approach the asymptotic solution at an equal rate as the friction Reynolds number (Re_τ) is increased. Specifically, the motions which contribute to the wall-normal variance and Reynolds shear stress are found to follow the asymptotic solution at a relatively lower Re_τ even when the streamwise variance does not, and this trend is attributed to the contribution from attached eddies. Based on these findings, the Reynolds shear stress cospectra, through its z -scaling, are used to assess the wall-normal limits where self-similarity applies within the TBL. The limits are found to be consistent with the recent observations that the self-similar region starts and ends at viscous scaled wall-distances of $\mathcal{O}(\sqrt{Re_\tau})$ and $\mathcal{O}(Re_\tau)$ respectively.

11:06AM H32.00003 Spectral link for the mean velocity profile in the atmospheric boundary layer¹

DONGRONG ZHANG, GUSTAVO GIOIA, PINAKI CHAKRABORTY, Okinawa Institute of Science and Technology — Turbulent flow in the atmospheric boundary layer is sheared and stratified. For this flow, we consider the mean velocity profile (MVP), the vertical profile of the time-averaged horizontal wind velocity. We employ the theoretical framework of the spectral link, originally proposed for MVP in sheared flows (Gioia et al., 2010) and later extended to stratified flows (Katul et al., 2011). Accounting for the whole structure of the turbulent energy spectrum—the energetic range, the inertial range, and the dissipative range—we examine the scaling of the MVP in the "wall coordinates" and in the Monin–Obukhov similarity coordinates, for both stable and unstable stratification. Our results are in excellent accord with field measurements and numerical simulations.

¹Okinawa Institute of Science and Technology

11:19AM H32.00004 Onset of spatio temporal disorder described by directed percolation¹

TOM WESTER, DOMINIK TRAPHAN, GERD GLKER, JOACHIM PEINKE, ForWind - Center for Wind Energy Research, Institute of Physics, University of Oldenburg, Germany, AG TWIST TEAM — The energy transport and mixing behavior of a fluid strongly depends on the state of the flow. These properties change drastically if the flow changes from laminar to turbulent state. This transition is a very complex and highly unsteady phenomenon, which is not fully understood up to now. The biggest problem is the characterization of the onset of spatio temporal disorder. This means that turbulent spots in the flow field irregularly spread or decay on their way downstream. In this presentation we will show that this critical behavior of turbulent spreading in the flow can be described by the directed percolation model. This approach was already used for a transitive channel flow, pipe flows or different couette flows. The charm of this model is the complete characterization of the whole transition with only a few unique exponents. In contrast to the majority of previous studies, the underlying data base of this study is acquired experimentally by high-speed Particle Image Velocimetry. Thus the evolving flow can be captured in a highly resolved spatio-temporal manner. In this way it is easily possible to determine the critical exponents which describe the transient area between laminar and turbulent flow. The results will be presented and compared to theoretical expectations.

¹DAAD, DFG

11:32AM H32.00005 Spectral stochastic estimation of high-Reynolds-number wall-bounded turbulence for a refined inner-outer interaction model

WOUTIJN J. BAARS, NICHOLAS HUTCHINS, IVAN MARUSIC, University of Melbourne — For wall-bounded flows, the model of Marusic, Mathis and Hutchins (2010) allows one to predict the statistics of the streamwise fluctuating velocity in the inner-region, from a measured input signal in the logarithmic region. Normally, a user-defined portion of the input forms the large-scale content in the prediction. Incoherent smaller scales are then fused to the prediction via universally expressed fluctuations that are subject to an amplitude modulation. Here we present a refined version of the model using spectral linear stochastic estimation, which eliminates a user-defined scale-separation of the input. An empirically-derived transfer kernel comprises an implicit filtering via a scale-dependent gain and phase—this kernel captures the coherent portion in the prediction. An additional refinement of the model embodies a relative shift between the stochastically estimated scales in the prediction and the modulation envelope of the universal small-scales. Predictions over a three-decade span of Reynolds numbers, $Re_\tau \sim \mathcal{O}(10^3)$ to $\mathcal{O}(10^6)$, highlight promising applications of the refined model to high-Reynolds-number flows, in which coherent scales become the primary contributor to the fluctuating energy.

11:45AM H32.00006 Scaling properties of the mean wall-normal velocity in the zero pressure gradient boundary layer

TIE WEI, New Mexico Institute of Mining and Technology, JOSEPH KLEWICKI, University of Melbourne — The scaling properties of the mean wall-normal velocity, $V(x, y)$, in zero-pressure-gradient laminar and turbulent boundary layer flows is investigated using numerical simulation data, physical experiment data, and integral analyses of governing equations. The maximum mean wall-normal velocity, V_∞ , and the boundary layer thickness, δ , are evidenced to be the proper scaling for V over most if not the entire boundary layer. This is different from the behavior of the mean streamwise velocity (U) or the turbulent shear stress ($T = -\rho\langle uv \rangle$), which depend on different characteristic length scales in the regions near to and away from the surface. Insights pertaining to this are further surmised from an analytical relationship for the ratio of the displacement to momentum thickness, i.e., shape factor, H . Integral analyses using the continuity and mean momentum equation show that $(U_\infty V_\infty)/u_\tau^2 = H$, where u_τ is the friction velocity. Both the laminar similarity solution and DNS data in post-transitional flows convincingly support this relation. Over the transitional regime, sufficiently high quality data is still lacking to check if this relation remains valid.

11:58AM H32.00007 Dissipation scaling in constant-pressure turbulent boundary layers¹

JOVAN NEDIC, STAVROS TAVOURARIS, University of Ottawa, IVAN MARUSIC, University of Melbourne — We use previous direct numerical simulations and experimental data to investigate the streamwise and wall-normal evolution of the dissipation parameter C_ε (namely the dissipation rate scaled by appropriate powers of the local turbulent kinetic energy and integral length scale) in the outer region of spatially evolving turbulent boundary layers. For $Re_\theta \geq 10,000$, C_ε is essentially constant in the streamwise direction, but varies measurably in the wall-normal direction. For lower Re_θ , however, C_ε changes in both directions. The constancy of C_ε is a central assumption of turbulence models based on the eddy viscosity concept and so they would inadequately represent wall bounded flows as they evolve spatially, a scenario that is common in engineering and atmospheric science applications. Accounting for the dependence of C_ε on the local Re_λ provides a means for possibly improving such models.

¹Funding provided by the Natural Sciences and Engineering Research Council of Canada (NSERC)

12:11PM H32.00008 New multi-scale causality analysis of streak-roll interactions in wall-bounded turbulence¹, X. SAN LIANG, Nanjing Institute of Meteorology, ADRIAN LOZANO-DURAN, Center for Turbulence Research, Stanford University — An important observation in nonlinear dynamical systems in general, and turbulent flows in particular, is that, as time goes on, the correlation between two events may be lost and re-emerge. How the causality evolves between them is hence of particular interest. Using a newly developed rigorous causality analysis based on the information transfer, we have examined the causality evolution between the streaks and rolls within the wall-bounded turbulence of a channel flow model with doubly periodic boundaries. The streaks are represented with the low principal component modes of the streamwise velocity U , while the rolls are reflected in the spanwise velocity W and vertical velocity V . It is found that the causal relation mainly occurs between the lower U modes, i.e., the streak structures, and V and W . For U and V , the causality is almost one-way, i.e., from the streaks to V , and that from the first two U modes (domain modes hereafter) dominates. In contrast, no causal relation has been identified between the domain modes and W . The $W-U$ causality is between W and the non-domain lower U modes, which are mutually causal, though the influence from the latter to the former dominates.

¹Funded by the Center for Turbulence Research Summer Program

12:24PM H32.00009 Statistics of passive scalar released from a point source in a turbulent boundary layer, KAPIL CHAUHAN, MURALI KRISHNA TALLURU, The University of Sydney, JIMMY PHILIP, The University of Melbourne — Measurements in a turbulent boundary layer are performed to document the statistics of a passive scalar when released from a point source in the logarithmic region. The nominal Reynolds number is $Re_\tau = \delta U_\tau / \nu \approx 8500$, where $\delta \approx 0.36$ m is the boundary layer thickness, U_τ is the skin-friction velocity and ν is the kinematic viscosity. Simultaneous single-point measurements are performed using a combination of hot-wire and photo-ionisation detector traversing in the wall-normal direction. The tracer gas (1.5% iso-butylene) is released at a streamwise distance, $s_x/\delta = 1$, upstream of the test location with the exit velocity matched to the local mean velocity at the source height. Preliminary results adhere to the known reflected Gaussian behaviour for the mean and variance profiles of scalar fluctuations. Also, we find support for the exponential probability distribution of scalar at $z = s_z$, which is extended to other wall-normal locations. Further, results on the interaction between large-scale velocity fluctuations and scalar fluctuations will be discussed.

12:37PM H32.00010 Scaling analysis of the mean and variariance of temperature in a developing thermal boundary layer¹, CLAYTON BYERS, MARCUS HULTMARK, Princeton University — A developing thermal boundary layer in a turbulent boundary layer is investigated both theoretically and experimentally. A scaling analysis of the mean temperature field and temperature variance, $\frac{1}{2}\overline{\theta^2}$, is developed by utilizing the Asymptotic Invariance Principle developed by George and Castillo (1997), including the possible effects of the Reynolds and Prandtl number. The derived solution for the inner and outer scaling is then used to develop a heat transfer law for the wall heat flux, q_w . The condition of constant wall temperature is utilized, with an analysis of the temperature field treated as a passive scalar through ensuring the temperature differences remain small. Data collection is performed with a nanoscale temperature sensor, providing an improvement to performance over previous cold wire data acquisition.

¹Supported by NSF (CBET-1510100 program manager Dimitrios Papavassiliou) and ONR (N00014-12-1-0875 and N00014-12-1-0962 program manager Ki-Han Kim).

Monday, November 21, 2016 10:40AM - 12:50PM –
Session H33 Boundary Layers: General Topics Oregon Ballroom 202 - Brian Elbing, Oklahoma State University

10:40AM H33.00001 Reexamination of the Classical View of how Drag-Reducing Polymer Solutions Modify the Mean Velocity Profile: Baseline Results¹, YASAMAN FARSIANI, JACQUELYNE BAADE, BRIAN ELBING, Oklahoma State University — Recent numerical and experimental data have shown that the classical view of how drag-reducing polymer solutions modify the mean turbulent velocity profile is incorrect. The classical view is that the log-region is unmodified from the traditional law-of-the-wall for Newtonian fluids, though shifted outward. Thus the current study reexamines the modified velocity distribution and its dependence on flow and polymer properties. Based on previous work it is expected that the behavior will depend on the Reynolds number, Weissenberg number, ratio of solvent viscosity to the zero-shear viscosity, and the ratio between the coiled and fully extended polymer chain lengths. The long-term objective for this study includes a parametric study to assess the velocity profile sensitivity to each of these parameters. This study will be performed using a custom design water tunnel, which has a test section that is 1 m long with a 15.2 cm square cross section and a nominal speed range of 1 to 10 m/s. The current presentation focuses on baseline (non-polymeric) measurements of the velocity distribution using PIV, which will be used for comparison of the polymer modified results. Preliminary polymeric results will also be presented.

¹This work was supported by NSF Grant 1604978.

10:53AM H33.00002 Scaling properties and turbulence modulation of flows with variable density and viscosity¹, RENE PECNIK, ASHISH PATEL, JURRIAAN PEETERS, BENDIKS JAN BOERSMA, Delft University of Technology — In order to identify the effect of variable density and viscosity on turbulence, we have performed DNS of canonical channel flows. The channel walls are isothermal and the flow is heated volumetrically to obtain gradients in temperature and thus in density and viscosity. Several constitutive relations for density ρ and viscosity μ as a function of temperature are used to create a database. We

parametrize the influence of density and viscosity in terms of gradients in semi-local Reynolds number $Re_\tau^* = \frac{\sqrt{\rho/\bar{\rho}_w}}{\mu/\bar{\mu}_w} Re_\tau$ (bar denotes Reynolds averaging, subscript w denotes wall value and Re_τ is the friction Reynolds number). The dominant factors that influence the turbulence are then attributed first to changes in viscous length scales and second to structural changes in turbulence. While the change in viscous length scales is accounted for by the semi-local scaling, structural changes remain for cases with gradients in Re_τ^* . Additionally, budgets of streamwise and spanwise vorticity equations are studied and the role of terms that are not accounted for by the semi-local framework, e.g. baroclinic torque, are also discussed.

¹Part of the work has been supported by the Center for Turbulence Research during the summer program 2016.

11:06AM H33.00003 PIV Measurements of Turbulent Pipe Flow with Drag-Reducing Megasupramolecules¹, DAVID HUYNH, RYAN McMULLEN, BEVERLEY MCKEON, REDMOND LHOТА, MING-HSIN WEI, JULIA KORNFIELD, Caltech — Toms (1948) was the first to observe that dissolving small amounts of high-molecular weight (HMW) polymers into a liquid can drastically reduce turbulent drag. Ever since, studying polymers in turbulence has been of great fundamental interest, as it can potentially provide insight into the self-sustaining mechanisms of wall turbulence. HMW polymers commonly employed for drag-reduction studies are plagued by chain scission due to the high shear rates accompanying turbulent flow at practical Reynolds numbers (Re); this shear degradation reduces the length of the polymer molecules, diminishing their effectiveness for drag-reduction. However, Wei et al. (2015) have recently developed “megasupramolecules” that perform comparably to traditional HMW polymers and circumvent the shear degradation problem by using end-associating polymers that can break and reassociate reversibly. Particle image velocimetry is used in specialized turbulent pipe flow experiments in the range $Re \approx 7.5 \times 10^4 - 1.2 \times 10^5$ to investigate and compare the drag and turbulence characteristics of the (Newtonian) baseline, traditional HMW polymer solutions, and megrasupramolecules.

¹The support of The Dow Corporation is gratefully acknowledged

11:19AM H33.00004 Stability analysis of two immiscible fluids in a shear driven flow: a DNS study¹, EDGARDO J. GARCIA CARTAGENA, The University of Texas at Dallas, MATTEO BERNARDINI, Sapienza University of Rome, ISNARDO ARENAS, The University of Texas at Dallas, ALIREZA MOHAMMADI, Princeton University, G. VALERIO IUNGO, The University of Texas at Dallas, ALEXANDER J. SMITS, Princeton University, STEFANO LEONARDI, The University of Texas at Dallas — Numerical studies of the flow over either super hydrophobic surfaces or liquid infused surfaces have shown that a large drag reduction (>10%) can be obtained if the flow remains in the Cassie state, thus stability of the interface plays a crucial role to achieve drag reduction. Direct Numerical Simulations of two immiscible fluids have been performed to assess how the stability of the interface depends on the viscosity ratio, thickness and Reynolds number of the two-layer flow. The flow is driven by the motion of one plate at constant velocity while the other plate is at rest. A finite difference code, based on a Runge-Kutta and fractional step method, has been combined to a level set method for tracking the interface between the two fluids. Results agree well with the linear theory until the nonlinear saturation. Once the fluctuations become large, a halving of the wavelength in the streamwise direction is observed for the least stable mode. The interaction between Tollmien-Schlichting waves and interfacial instabilities will be discussed at the meeting.

¹This work was supported under ONR MURI Grants N00014-12-0875 and N00014-12-1-0962, Program Manager Dr. Ki-Han Kim. Numerical simulations were performed on the Texas Advanced Computer Center

11:32AM H33.00005 Turbulent drag reduction over liquid infused surfaces¹, ISNARDO ARENAS, The University of Texas at Dallas, MATTEO BERNARDINI, Universita' di Roma La Sapienza, STEFANO LEONARDI, The University of Texas at Dallas — Numerical Simulations of two superposed fluids in a turbulent channel with a textured surface made of either longitudinal square bars or staggered cubes have been performed. The viscosity of the fluid inside the substrate is ten times smaller than that of the main stream ($m = \mu_1/\mu_2 = 0.1$ where the subscripts 1 and 2 indicate the fluid in the cavities and the overlying fluid respectively). The interface between the two fluids can move due to the turbulent pressure fluctuations and it is modeled with a Level Set Approach. Two cases are compared: $We = 0$, implying an interface sustained by the surface tension which can slip only in the horizontal direction, and $We = \infty$ where the interface can be displaced vertically and deform subject to wall normal stress. The textured surface made of staggered cubes is the most sensitive to the value of the surface tension, providing a drag reduction ranging between 15 – 30% for $We = 0$ and approximately 40% drag increase when $We = \infty$. On the other hand, longitudinal square bars, even with $We = \infty$ present a drag smaller than that over a smooth wall.

¹Numerical 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

11:45AM H33.00006 Thermal transport due to buoyant flow past a vertical, heated superhydrophobic surface with uniform stream-wise slip¹, MATTHEW SEARLE, DANIEL MAYNES, JULIE CROCKETT, Brigham Young University — An analytical investigation of thermal transport due to a steady, laminar, buoyancy-driven flow past a vertical superhydrophobic(SHPo) surface was performed. The surface temperature was constant and uniform and exceeded the temperature of the surrounding liquid. Uniform stream-wise hydrodynamic slip and temperature jump are imposed at the wall to model the SHPo surface. Applying an integral analysis within the boundary layer results in a system of differential equations which are solved numerically to obtain boundary layer thickness, maximum velocity in the profile, and local and average values of both the friction coefficient and the Nusselt number. The classical smooth hydrophobic scenario with no-slip and no temperature jump showed excellent agreement with previous analysis of the same problem. The influence of varying temperature jump length on the local and average values of the friction coefficient and the Nusselt number was obtained for Rayleigh number ranging from 10^4 to 10^9 and Prandtl number ranging from 2 to 11. Local and average Nusselt numbers decrease dramatically, concomitant with a decrease in the maximum fluid velocity, as the temperature jump length increases.

¹National Science Foundation(NSF) Grant No. CBET-1235881

11:58AM H33.00007 Two-Phase Flow Hydrodynamics in Superhydrophobic Channels, KIMBERLY STEVENS, JULIE CROCKETT, DANIEL MAYNES, BRIAN IVERSON, Brigham Young Univ - Provo — Superhydrophobic surfaces have been shown to reduce drag in single-phase channel flow; however, little work has been done to characterize the drag reduction found in two-phase channel flow. Adiabatic, air-water mixtures were used to gain insight into the effect of hydrophobicity on two-phase flows and the hydrodynamics which might be present in flow condensation. Pressure drop in a parallel plate channel with one superhydrophobic wall (cross-section 0.5 x 10 mm) and a transparent hydrophilic wall were explored. Data for air/water mixtures with superficial Reynolds numbers from 20-215 and 50-210, respectively, were obtained for superhydrophobic surfaces with three different cavity fractions. Agreement between experimentally obtained two-phase pressure drops and correlations in the literature for conventional smooth control surfaces was better than 20 percent. The reduction in pressure drop for channels with a single superhydrophobic wall were found to be more significant than that for single phase flow. The effect of cavity fraction on drag reduction was within experimental error.

12:11PM H33.00008 Magnetic Resonance Velocimetry Measurements of a Three-Dimensional Disturbance in a Laminar Boundary Layer, KE ZHANG, Xi'an Jiaotong University, AHMED NAGUIB, Michigan State University, TIM MICHAELIS, DANIEL FREUDENHAMMER, TU Darmstadt, SVEN GRUNDMANN, University of Rostock — 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 the present work, MRV is employed to provide boundary-layer-resolved measurements of a 3D disturbance created by a roughness element in a laminar boundary layer. Three-component mean-velocity-field data are captured over a volume of approximately $100 \times 100 \times 250 \text{ mm}^3$ with a voxel size of 1 mm^3 . The reported measurements are similar to those presented at the APS-DFD meeting last year, after improvement of the acrylic test section of the water-flow loop used in the experiments. The roughness element is mounted through the test-section's side wall where the boundary layer Reynolds number is 162, based on displacement thickness. The experiments are designed to investigate the effect of the roughness element's shape (cylindrical versus hemi-spherical) and height (for the cylindrical element). Details of the 3D velocity and vorticity fields in the disturbed boundary layer will be presented.

12:24PM H33.00009 Study of laminar boundary layer instability noise study on a controlled diffusion airfoil, PRATEEK JAISWAL, student, MARLENE SANJOSE, Researcher, STEPHANE MOREAU, Professor — Detailed experimental study has been carried out on a Controlled Diffusion (CD) airfoil at 5° angle of attack and at chord based Reynolds number of 1.5×10^5 . All the measurements were done in an open-jet anechoic wind tunnel. The airfoil mock-up is held between two side plates, to keep the flow two-dimensional. PIV measurements have been performed in the wake and on the boundary layer of the airfoil. Pressure sensor probes on the airfoil were used to detect mean airfoil loading and remote microphone probes were used to measure unsteady pressure fluctuations on the surface of the airfoil. Furthermore the far field acoustic pressure was measured using an $\frac{1}{2}$ inch ICP microphone. The results confirm very later transition of a laminar boundary layer to a turbulent boundary layer on the suction side of the airfoil. The process of transition of laminar to turbulent boundary layer comprises of turbulent reattachment of a separated shear layer. The pressure side of the boundary layer is found to be laminar and stable. Therefore tonal noise generated is attributed to events on suction side of the airfoil. The flow transition and emission of tones are further investigated in detail thanks to the complementary DNS study.

12:37PM H33.00010 Flow Field Classification Using Critical Point Matching¹, PAUL S. KRUEGER, SHEILA WILLIAMS, MICHAEL HAHSLER, ELI V. OLINICK, SMU — Classification of flow fields according to topological similarities can help reveal features of the flow generation and evolution for bluff body flows, and characterize different swimming maneuvers in aquatic locomotion, to name a few. Rigorous classification can be challenging, however, especially when complex flows are distorted by measurement uncertainties or variable flow generating conditions. The present work uses critical points of the velocity field to characterize the global flow topology. Flow fields are compared by finding a best match of critical points in two flow fields based on topological and location characteristics of the critical points together with general point set distance measures. The similarity between the flow fields is quantified based on the matched critical points. Applying clustering algorithms to a set of flow fields with quantified similarity can then be used to group flows with similar characteristics. This approach has been applied to generic 2D flow fields constructed using potential flow results and is able to correctly identify similar flow fields even after large distortions (up to 20% of the vortex separation) have been applied to the flows.

¹Support of NSF grant nos. 1115139 and 1557698, and the Lyle School of Engineering is gratefully acknowledged.

Monday, November 21, 2016 10:40AM - 12:50PM –
Session H34 Turbulence: LES Models and Methods Oregon Ballroom 203 - Mahdi Abkar, Stanford University

10:40AM H34.00001 Modeling and large-eddy simulation (LES) of a turbulent boundary layer over linearly-varying surface roughness¹, A. SRIDHAR, D. I. PULLIN, California Institute of Technology, W. CHENG, King Abdullah University of Science and Technology — An empirical model is presented, after Rotta (1962), that describes the development of a fully-developed turbulent boundary layer in the presence of surface roughness with nominal roughness length-scale k_s that varies with stream-wise distance x . For $Re_x = U_e(x)x/\nu$ large, use is made of the log-wake model of the local turbulent mean-velocity profile that contains the Hama roughness correction $\Delta U^+(k_s^+)$ for the asymptotic, fully rough regime. It is shown that the skin friction coefficient C_f is constant in x only for $k_s = \alpha x$, where α is a dimensionless number. For $U_e(x) = Ax^m$, this then gives a two-parameter (α, m) family of solutions for boundary-layer flows that are self similar in the variable $z/(\alpha x)$ where z is the wall-normal co-ordinate. Trends observed in this model are supported by wall-modeled LES of the zero-pressure-gradient turbulent boundary layer ($m = 0$) at very large Re_x . It is argued that the present results suggest that, in the sense that C_f is spatially constant and independent of Re_x , this class of flows can be interpreted as providing the asymptotically-rough equivalent of Moody-like diagrams for boundary layers in the presence of small-scale roughness.

¹Supported partially by KAUST OCRM Award No. URF/1/1394-01 and partially by NSF award CBET 1235605

10:53AM H34.00002 ABSTRACT WITHDRAWN –

11:06AM H34.00003 Subgrid-scale models for large-eddy simulation of rotating turbulent flows¹, MAURITS SILVIS, University of Groningen, XAVIER TRIAS, Technical University of Catalonia, Spain, MAHDI ABKAR, HYUNJI JANE BAE, ADRIAN LOZANO-DURAN, Stanford University, ROEL VERSTAPPEN, University of Groningen — This paper discusses subgrid models for large-eddy simulation of anisotropic flows using anisotropic grids. In particular, we are looking into ways to model not only the subgrid dissipation, but also transport processes, since these are expected to play an important role in rotating turbulent flows. We therefore consider subgrid-scale models of the form $\tau = -2\nu_t S + \mu_t(S\Omega - \Omega S)$, where the eddy-viscosity ν_t is given by the minimum-dissipation model, μ_t represents a transport coefficient; S is the symmetric part of the velocity gradient and Ω the skew-symmetric part. To incorporate the effect of mesh anisotropy the filter length is taken in such a way that it minimizes the difference between the turbulent stress in physical and computational space, where the physical space is covered by an anisotropic mesh and the computational space is isotropic. The resulting model is successfully tested for rotating homogeneous isotropic turbulence and rotating plane-channel flows.

¹The research was largely carried out during the CTR SP 2016. M.S. and R.V. acknowledge the financial support to attend this Summer Program

11:19AM H34.00004 Large-eddy simulations of viscoelastic isotropic turbulence with the FENE-P fluid, FERNANDO T. PINHO, Faculdade de Engenharia da Universidade do Porto (FEUP), PEDRO O. FERREIRA, CARLOS B. DA SILVA, Inst Superior Técnico (IST), IDMEC/FEUP COLLABORATION — A new subgrid-scale (SGS) model developed for large-eddy simulations (LES) of dilute polymer solutions described by the Finitely Extensible Nonlinear Elastic constitutive equation closed with the Peterlin approximation (FENE-P), is presented. The filtered conformation tensor evolution equation uses the *self-similarity* of the polymer stretching terms, and the *global equilibrium* of the trace of the conformation tensor, while the SGS stresses are modelled with the classical Smagorinsky model. The new closure is assessed in direct numerical simulations (DNS) of forced isotropic turbulence using classical *a-priori* tests, and in *a-posteriori* (LES) showing excellent agreement with all the exact (filtered DNS) results.

11:32AM H34.00005 A minimum dissipation scalar transport model for large-eddy simulation of turbulent flows, MAHDI ABKAR, HYUN J. BAE, PARVIZ MOIN, Center for Turbulence Research, Stanford University, Stanford, California 94305, USA — Minimum-dissipation models are a simple alternative to the Smagorinsky-type approaches to parameterize the sub-filter scale turbulent fluxes in large-eddy simulation. A recently derived minimum-dissipation model for sub-filter stress tensor is the AMD model (Rozema et al., Phys. Fluids, 2015) and has many desirable properties. It is more cost effective than the dynamic Smagorinsky model, it appropriately switches off in laminar and transitional flows, and it is consistent with the theoretic sub-filter stress tensor on both isotropic and anisotropic grids. In this study, an extension of this approach to modeling the sub-filter scalar flux is proposed. The performance of the AMD model is tested in the simulation of a high Reynolds number, rough wall, boundary layer flow with a constant and uniform surface scalar flux. The simulation results obtained from the AMD model show good agreement with well-established empirical correlations and theoretical predictions of the resolved flow statistics. In particular, the AMD model is capable to accurately predict the expected surface-layer similarity profiles and power spectra for both velocity and scalar concentration.

11:45AM H34.00006 Towards a transport model for epistemic UQ in RANS closures, WOUTER EDELING, GIANLUCA IACCARINO, Stanford Univ — Due to their computational efficiency, Reynold-Averaged Navier-Stokes (RANS) turbulence models remain a vital tool for modeling turbulent flows. However, it is well known that RANS predictions are locally corrupted by epistemic model-form uncertainty. Whereas some Uncertainty Quantification (UQ) approaches attempt to quantify this uncertainty by considering the model coefficients as random variables, we directly perturb the Reynold-stress tensor at locations in the flow domain where the modeling assumptions are likely to be invalid. Inferring the perturbations on a point-by-point basis would lead to a high-dimensional problem. To reduce the dimensionality, we propose separate model equations based on the transport of linear invariants of the anisotropy tensor. This provides us with a low-dimensional UQ framework where the invariant transport model decides on the magnitude and direction of the perturbations. Where the perturbations are small, the RANS result is recovered. Using traditional turbulence modeling practices we derive weak realizability constraints, and we will rely on Bayesian inference to calibrate the model on high-fidelity data. We will demonstrate our framework on a number of canonical flow problems where RANS models are prone to failure.

11:58AM H34.00007 Three-dimensional Diffusive Strip Method, DANIEL MARTINEZ-RUIZ, PATRICE MEUNIER, LAURENT DUCHEMIN, EMMANUEL VILLERMAUX, Aix Marseille Université, CNRS, Centrale Marseille, IRPHE UMR 7342, 13384 Marseille, France — The Diffusive Strip Method (DSM) is a near-exact numerical method developed for mixing computations at large Peclet number in two-dimensions (*J. Fluid Mech.* **662**, (2010)). The method consists in following stretched material lines to compute a-posteriori the resulting scalar field is extended here to three-dimensional flows, following surfaces. We describe its 3D peculiarities, and show how it applies to a simple Taylor-Couette configuration with non-rotating boundary conditions at the top end, bottom and outer cylinder. This flow produces an elaborate, although controlled, steady 3D flow which relies on the Ekman pumping arising from the rotation of the inner cylinder is both studied experimentally, and numerically modeled. A recurrent two-cells structure appears formed by stream tubes shaped as nested tori. A scalar blob in the flow experiences a Lagrangian oscillating dynamics with stretchings and compressions, driving the mixing process, and yielding both rapidly-mixed and nearly pure-diffusive regions. A triangulated-surface method is developed to calculate the blob elongation and scalar concentration PDFs through a single variable computation along the advected blob surface, capturing the rich evolution observed in the experiments.

12:11PM H34.00008 Anomalous Transport Theory, MEHDI SAMIEE¹, MOHSEN ZAYERNOURI², Michigan State Univ — We propose a generalizing fractional order transport model of advection-diffusion kind with fractional time- and space-derivatives, governing the evolution of passive scalar turbulence. This approach allows one to incorporate the nonlocal and memory effects in the underlying anomalous diffusion i.e., *sub-to-standard* diffusion to model the trapping of particles inside the eddied, and *super-diffusion* associated with the sudden jumps of particles from one coherent region to another. For this nonlocal model, we develop a high order numerical (spectral) method in addition to a fast solver, examined in the context of some canonical problems.

”A Fractional PDE Approach to Turbulent Mixing; Part II: Numerical Simulation”

”A Fractional PDE Approach to Turbulent Mixing; Part I: an Anomalous Transport Theory”

12:24PM H34.00009 A Fractional PDE Approach to Turbulent Mixing; Part II: Numerical Simulation, MEHDI SAMIEE¹, MOHSEN ZAYERNOURI², Michigan State Univ — We propose a generalizing fractional order transport model of advection-diffusion kind with fractional time- and space-derivatives, governing the evolution of passive scalar turbulence. This approach allows one to incorporate the nonlocal and memory effects in the underlying anomalous diffusion i.e., *sub-to-standard* diffusion to model the trapping of particles inside the eddied, and *super-diffusion* associated with the sudden jumps of particles from one coherent region to another. For this nonlocal model, we develop a high order numerical (spectral) method in addition to a fast solver, examined in the context of some canonical problems.

¹PhD student, Department of Mechanical Engineering, & Department Computational Mathematics, Science, and Engineering

²Assistant Professor, Department Computational Mathematics, Science, and Engineering, & Department of Mechanical Engineering

12:37PM H34.00010 Quantum speed-up for turbulent mixing simulation¹ , GUANGLEI XU,

ANDREW DALEY, Department of Physics and SUPA, University of Strathclyde, PEYMAN GIVI, Swanson School of Engineering, University of Pittsburgh, ROLANDO SOMMA, Theoretical Division, Los Alamos National Laboratory — Quantum computing techniques have the potential in the future to generate revolutionary advances in many types of computation. The necessary hardware is under rapid development, making it an opportune time to identify possible specific applications across a range of fields, and properly identify the potential of this new paradigm of computing. Turbulent mixing simulation is important in a variety of fields, and is typically accomplished by Monte Carlo methods. To reach high precision in estimating parameters often requires vast computational resources. We have developed a quantum algorithm for turbulent mixing simulation that provides a quadratic speed-up over Monte Carlo methods in terms of number of repetitions needed to achieve designated accuracy. Taking the example of binary scalar mixing process described by a coalescence/dispersion model, we demonstrate the advantages of our quantum algorithm by illustrating comparisons of statistical error scaling to repetition number between Monte Carlo method and quantum algorithm. This is an important starting point to further understand how quantum algorithms can be directly applied in fluid dynamics, and to estimate the timescales on which quantum hardware will have useful applications in this area of science.

¹This work was supported by AFOSR grant FA9550-12-1-0057.

Monday, November 21, 2016 10:40AM - 12:50PM –

Session H35 Turbulence: Theory Oregon Ballroom 204 - Laurent Chevillard, Laboratoire de Physique de l'ENS Lyon

10:40AM H35.00001 A dissipative random velocity field for fully developed fluid

turbulence , LAURENT CHEVILLARD, Laboratoire de Physique de l'ENS Lyon, RODRIGO PEREIRA, Instituto de Fisica, Universidade Federal do Rio de Janeiro, C.P. 68528, 21945-970, Rio de Janeiro, RJ, Brazil, CHRISTOPHE GARBAN, Institut Camille Jordan, Universite Lyon 1, Lyon, France — We investigate the statistical properties, based on numerical simulations and analytical calculations, of a recently proposed stochastic model for the velocity field of an incompressible, homogeneous, isotropic and fully developed turbulent flow. A key step in the construction of this model is the introduction of some aspects of the vorticity stretching mechanism that governs the dynamics of fluid particles along their trajectory. An additional further phenomenological step aimed at including the long range correlated nature of turbulence makes this model depending on a single free parameter that can be estimated from experimental measurements. We confirm the realism of the model regarding the geometry of the velocity gradient tensor, the power-law behaviour of the moments of velocity increments, including the intermittent corrections, and the existence of energy transfers across scales. We quantify the dependence of these basic properties of turbulent flows on the free parameter and derive analytically the spectrum of exponents of the structure functions in a simplified non dissipative case. A perturbative expansion shows that energy transfers indeed take place, justifying the dissipative nature of this random field.

10:53AM H35.00002 Dissipation in non-equilibrium turbulence , WOUTER BOS, CNRS - LMFA -

Ecole Centrale de Lyon, ROBERT RUBINSTEIN, None — For about a decade, experimental and numerical studies have reported on the existence of an anomalous behaviour of the viscous dissipation rate in unsteady turbulence (see for instance Vassilicos, Annu. Rev. Fluid Mech. 2015). It appears that the short-time transient dynamics can be described by a universal power law, incompatible with Taylor's 1935 dissipation rate estimate. We show that these results can be explained using a non-equilibrium energy distribution, obtained from a low-frequency perturbative expansion of simple spectral closure. The resulting description is fairly simple. In particular, during the transient, according to the predictions, the normalized dissipation rate C_ϵ evolves as a function of the Taylor-scale Reynolds number R_λ following the relation $C_\epsilon \sim R_\lambda^{-15/14}$, in close agreement with experimental and numerical observations.

11:06AM H35.00003 A Lagrangian fluctuation-dissipation relation for scalar turbulence , THEODORE DRIVAS, GREGORY EYINK, The Johns Hopkins University — An exact relation is derived between the dissipation of scalar fluctuations and the variance of the scalar inputs (due to initial scalar values, scalar sources, and boundary fluxes) as those are sampled by stochastic Lagrangian trajectories. Previous work on the Kraichnan (1968) model of turbulent scalar advection has shown that anomalous scalar dissipation, non-vanishing in the limit of vanishing viscosity and diffusivity, is in that model due to Lagrangian spontaneous stochasticity, or non-determinism of the Lagrangian particle trajectories in the limit. We here extend this result to scalars advected by any incompressible velocity field. For fluid flows in domains without walls (e.g. periodic boxes) and for insulating/impermeable walls with zero scalar fluxes, we prove that anomalous scalar dissipation and spontaneous stochasticity are completely equivalent. For flows with imposed scalar values or non-vanishing scalar fluxes at the walls, spontaneous stochasticity still implies anomalous scalar dissipation but simple examples show that a distinct mechanism of non-vanishing dissipation can be thin scalar boundary layers near the walls. As an example, we consider turbulent Rayleigh-Benard convection. We here obtain an exact relation between steady-state thermal dissipation and the time for diffusive tracer particles released at the top or bottom wall to mix to their final uniform value near those walls. We show that an "ultimate regime" of turbulent convection as predicted by Kraichnan (1962) will occur at high Rayleigh numbers, unless this near-wall mixing time is asymptotically much longer than the large-scale circulation time.

11:19AM H35.00004 A new approach to Lagrangian investigations of isotropic

turbulence¹ , MANUEL BARJONA, CARLOS B. DA SILVA, Inst Superior Tecnico (IST), IDMEC TEAM — A new numerical approach is used in conjunction with direct numerical simulations (DNS) of statistically stationary (forced) isotropic turbulence to investigate the high Reynolds number scaling properties of turbulence characteristics in a Lagrangian frame. The new method provides an alternative route to the determination of the classical Lagrangian turbulence quantities, such as the second order Lagrangian velocity structure function and two point particle separation, at a much higher Reynolds number than as obtained in previous numerical simulations, and displays excellent agreement with the classical theoretical predictions and existing numerical simulations and experimental data.

¹The authors acknowledge the Laboratory for Advanced Computing at University of Coimbra for providing HPC, computing, consulting resources that have contributed to the research results reported within this paper. URL <http://www.lca.uc.pt>

11:32AM H35.00005 Multi-Scale Analysis of Lagrangian Properties of Turbulence , MICHAEL WILCZEK, CRISTIAN LALESCU, Max Planck Institute for Dynamics and Self-Organization — Turbulence is a multi-scale problem in space and time with a broad range of strongly interacting degrees of freedom. Lagrangian tracer particles advected with the flow sample this spatio-temporal complexity. This naturally leads to the question of how Lagrangian properties are affected by the scales of turbulence. We attempt to answer this question numerically and theoretically adopting a coarse-graining approach. In an extensive DNS (direct numerical simulation) study, we track tracer particles advected by spatially coarse-grained velocity fields. This allows to distinguish the impact of large-scale sweeping effects and small-scale intermittency on Lagrangian aspects of turbulence. In this presentation we will present results on Lagrangian particle dispersion and velocity fluctuations for various coarse-graining scales. The results will furthermore be discussed in the context of Eulerian-Lagrangian bridging relations.

11:45AM H35.00006 Turbulence screening suppresses long-range pressure contributions, DIMITAR VLAYKOV, MICHAEL WILCZEK, Max Planck Institute for Dynamics and Self-Organization — The complexity in turbulence can be seen to stem from the non-linearity and non-locality of the governing Navier-Stokes equations. The non-linearity gives rise to structures on many scales with varying topologies – from strain sheets to vortex tubes. In incompressible flows, these structures determine the pressure field through a Poisson relation. This in turn describes the non-locality of incompressible flows – formally the pressure at any point is determined by the competition of strain and enstrophy over the entire flow. We show how due to the relative compactness and close proximity of extreme strain and vortex regions a type of effective turbulence screening emerges. We characterize this effect in statistically stationary homogeneous and isotropic turbulence by considering the spatial (two-point) statistics of the velocity gradient fields. This clarifies the observation from both experiments and numerical simulations that a relatively small neighbourhood – comparable with the small turbulence scales, contains the large majority of the information about the pressure at a given point. We characterize the properties of this neighbourhood as a function of global flow parameters, like Reynolds number, as well as local flow properties, e.g. local topology and dissipation.

11:58AM H35.00007 Leith diffusion model for homogeneous anisotropic turbulence, ROBERT RUBINSTEIN, Computational Aerosciences Branch, NASA Langley Research Center, Hampton VA, TIMOTHY CLARK, Department of Mechanical Engineering, University of New Mexico, Albuquerque NM, SUSAN KURIEN, Theoretical Division, Los Alamos National Laboratory, Los Alamos NM — A new spectral closure model for homogeneous anisotropic turbulence is proposed. The systematic development begins by closing the third-order correlation describing nonlinear interactions by an anisotropic generalization of the Leith diffusion model for isotropic turbulence. The correlation tensor is then decomposed into a tensorially isotropic part, or directional anisotropy, and a trace-free remainder, or polarization anisotropy. The directional and polarization components are then decomposed using irreducible representations of the $SO(3)$ symmetry group. Under the ansatz that the decomposition is truncated at quadratic order, evolution equations are derived for the directional and polarization pieces of the correlation tensor. Numerical simulation of the model equations for a freely decaying anisotropic flow illustrate the non-trivial effects of spectral dependencies on the different return-to-isotropy rates of the directional and polarization contributions.

12:11PM H35.00008 A generalized four-fifth law for compressible turbulence¹, HUSSEIN ALUIE, University of Rochester and Laboratory for Laser Energetics — Kolmogorov's 4/5-th law is a celebrated exact result of incompressible turbulence, and is key to the formulation of his 1941 phenomenology. We will present its generalization to compressible turbulence.

¹Partial support was provided by NSF Grant OCE-1259794, US Department of Energy (US DOE) Grant DE-SC0014318, and the LANL LDRD program through project number 20150568ER.

12:24PM H35.00009 The structure of the extreme Lyapunov exponents in the inertial scales of turbulence¹, ALBERTO VELA-MARTIN, JAVIER JIMENEZ, Universidad Politecnica de Madrid — A fully reversible homogeneous isotropic turbulent system is constructed using inviscid LES to model energy fluxes in the far inertial range. Reversibility is exploited to efficiently calculate the highest/most unstable and lowest/most stable short-time Lyapunov exponents (STLE) of the system. When restricted to inertial modes, both extreme STLE have similar absolute value and inverse sign, suggesting the Hamiltonian nature of inertial dynamics. Their associated short-time Lyapunov vectors (STLV), which are complete flow fields that provide information on the perturbations to which the system is most/least sensitive, are found to be concentrated in small regions in physical space. The analysis of the structure of the STLV reveals that these small regions, where intense expansive and contractive events take place, are strongly dominated by the strain field of the flow. These regions are also characterized by a preferential alignment of the field of the STLV with the different eigenvectors of the strain tensor. However, no strong correlation of the STLV with the vorticity field is found. These results emphasize the active role of the strain in turbulence dynamics.

¹Funded by the ERC COTURB project

12:37PM H35.00010 Analytic prediction for planar turbulent boundary layers, ZHEN-SU SHE, XI CHEN, College of Engineering, Peking University — Analytic predictions of mean velocity profile (MVP) and streamwise (x) development of related integral quantities are presented for flows in channel and turbulent boundary layer (TBL), based on a symmetry analysis of eddy length and total stress. Specific predictions include the relations for momentum Reynolds number (Re_θ) with friction Re_τ and streamwise $Re_x:Re_\theta \approx 3.27Re_\tau$ and $Re_x/Re_\theta = 4.94[(\ln Re_\theta + 1.88)^2 + 1]$; the streamwise development of the friction velocity $u_\tau:U_e/u_\tau \approx 2.22 \ln Re_x + 2.86 - 3.83 \ln(\ln Re_x)$, and of the boundary layer thickness $\delta_e:x/\delta_e \approx 7.27 \ln Re_x - 5.18 - 12.52 \ln(\ln Re_x)$, which are fully validated by recent reliable data.

Monday, November 21, 2016 10:40AM - 12:50PM –
Session H36 Drops: Wetting and Moving Contact Line Effects Portland Ballroom 251 - Laurent Limat, Paris Diderot University

10:40AM H36.00001 Homogeneous deposition of particles on hydrogels by absorption, FRANÇOIS BOULOGNE, Paris Diderot University, FRANÇOIS INGREMEAU, Princeton University, JULIEN DERVAUX, LAURENT LIMAT, Paris Diderot University, HOWARD STONE, Princeton University — A drying drop containing solid particles, such as coffee, leaves a ring stain resulting from the accumulation of the particles near a contact line. In many industrial applications such as printing, coating or biological microtechnologies, these inhomogeneities must be avoided. To suppress the coffee stain effect, different strategies have been developed. In the present work, we propose to substitute the drying by absorption in hydrogels to extract the solvent of a colloidal drop. We study the deposition mechanisms of micrometer-sized particles on the surface of swelling hydrogels. To the best of our knowledge, we show for the first time that the particle deposition on these gels is homogeneous. Using fluorescence microscopy coupled with particle tracking techniques, we record the flow field inside the droplet and analyze the particle deposition mechanism. We rationalize our findings with a theoretical model for the absorption and the particle deposition dynamics that enables the measurement of the diffusion coefficient in the gels.

10:53AM H36.00002 A scale-dependent model for direct computation of dynamic contact lines, S. ZALESKI, Sorbonne Universites, UPMC Univ Paris 06, CNRS, UMR 7190, France, S. AFKHAMI, New Jersey Institute of Technology, A. GUION, J. BUONGIORNO, Massachusetts Institute of Technology — When using numerical schemes for the simulation of moving contact lines with the classical "no-slip" boundary condition, the numerical solutions become dependent on grid spacing. Numerical approaches that account for the slip of the contact line avoid this difficulty; a numerically feasible slip length however can often be much larger than the physically "true" one. Afkhami et al. [J. Comp. Phys., 228:5370–5389, 2009] addressed this issue, where they proposed a numerical model for the implementation of contact angle based on the mesh size that resulted in mesh independent solutions. Here we refine and apply their numerical observation by studying the problem of coating a plate withdrawn from a viscous liquid reservoir. We consider a partially wetting liquid and show that depending on the capillary number, either a stationary meniscus is formed or a liquid film is deposited on the substrate, known as the transition to a Landau–Levich–Derjaguin film. We derive an effective numerical boundary condition model for the computation of the transition capillary number. The model can be thought of as a large scale solution in an asymptotic matching procedure.

11:06AM H36.00003 Interfacial transport alone accounts for coffee-ring deposition, VAHID VANDADI, SAEED JAFARI KANG, University of Nevada, Reno, JAMES D. FELSKY, State University of New York at Buffalo, HASSAN MASOUD, University of Nevada, Reno — When a colloidal sessile droplet dries on a substrate, the suspended particles usually deposit on the surface in a ring-like pattern. The phenomenon is commonly known as the "coffee-ring" effect and it is widely believed to stem from the transport of solutes towards the pinned contact line by the evaporation-induced flow inside the drop. It is, therefore, assumed that the liquid-gas interface does not play an active role in shaping the deposition pattern. Here, we propose an alternative mechanism for the coffee-ring deposition, in which the particles first intersect the receding free surface and then are transported along the interface until they deposit at the edge. That the interface "captures" the solutes as the evaporation proceeds is supported by a Lagrangian tracking of particles advected by the flow field within the droplet. We model the interfacial adsorption and transport of particles by a one-dimensional advection-generation equation in a toroidal coordinate system and show that the theory adequately accounts for the coffee-ring effect. Using this model, we study the final deposition pattern on hydrophilic and hydrophobic surfaces under diffusive and uniform evaporation fluxes.

11:19AM H36.00004 Passive bloodstains: from an impact energy to a final dried pattern¹, FIONA SMITH, DAVID BRUTIN, Aix-Marseille University — Tracking down the origin of a blood droplet present on a crime scene has become of major importance in bloodstain pattern analysis. Passive bloodstains are not yet well understood. Accordingly the purpose of this research is to provide new tools to forensic investigators in the analysis of bloodstains arising from blood droplets dripping naturally. The study aims to understand the link between the final dried pattern of a passive bloodstain and its impact energy. Currently no such tool exists, and no correlation has yet been proven. This research was therefore focusing on a new parameter, the thicker outer rim observed on the dried final pattern. To do so, we created several passive bloodstains with different impact energies. A correlation was highlighted between the inner diameter, the maximum spreading diameter, the initial diameter of a blood droplet and its impact energy. This correlation shows how the drying mechanism of a blood droplet is influenced by its impact energy as it alters the red blood cells dispersion inside the droplet. The biological deposit and the final dried pattern are subsequently modified.

¹ANR funded project: D-Blood Project

11:32AM H36.00005 Active depinning of bacterial droplets: the collective surfing of Bacillus subtilis¹, MARC HENNES, JULIEN TAILLEUR, ADRIAN DAERR, Laboratoire MSC, University Paris Diderot (Paris 7) — How systems are endowed with migration capacity is a fascinating question with implications ranging from the design of novel active systems to the control of microbial populations. Bacteria, which can be found in a variety of environments, have developed among the richest set of locomotion mechanisms both at the microscopic and collective levels. Here, we uncover experimentally a new mode of collective bacterial motility in humid environment through the depinning of bacterial droplets. While capillary forces are notoriously enormous at the bacterial scale, even capable of pinning water droplets of millimetric size on inclined surfaces, we show that bacteria are able to harness a variety of mechanisms to unpin contact lines, hence inducing a collective sliding of the colony. Contrary to flagella-dependent migration modes like swarming we show that this much faster colony surfing still occurs in mutant strains of *Bacillus subtilis* lacking flagella. The diversity of mechanisms involved in the active unpinning seen in our experiments suggests that collective surfing should be a generic mode of migration of microorganisms in humid environments.

¹Bacttern Grant

11:45AM H36.00006 Sessile nanodroplets on elliptical patches of enhanced lyophilicity, IVAN DEVIC, University of Twente, GIUSEPPE SOLIGNO, MARJOLEIN DIJKSTRA, REN VAN ROIJ, Utrecht University, XUEHUA ZHANG, RMIT University, Melbourne, DETLEF LOHSE, University of Twente — We theoretically investigate how nanodroplets wet lyophilic elliptical patches in a lyophobic surrounding on a flat substrate. To do so, we minimize the interfacial energy of the nanodroplet using Surface evolver and Monte Carlo calculations, finding decent agreement between the two methods. We can observe four different wetting phases, which are controlled by the aspect ratio of the ellipse and the Young's contact angles. Of particular interest is the behavior of the contact angle of the nanodroplet along the contact line, with which we can explain why the wetting phase transitions occur. We find that the contact angle of the nanodroplet can only change along the rim of the elliptical patch, while the nanodroplet satisfy Young's equation once the contact line is either inside or outside of the patch. The contact line of the nanodroplet, depending on Young's angles and the aspect ratio of the elliptical patch, may be completely pinned to the rim of the elliptical patch, while for some cases we find that the nanodroplet starts to expand to lyophobic surrounding, although there still is lyophilic surrounding available to wet.

11:58AM H36.00007 Diffuse-interface modeling of three-phase interactions, PATRICK ANDERSON, Eindhoven Univ of Technology, JANG MIN PARK, School of Mechanical Engineering, Yeungnam University, 38541 Gyeongsan, Republic of Korea — In this work, a numerical model is developed to study the three-phase interactions which take place when two immiscible drops suspended in a third immiscible liquid are brought together. The diffuse-interface model coupled with the hydrodynamic equations are solved by a standard finite element method. Partial and complete engulfing between two immiscible drops are studied, and the effects of several parameters are discussed. In the partial-engulfing case, two stages of wetting and pulling are identified, which qualitatively agrees with experiment. In the complete-engulfing case, three stages of wetting and/or penetration, pulling and spreading are identified.

12:11PM H36.00008 Analysis of subphase gas/fluid effects on longitudinal flow over unidirectional superhydrophobic surfaces¹, DARREN CROWDY, Imperial College London — An analysis of the slip properties associated with longitudinal shear flow over a unidirectional superhydrophobic surface comprising periodic rectangular grooves of arbitrary geometry is described. We produce analytical expressions for the hydrodynamic slip lengths (in various perturbative limits) when the effects of a subphase viscous fluid occupying the grooves is taken into account.

¹EPSRC

12:24PM H36.00009 Singular effective slip length for longitudinal flow over a dense bubble mattress, ORY SCHNITZER, Department of Mathematics, Imperial College London — We derive accurate asymptotic expansions in the small-solid-fraction limit $\epsilon \ll 1$ for the effective slip length characterising unidirectional liquid flow over a 'bubble mattress' — a periodically grooved surface, with trapped bubbles protruding between solid ridges. The slip length diverges in this limit: inversely with $\sqrt{\epsilon}$ for contact angles θ near $\pi/2$, and logarithmically for $0 \leq \theta < \pi/2$. The analysis of the velocity field entails matching 'inner' expansions valid close to the solid segments with 'outer' expansions valid on the scale of the periodicity, where the protruding bubbles appear to touch. For θ close to $\pi/2$, the inner-region geometry is narrow and the analysis there resembles lubrication theory; for smaller contact angles the inner region is resolved using a Schwarz-Christoffel mapping. In both cases the outer problem is solved using a mapping from a degenerate curvilinear triangle to an auxiliary half plane. The asymptotic analysis explicitly illustrates the logarithmic-to-algebraic transition, and yields a uniformly valid approximation for the slip length for arbitrary contact angles $0 \leq \theta \leq \pi/2$. We demonstrate good agreement with a numerical solution (courtesy of Ms Elena Luca).

12:37PM H36.00010 Dewetting on microstructured substrates¹, TAEHONG KIM, WONJUNG KIM, Sogang Univ — A thin liquid film has an equilibrium thickness in such a way as to minimize the free energy. When a liquid film thickness is out of its equilibrium, the film seeks its equilibrium state, resulting in dynamics of liquid film, which are referred to as wetting and dewetting, depending on the flow direction. We here present a combined experimental and theoretical investigation of dewetting on a substrate with parallel microstructures. Our experiments show that residue may remain on the substrate after dewetting, and residue morphologies can be classified into three modes. Based on our experimental observations, we elucidate how the modes depend on the pattern morphology and contact angle, and develop a model for the contact line motion. Our results provide a basis for controlling the thickness film, which is important for many practical applications such as oil recovery, detergency, lithography, and cleaning.

¹This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP) (No.2015R1A2A2A04006181).

Monday, November 21, 2016 10:40AM - 12:50PM –
Session H37 Drops: Impacts with Fluid Surfaces Portland Ballroom 252 - Tadd Truscott, Utah State University

10:40AM H37.00001 The water entry of water, NATHAN SPEIRS, ZHAO PAN, Utah State University, JESSE BELDEN, Naval Undersea Warfare Center, TADD TRUSCOTT, Utah State University — Though water entry has been studied for over a century, there has been a disconnect between solid object water entry and research on liquid impacting on a liquid pool. In addition, few have studied multiple objects impacting a liquid bath sequentially. We show that the impact of multi-droplet streams and liquid jets on a liquid pool display similar behavior to solid body water entry. In particular, the cavities of both droplet streams and jets exhibit three types of cavity seal previously found for hydrophobic spheres at low Bond numbers. Additionally, low-frequency droplet streams create three novel cavity seal types, which can be predicted with a new non-dimensional frequency. The cavity depth for both droplet and jet impact is rationalized by an energy scaling analysis. Finally, we examine the similarities and differences in cavity dynamics for multi-droplet streams and continuous liquid jets.

10:53AM H37.00002 Droplet impact on liquid films in the presence of surfactants¹, KHADIJA JAIS, NATALIE YIP, Imperial College London, ZHIZHAO CHE, Tianjin University, China, OMAR MATAR, Imperial College London — In this study of droplet impact on liquid films, surfactants are added to the droplet, the liquid film or both, and the effects of different surfactant concentrations are investigated using high-speed imaging. The results show that surfactants suppress partial coalescence, due to damping of the capillary waves. Rebounding occurs more frequently when surfactants are added, as the surfactant molecules resist the drainage of the intervening air layer. When the droplet deforms the surfactant film, there is an uneven distribution of surfactant molecules along the interface, resulting in a surface tension gradient and a Marangoni stress. The Marangoni stress acts to even out the surface tension gradient and to aid rebounding. Surfactant droplet ruptures the film with a much lower surfactant concentration, leaving an apparent dry region on the substrate at the impact point. This is likely due to Marangoni stresses where the film pulls the droplet apart. As the film thickness is increased, a Worthington jet is formed, with secondary droplet(s) ejected from the jet only when surfactants are present. This study reveals that the presence of surfactants can significantly alter the impact process of droplets on liquid films.

¹EPSRC UK Programme Grant MEMPHIS (EP/K003976/1)

11:06AM H37.00003 Drop impact on liquid film: dynamics of interfacial gas layer, XIAOYU TANG, ABHISHEK SAHA, Princeton University, CHUNG K. LAW, Princeton University, Tsinghua University, CHAO SUN, Tsinghua University — Drop impacting liquid film is commonly observed in many processes including inkjet printing and thermal sprays. Owing to the resistance from the interfacial gas layer trapped between the drop and film surface, impact may not always result in coalescence; and as such investigating the behavior of the interfacial gas layer is important to understand the transition between bouncing and merging outcomes. The gas layer is, however, not easily optically accessible due to its microscopic scale and curved interfaces. We report the measurement of this critical gas layer thickness between two liquid surfaces using high-speed color interferometry capable of measuring micron and submicron thicknesses. The complete gas layer dynamics for the bouncing cases can be divided into two stages: the approaching stage when the drop squeezes the gas layer at the beginning of the impact, and the rebounding stage when the drop retracts and rebounds from the liquid film. The approaching stage is found to be similar across wide range of conditions studied. However, for the rebounding stage, with increase of liquid film thickness, the evolution of gas layer changes dramatically, displaying a non-monotonic behavior. Such dynamics is analyzed in lights of various competing timescales.

11:19AM H37.00004 Water drop impact onto oil covered solid surfaces¹ , NINGLI CHEN,

Nanjing University of Aeronautics and Astronautics, HUANCHEN CHEN, ALIDAD AMIRFAZLI, York University — Droplet impact onto an oily surface can be encountered routinely in industrial applications; e.g., in spray cooling. It is not clear from literature what impact an oil film may have on the impact process. In this work, water drop impact onto both hydrophobic (glass) and hydrophilic (OTS) substrates which were covered by oil films (silicone) of different thickness (5um-50um) and viscosity (5cst-100cst) were performed. The effects of drop impact velocity, film thickness, and viscosity of the oil film and wettability of the substrate were studied. Our results show that when the film viscosity and impact velocity is low, the water drop deformed into the usual disk shape after impact, and rebounded from the surface. Such rebound phenomena disappears, when the viscosity of oil becomes very large. With the increase of the impact velocity, crown and splashing appears in the spreading phase. The crown and splashing behavior appears more easily with the increase of film thickness and decrease of its viscosity. It was also found that the substrate wettability can only affect the impact process in cases which drop has a large Webber number ($We = 594$), and the film's viscosity and thickness are small.

¹This work was support by National Natural Science Foundation of China and the project number is 51506084.

11:32AM H37.00005 Numerical Study of High-Speed Droplet Impact on Wet Surfaces and its Potential for Removing Small Particles from the Surfaces , TOMOKI KONDO, KEITA ANDO, Department of Mechanical Engineering, Keio University —In liquid jet cleaning, high-speed droplet impact on wet surfaces is an important phenomenon to remove small-sized contaminant particles from the surfaces. Here, we consider high-speed droplet impact on a rigid wall covered with a liquid film in order to investigate shear flow created at the wall after the impact and its role of removing small particles. 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. The attached particles are assumed to be so small that the base liquid flow is undisturbed and flow around the particles is creeping. Stokes hydrodynamic force on the particles under the shear flow is evaluated in a one-way-coupling way. The particle removal is judged by a balance between the hydrodynamic force and particle adhesion of van der Waals type, with varying impact speed and film thickness.**11:45AM H37.00006 Continuous impact of microdrops on a liquid pool , JAE HONG LEE, SEUNGHO KIM, HO-YOUNG KIM, Seoul Natl Univ —**A single liquid drop impacting on a liquid pool generates a hemispherical crater, while impact of a stream of microdrops leads to a severely elongated crater whose depth can reach hundreds times the diameter of the impacting drop. Here we investigate experimentally and theoretically the evolution of the crater formed by continuous impact of microdrops, or a drop train. The crater is observed to elongate only up to a certain length at a constant rate and then be pinched off near the pool surface to convert into a cusp. We rationalize the constant elongation rate by assuming the crater growth as a superposition of crater formation due to individual drops. Also, we predict the maximum depth of the crater as a function of liquid properties and diameter, impact velocity and frequency of drops. Finally, we theoretically model the cusp shape, which agrees well with experiment.**11:58AM H37.00007 Oblique drop impact onto a deep liquid pool , MARISE V. GIELEN, PASCAL SLEUTEL, Physics of Fluids group, Faculty of Science and Technology, University of Twente, The Netherlands, JOS BENSCHOP, MICHEL RIEPEN, VICTORIA VORONINA, ASML The Netherlands B.V., The Netherlands, DETLEF LOHSE, JACCO H. SNOEIJER, MICHEL VERSLUIS, HANNEKE GELDERBLOM, Physics of Fluids group, Faculty of Science and Technology, University of Twente, The Netherlands —**While perpendicularly drop impact onto a deep liquid pool is widely studied, the dynamics after oblique drop impact remain to be quantified. Here we study, for the first time, oblique drop impact experiments onto a deep liquid pool using ultrafast imaging. We quantify the splashing behavior and derive a model to describe the splashing threshold based on the impact angle and Weber number of the impacting drop. In addition, we study the cavity formation below the water surface and quantify the cavity depth and displacement. Based on the asymmetric cavity dynamics, we develop a method to predict the direction in which a jetted droplet can escape the cavity.**12:11PM H37.00008 ABSTRACT WITHDRAWN —****12:24PM H37.00009 Droplet absorption by helically-supported capillary channels , MAVERICK TERRAZAS, DAVID THIESSEN, Washington State University —**A large aspect ratio capillary channel formed by filling a stretched spring with water under conditions of low transverse Bond number and connected to a constant low-pressure reservoir is shown to absorb water droplets that impinge on the channel. Experimental results for low Weber number droplet absorption using a half-second freefall apparatus in the lab will be presented as well as 1-g results for moderate Weber numbers where inertia dominates gravity. The effects of several variables on absorption including droplet Weber number, impact parameter, reservoir pressure, and spring pitch are examined. Channels of this type are envisioned for passive phase separation applications in microgravity or for Earth-based technologies requiring phase separation under low Bond number conditions and perhaps in large aspect ratio channels.**12:37PM H37.00010 Hydrodynamics and PIV study in the impingement zone formed by a droplet train¹ , ANOOP KANJIRAKAT, REZA SADR, Texas A and M University at Qatar, TAOLUE ZHANG, JAYAVEERA MUTHUSAMY, JORGE ALVARADO, Texas A and M University, TEXAS A AND M UNIVERSITY AT QATAR COLLABORATION, TEXAS A AND M UNIVERSITY COLLEGE STATION COLLABORATION —**Droplet impingement is encountered in numerous technical applications, such as ink jet printing, spray cooling, and fuel injection in internal combustion engines. Even though many studies in droplet impingement were conducted in past, not many have measured the near-wall velocities in the droplet impingement zone. With the goal of gaining a better understanding of the hydrodynamics in the impingement zone, well-controlled experiments are performed in combination with micro-PIV measurements and numerical simulations. Hydrodynamics of HFE-7100 droplets generated using a piezoelectric droplet generator, impinging on a pre-wetted surface is investigated. Micro-PIV studies in the high-velocity impingement zone are performed using one-micron meter fluorescent particles dispersed in HFE-7100 along with the double exposed images. Three-dimensional and 2D-axisymmetric numerical modeling for a transient droplet crown development is performed. The interface between the gas and the liquid is modeled using a Volume of Fluid (VOF) method. Numerical simulation results obtained are observed to be in good agreement with that of the experimental observations.

¹Supported by National Priority Research Program (NPRP) of Qatar National Research Fund (QNRF), Grant no.: NPRP 6-1304-2-525

Monday, November 21, 2016 10:40AM - 12:50PM —
Session H38 Flow Instability: Transition Portland Ballroom 255 - Nigel Goldenfeld, University of Illinois at Urbana-Champaign

10:40AM H38.00001 Is there universal predator-prey dynamics at the laminar-turbulent phase transition?, HONG-YAN SHIH, NIGEL GOLDENFELD, Department of Physics, University of Illinois at Urbana-Champaign — Direct numerical simulation of pipe flow shows that transitional turbulence is dominated by two collective modes: a longitudinal mode for small-scale turbulent fluctuations whose anisotropy induces an emergent large-scale azimuthal mode (so-called zonal flow) that inhibits anisotropic Reynolds stress [1]. This activation-inhibition interaction leads to stochastic predator-prey-like dynamics, from which it follows that the transition to turbulence belongs to the directed percolation universality class [1]. Here we show how predator-prey dynamics arises by deriving phenomenologically an effective field theory of the transition from a coarse-graining of the Reynolds equation. The rigorous mapping between the conserved currents in Rayleigh-Bénard convection (RBC), Taylor-Couette and pipe flows [2] suggests that the zonal flow-turbulence scenario might occur in these systems, consistent with observations of zonal flows in two-dimensional RBC [3], and bursts of transitional turbulence in Couette flow that follow the critical scalings of directed percolation [4]. [1] H.-Y. Shih et al., Nat. Phys. 12, 245 (2016). [2] B. Eckhardt et al., EPL 78, 24001 (2007). [3] D. Goluskin et al., J. Fluid Mech. 759, 360 (2014). [4] G. Lemoult et al., Nat. Phys. 12, 254 (2016).

10:53AM H38.00002 Flow structure in self-sustaining and intermittently turbulent reciprocating channel flow, ALIREZA EBADI, CHRISTOPHER WHITE, University of New Hampshire, YVES DUBIEF, University of Vermont, UVM TEAM, UNH TEAM — The leading order terms in the Reynolds-averaged momentum equation are studied to better understand the underlying mechanism of transition to turbulence in reciprocating channel flow. The balance of the leading order terms confirms that fully-developed turbulence first emerges at the early phases in the decelerating portion of the cycle. The underlying mechanism of this transition appears to be the emergence of an internal layer that first develops during the late phases of the accelerating portion of the cycle. In the absence of this internal layer, the flow remains transitional over the entire cycle. The turbulent structure associated with the internal layer is investigated using different flow structure identification schemes. In particular, the Q-R criteria and the triple decomposition of the strain rate tensor.

11:06AM H38.00003 Transitions to Turbulence in an Electromagnetically-Driven 2D Fluid¹, LOGAN KAGEORGE, Georgia Inst of Tech, JEFFREY TITHOF, University of Rochester, BALACHANDRA SURI, RAVI PALLANTLA, ROMAN GRIGORIEV, MICHAEL SCHATZ, Georgia Inst of Tech — We present an experimental and numerical analysis of the transition to turbulence for a quasi-two-dimensional liquid. Our system is a Kolmogorov-like flow, realized as a Lorentz-forced thin fluid layer, which exhibits shearing-induced vortex pattern formation. The system dynamics are quantified using particle image velocimetry to create time-resolved velocity fields. We focus on the series of bifurcations leading to spatiotemporally chaotic behavior and quantitatively compare these results with simulations of an identical system to adjust system-specific parameters in accordance with first-principle modifications to the Navier-Stokes equations.

¹This work is supported by the National Science Foundation (NSF CMMI 12-34436).

11:19AM H38.00004 Experiments on Laminar to Turbulence Transition and Relaminarization in Pulsatile Flows, JOAN GOMEZ, The City College of New York , OLEK GOUSHCHA, Manhattan College, YIANNIS ANDREOPoulos, The City College of New York — Biological flows display laminar-turbulence-laminar transitions due to the cyclic nature of a beating heart. Addressing the question of how turbulence appears, decays and is suppressed in the cardiovascular system, particularly in the large arteries, is challenging due to flow unsteadiness, very complicated geometry and flow-wall interaction. In the present work we have designed and tested a facility to simulate unsteady pulsatile flows and the onset of transition under varying Reynolds and Womersley numbers. A moving piston is used to generate a flow pulsation and control the velocity amplitude. Time-Resolved Particle Image Velocimetry (TR-PIV) techniques were used to acquire velocity data on the plane of a CW laser illumination. Two different decompositions were applied to analyze the non-stationary and non-linear time-dependent data, the Empirical Mode Decomposition (EMD) and the Trend Removal Method (TRM). Two flow regimes were found, one in which the pulsatile flow exhibits phase-locked turbulence which is associated with the stabilizing effects of longitudinal straining during acceleration and a second where transition occurs very close to the wall while the core remains laminar.

11:32AM H38.00005 Routes to turbulence in the rotating disk boundary-layer of a rotor-stator cavity, EUNOK YIM, ERIC SERRE, DENIS MARTINAND, Aix-Marseille Universite, CNRS, Ecole Centrale Marseille, Laboratoire M2P2 UMR 7340, Marseille, France, JEAN-MARC CHOMAZ, LadHyX, CNRS, Ecole polytechnique, Palaiseau, France — The rotating disk is an important classical problem, due to the similarities between the 3D boundary layers on a disk and a swept aircraft wing. It is nowadays admitted that a direct transition to turbulence may exist through a steep-fronted nonlinear global mode located at the boundary between the locally connectively and absolutely unstable regions (Pier 2003; Viaud et al. 2008, 2011; Ibayama et al. 2014 and others). However, recent studies (Healey 2010; Harris et al. 2012; Ibayama et al. 2013) suggest that there may be an alternative route starting at lower critical Reynolds number, based on convective travelling waves but this scenario is still not fully validated and proven. To better characterize such transition, direct numerical simulations are performed in a closed cylindrical rotor-stator cavity (without hub) up to $Re = O(10^5)$. All boundaries are no slip and for the stable region around the rotation axis prevents the disturbances coming from the very unstable stator boundary to disturb the rotor boundary layer. Different transition scenarios to turbulence are investigated when the rotor boundary layer is forced at different positions and forcing amplitude. The associated dynamics of coherent structures in various flow regions are also investigated when increasing Re .

11:45AM H38.00006 Coherent structures in the asymptotic suction boundary layer over a heated plate, STEFAN ZAMMERT, TU Delft, BRUNO ECKHARDT, Philipps-Universitt Marburg — The asymptotic suction boundary layer over a heated plate [S. Zammert et al. arXiv:1605.06956] is a good point of entry to study the dynamics of thermal boundary layers by means of dynamical systems theory. We analyze the stability of this flow in dependence on the Reynolds, Rayleigh and Prandtl numbers and identify the bifurcating secondary solutions. It turns out that in contrast to the Rayleigh-Bénard problem the base flow becomes unstable in a subcritical bifurcation. In the subcritical range the secondary solutions are the starting point for a bifurcation cascade that creates a chaotic attractor. As in other subcritical flow, a boundary crisis bifurcation turns this attractor into a chaotic saddle causing transient chaotic motion in the subcritical range. We also calculate mean turbulent profiles and their scaling with the Rayleigh and Prandtl number. It turned out that the turbulent flow in the system is characterized by large-scale coherent structures which extend surprisingly far above the plate.

11:58AM H38.00007 Canonical Nonlinear Viscous Core Solution in pipe and elliptical geometry¹, OZGE OZCAKIR, Monash University — In an earlier paper (Ozcakir et. al.(2016)), two new nonlinear traveling wave solutions were found with collapsing structure towards the center of the pipe as Reynolds number $R \rightarrow \infty$, which were called Nonlinear Viscous Core (NVC) states. Asymptotic scaling arguments suggested that the NVC state collapse rate scales as $R^{-1/4}$ where axial, radial and azimuthal velocity perturbations from Hagen-Poiseuille flow scale as $R^{-1/2}$, $R^{-3/4}$ and $R^{-3/4}$ respectively, while $(1 - c) = O(R^{-1/2})$ where c is the traveling wave speed. The theoretical scaling results were roughly consistent with full Navier-Stokes numerical computations in the range $10^5 < R < 10^6$. In the present paper, through numerical solutions, we show that the scaled parameter free canonical differential equations derived in Ozcakir et. al.(2016) indeed has solution that satisfies requisite far-field conditions. We also show that these are in good agreement with full Navier-Stokes calculations in a larger R range than previously calculated (R upto 10^6). Further, we extend our study to NVC states for pipes with elliptical cross-section and identify similar canonical structure in these cases.

¹National Science Foundation NSF-DMS-1515755, EPSRC grant EP/1037948/1

12:11PM H38.00008 Patterns of the turbulent Taylor-Couette flow¹, ARNAUD PRIGENT, ABDESSAMAD TALIOUA, INNOCENT MUTABAZI, LOMC - CNRS UMR 6294 — We are interested in the study of the transition to turbulence in the Taylor-Couette flow, the flow between two independently rotating coaxial cylinders. Once the geometry is fixed, the flow is controlled by the inner and outer Reynolds numbers and present a large variety of flow regimes. In counter-rotation, the transition is characterized by a succession of more or less turbulent flow regimes: intermittency with turbulent spots, spiral turbulence, featureless turbulence. For larger values of the inner Reynolds number, turbulent Taylor roll re-emerge from the featureless turbulence and remain for very large values of the Reynolds numbers. Bifurcations between different turbulent rolls states are even observed in the ultimate turbulence regime. Nevertheless the transition from the featureless turbulence to the turbulent rolls still requires a detailed study and the mechanism which causes and sustains turbulent spots or turbulent spirals remains unknown. In this study we present new experimental information on the organization of the flow for the different regimes with turbulence. The experiments are conducted in a Taylor-Couette flow with $\eta = 0.8$. Stereo-Particle Image Velocimetry measurements and visualizations of the different flow regimes are realized and discussed.

¹This work was supported by the ANR TRANSFLOW - ANR-13-BS09-0025

12:24PM H38.00009 Perturbing turbulence beyond collapse, JAKOB KHNEN, DAVIDE SCARSELLI, BJRN HOF, IST Austria, NONLINEAR DYNAMICS AND TURBULENCE GROUP TEAM — Wall-bounded turbulent flows are considered to be in principle stable against perturbations and persist as long as the Reynolds number is sufficiently high. We show for the example of pipe flow that a specific perturbation of the turbulent flow field disrupts the genesis of new turbulence at the wall. This leads to an immediate collapse of the turbulent flow and causes complete relaminarisation further downstream. The annihilation of turbulence is effected by a steady manipulation of the streamwise velocity component only, greatly simplifying control efforts which usually require knowledge of the highly complex three dimensional and time dependent velocity fields. We present several different control schemes from laboratory experiments which achieve the required perturbation of the flow for total relaminarisation. Transient growth, a linear amplification mechanism measuring the efficiency of eddies in redistributing shear that quantifies the maximum perturbation energy amplification achievable over a finite time in a linearized framework, is shown to set a clear-cut threshold below which turbulence is impeded in its formation and thus permanently annihilated.

12:37PM H38.00010 Stability and sensitivity analysis of experimental data for passive control of a turbulent wake, LORENZO SICONOLFI, SIMONE CAMARRI, University of Pisa, RENZO TRIP, JENS H. M. FRANSSON, Linne' Flow Centre, KTH Mechanics — When the linear stability analysis is applied to the mean flow field past a bluff body, a quasi-marginally stable mode is identified, with a frequency very close to the real vortex shedding one. A formally consistent approach to justify this kind of analysis is based on a triple decomposition of the flow variables[1]. With this formalism, the adjoint-based sensitivity analysis can be extended to investigate passive controls of high-Reynolds-number wakes (e.g.[2]). The objective of the present work is to predict the effect of a small control cylinder on the vortex shedding frequency in a turbulent wake with an analysis which solely relies on PIV measurements available for the considered flow. The key ingredient of the numerical analysis is an ad-hoc tuned model for the mean flow field, built using an original procedure which includes all the experimental information available on the flow. This analysis is here applied to the wake flow past a thick porous plate at Reynolds numbers in the range between $Re=6.7 \times 10^3$ and $Re=5.3 \times 10^4$. It is shown that the derived control map agrees reasonably well with the equivalent map obtained experimentally. [1]Viola,F.,Iungo,G.V.,Camarri,S.,Gallaire,F.,J.FluidMech.750(R1),2014 [2]Meliga,P.,Pujals,G.,Serre,E.,Phys.Fluids24(061701),2012

Monday, November 21, 2016 10:40AM - 12:50PM –
Session H39 Bio: Experiment and Theory of Microswimming Portland Ballroom 256 - Daphne Klotsa, University of North Carolina at Chapel Hill

10:40AM H39.00001 Onset of motion and collective behavior for two-sphere swimmers, DAPHNE KLOTSA, University of North Carolina at Chapel Hill, KYLE BALDWIN, RICHARD HILL, ROGER BOWLEY, MICHAEL SWIFT, University of Nottingham, SHANNON JONES, University of North Carolina at Chapel Hill — We describe experiments and simulations demonstrating the propulsion of a neutrally buoyant swimmer that consists of a pair of spheres attached by a spring, immersed in a vibrating fluid at intermediate Reynolds numbers. The vibration of the fluid induces relative motion of the spheres which, for sufficiently large amplitudes, can lead to motion of the center of mass of the two spheres. We find that the swimming speed obtained from both experiment and simulation agree and collapse onto a single curve if plotted as a function of the streaming Reynolds number, suggesting that the propulsion is related to streaming flows. There appears to be a critical onset value of the streaming Reynolds number for swimming to occur. We observe a change in the streaming flows as the Reynolds number increases, from that generated by two independent oscillating spheres to a collective flow pattern around the swimmer as a whole. The mechanism for swimming is traced to a strengthening of a jet of fluid in the wake of the swimmer and steady streaming. We discuss the collective behavior of such swimmers.

10:53AM H39.00002 An underwater robo-leader for collective motion studies, YAIR SANCHEZ, MONICA M. WILHELMUS, Univ of California - Riverside — A wide range of aquatic species, from bacteria to large tuna, exhibits collective behavior. It has long been hypothesized that the formation of complex configurations brings an energetic advantage to the members of a group as well as protection against larger predators or harmful agents. Lately, however, laboratory experiments have suggested that both the physics and the behavioral aspects of collective motion yield more complexity than previously attributed. With the goal to understand the fluid mechanical implications behind collective motion in a laboratory setting, we have developed a new device to induce this behavior on demand. Following recent studies of lab-induced vertical migration of Artemia salina, we have designed and constructed a remotely controlled underwater robotic swimmer that acts as a leader for groups of phototactic organisms. Preliminary quantitative flow visualizations done during vertical migration of brine shrimp show that this new instrument does induce collective motion in the laboratory. With this setup, we can address the hydrodynamic effect of having different swarm configurations, a variable that so far has been challenging to study in a controllable and reproducible manner.

11:06AM H39.00003 Flow caused by the stalk contraction of Vorticella¹, SANGJIN RYU, EUN-GUL CHUNG, DAVID ADMIRAAL, University of Nebraska-Lincoln — Vorticella is a stalked protozoan, and its ultrafast stalk contraction moves the spherically-shrunken cell body (zoid) and thus causes surrounding water to flow. Because the fluid dynamics of this water flow is important for understanding the motility of Vorticella, we investigated the flow based on various fluid dynamics approaches. To find why Vorticella contracts its stalk, we propose a hypothesis that the protist utilizes the contraction-induced water flow to augment transport of food particles. This hypothesis was investigated using a computational fluid dynamics (CFD) model, which was validated with an experimental scale model of Vorticella. The CFD model enabled calculating the motion of particles around Vorticella and thus quantifying the transport effect of the stalk contraction. Also, we have developed a hydrodynamic drag model for easier estimation of Vorticella's contractility without using the CFD model. Because the contractile force of the stalk equals the drag on the moving zoid, the model enabled evaluating the contractile force and energetics of Vorticella based on its contraction speed. Analyses using the drag model show that the stalk contractility of Vorticella depends on the stalk length.

¹This study was supported by UNL Layman Seed Grant and Nebraska EPSCoR First Award Grant.

11:19AM H39.00004 Simple low Reynolds number microswimmers, U KEI CHEANG, Drexel University, MIN JUN KIM, Southern Methodist University — An extremely simple low Reynolds number microswimmer had been observed to swim in bulk fluid. The development of microscopic swimmers had been hindered by technical limitations in micro- and nanofabrication. To address this practical problem, the minimal geometrical requirements for swimming in low Reynolds number has been investigated. Micro- and nanofabrication of complex shapes with specialized materials, such as helices or flexible bodies, on a massive scale requires sophisticated state of the art technologies which have size limitations. In contrast, simple shaped structures, such as spherical particles, can be synthesized massively using chemical methods with relative ease at low costs. In this work, simple microswimmers were fabricated by conjugating two microbeads with debris attached to their surface. The debris allow the 2-bead structures to have two or more planes of symmetry, thus, allowing them to swim in bulk fluid at low Reynolds number. The microswimmers are magnetically actuated and controlled via a rotating magnetic field generated by an electromagnetic coil system. The microswimmers' velocity profiles had been characterized with respect to increasing rotating frequency. Furthermore, the motion of the microswimmers were analyzed using image processing. Finally, their swimming capability had been shown through experiments by steering the microswimmers in any desired direction.

11:32AM H39.00005 The geometry and fluid dynamics of two- and three-dimensional maneuvers of burrowing and swimming *C. elegans*, JERZY BLAWZDZIEWICZ, ALEJANDRO BILBAO, AMAR PATEL, Department of Mechanical Engineering, Texas Tech University, MIZANUR RAHMAN, SIVA A. VANAPALLI, Department of Chemical Engineering, Texas Tech University — In its natural environment, which is decomposing organic matter and water, *C. elegans* swims and burrows in 3D complex media. Yet quantitative investigations of *C. elegans* locomotion have been limited to 2D motion. 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. Here, we follow with the first full 3D description of how *C. elegans* moves in complex 3D environments. We show that the nematode can explore 3D space by combining 2D turns with roll maneuvers that result in rotation of the undulation plane around the direction of motion. Roll motion is achieved by superposing a 2D curvature wave with nonzero body torsion; 2D turns (within the current undulation plane) are attained by variation of undulation wave parameters. Our results indicate that while hydrodynamic interactions reduce angles of 2D turns, the roll efficiency is significantly enhanced. This hydrodynamic effect explains the rapid nematode reorientation observed in 3D swimming.

11:45AM H39.00006 Identification of internal properties of fibers and micro-swimmers, FRANCK PLOURABOUE, IBRAHIMA THIAM, BLAISE DELMOTTE, ERIC CLIMENT, IMFT UMR 5502 CNRS-INPT-UPS, PSC COLLABORATION — In this presentation we discuss the identifiability of constitutive parameters of passive or active micro-swimmers. We first present a general framework for describing fibers or micro-swimmers using a bead-model description. Using a kinematic constraint formulation to describe fibers, flagellum or cilia, we find explicit linear relationship between elastic constitutive parameters and generalised velocities from computing contact forces. This linear formulation then permits to address explicitly identifiability conditions and solve for parameter identification. We show that both active forcing and passive parameters are both identifiable independently but not simultaneously. We also provide unbiased estimators for elastic parameters as well as active ones in the presence of Langevin-like forcing with Gaussian noise using normal linear regression models and maximum likelihood method. These theoretical results are illustrated in various configurations of relaxed or actuated passives fibers, and active filament of known passive properties, showing the efficiency of the proposed approach for direct parameter identification. The convergence of the proposed estimators is successfully tested numerically.

11:58AM H39.00007 Hydrodynamics of Microbial Filter-Feeding¹, ANDERS ANDERSEN, Department of Physics and Centre for Ocean Life, Technical University of Denmark, LASSE TOR NIELSEN, National Institute of Aquatic Resources and Centre for Ocean Life, Technical University of Denmark, JULIA DOLGER, Department of Physics and Centre for Ocean Life, Technical University of Denmark, THOMAS KIORBOE, National Institute of Aquatic Resources and Centre for Ocean Life, Technical University of Denmark — Microbial filter-feeders form an important group of plankton with significance to the aquatic food webs. While the concept of filter-feeding is straightforward, our quantitative understanding of microbial filter-feeding leaves a lot to be desired. As a model organism, we focus on the filter-feeding choanoflagellate *Diaphanoeca grandis*. We quantify the feeding flow using particle tracking, and demonstrate that hydrodynamic theory underestimates the observed clearance rate by an order of magnitude. We find similar discrepancies for other choanoflagellate species, highlighting an apparent paradox. To resolve the paradox we argue that *D. grandis* and other choanoflagellates must have so far unbeknownst morphological features. Specifically, we suggest a flagellar vane that connects the flagellum to the filter, as known in choanocytes of sponges, creating a radically different, and order of magnitude more capable, pumping mechanism.

¹The Centre for Ocean Life is a VKR Centre of Excellence supported by the Villum Foundation.

12:11PM H39.00008 Feeding, Swimming and Navigation of Colonial Microorganisms

, JULIUS KIRKEGAARD, AMBRE BOUILLANT, ALAN MARRON, KYRIACOS LEPTOS, RAYMOND GOLDSTEIN, Univ of Cambridge — Animals are multicellular in nature, but evolved from unicellular organisms. In the closest relatives of animals, the choanoflagellates, the unicellular species *Salpingoeca rosetta* has the ability to form colonies, resembling true multicellularity. In this work we use a combination of experiments, theory, and simulations to understand the physical differences that arise from feeding, swimming and navigating as colonies instead of as single cells. We show that the feeding efficiency decreases with colony size for distinct reasons in the small and large Péclet number limits, and we find that swimming as a colony changes the conventional active random walks of microorganism to stochastic helices, but that this does not hinder effective navigation towards chemoattractants.

12:24PM H39.00009 Three-dimensional simulation of pseudopod-driven swimming of amoeboid cells¹

, ERIC CAMPBELL, PROSENJIT BAGCHI, Rutgers University — Pseudopod-driven locomotion is common in eukaryotic cells, such as amoeba, neutrophils, and cancer cells. Pseudopods are protrusions of the cell body that grow, bifurcate, and retract. Due to the dynamic nature of pseudopods, the shape of a motile cell constantly changes. The actin-myosin protein dynamics is a likely mechanism for pseudopod growth. Existing theoretical models often focus on the acto-myosin dynamics, and not the whole cell shape dynamics. Here we present a full 3D simulation of pseudopod-driven motility by coupling a surface-bound reaction-diffusion (RD) model for the acto-myosin dynamics, a continuum model for the cell membrane deformation, and flow of the cytoplasmic and extracellular fluids. The whole cell is represented as a viscous fluid surrounded by a membrane. A finite-element method is used to solve the membrane deformation, and the RD model on the deforming membrane, while a finite-difference/spectral method is used to solve the flow fields inside and outside the cell. The fluid flow and cell deformation are coupled by the immersed-boundary method. The model predicts pseudopod growth, bifurcation, and retraction as observed for a swimming amoeba. The work provides insights on the role of membrane stiffness and cytoplasmic viscosity on amoeboid swimming.

¹Funded by NSF CBET 1438255

12:37PM H39.00010 Fluid pumping using magnetic cilia¹

, SRINIVAS HANASOGE, MATT BALLARD, ALEXANDER ALEXEEV, PETER HESKETH, Georgia Tech, WOODRUFF SCHOOL OF MECHANICAL ENGINEERING TEAM — Using experiments and computer simulations, we examine fluid pumping by artificial magnetic cilia fabricated using surface micromachining techniques. An asymmetry in forward and recovery strokes of the elastic cilia causes the net pumping in a creeping flow regime. We show this asymmetry in the ciliary strokes is due to the change in magnetization of the elastic cilia combined with viscous force due to the fluid. Specifically, the time scale for forward stroke is mostly governed by the magnetic forces, whereas the time scale for the recovery stroke is determined by the elastic and viscous forces. These different time scales result in different cilia deformation during forward and backward strokes which in turn lead to the asymmetry in the ciliary motion. To disclose the physics of magnetic cilia pumping we use a hybrid lattice Boltzmann and lattice spring method. We validate our model by comparing the simulation results with the experimental data. The results of our study will be useful to design microfluidic systems for fluid mixing and particle manipulation including different biological particles.

¹USDA and NSF

Monday, November 21, 2016 10:40AM - 12:37PM –

Session H40 Porous Media Flows: Imbibition, Injection and Wicking Portland Ballroom
253-258-254-257 - German Drazer, Rutgers University

10:40AM H40.00001 Using liquid droplet penetration experiments to determine wetting properties of powders

, GERARDO CALLEGARI, ZHANJI LIU, YIFAN WANG, FERNANDO MUZZIO, GERMAN DRAZER, Rutgers, The State University of New Jersey — We show that the spontaneous penetration of droplets on a powder bed provides a simple way to characterize the wetting properties of a test liquid. Specifically, we demonstrate that calculating the appropriate dimensionless penetrating volume and time, and performing supplementary experiments with a reference liquid, it is possible to obtain the contact angle between the test liquid and the powder. Interestingly, the proposed analysis lets us calculate the contact angle without having to compute the solution to the 3D penetration problem. This approach is valid when the contact area between the droplet and the powder bed remains constant, which is a good approximation in many powder systems. We first test the validity of our approach by studying droplets of different sizes and show that the non-dimensional penetration curves are independent of the initial volume and contact radius of the drops, as predicted. We then use a reference liquid (silicone oil) to measure the contact angle of water on three powder systems with increasing number of pharmaceutical components and different processing conditions known to affect blend wettability. We show that the proposed method is able to capture the overlubrication of the blend, a well-known effect in pharmaceutical manufacturing.

10:53AM H40.00002 Fluid-driven Fractures and Backflow in a Multilayered Elastic Matrix¹

, SAMUEL SMIDDY, Department of Chemical and Biological Engineering, Princeton University, CHING-YAO LAI, HOWARD STONE, Department of Mechanical and Aerospace Engineering, Princeton University — We study the dynamics when pressurized fluid is injected at a constant flow rate into a multi-layered elastic matrix. In particular, we report experiments of such crack propagation as a function of orientation and distance from the contact of the layers. Subsequently we study the shape and propagation of the fluid along the contact of layers as well as volume of fluid remaining in the matrix once the injection pressure is released and “flowback” occurs. The experiments presented here may mimic the interaction between hydraulic fractures and pre-existing fractures and the dynamics of flowback in hydraulic fracturing.

¹Study made possible by the Andlinger Center for Energy and the Environment and the Fred Fox Fund

11:06AM H40.00003 Sessile drops imbibition over nano porous substrates : application for art painting restoration¹, MARGUERITE LÉANG, Laboratory FAST, Orsay, France - Laboratory Léon Brillouin, CEA Saclay, France, LUDOVIC PAUCHARD, Laboratory FAST, Orsay, France, LAY-THENG LEE, FRÉDÉRIC OTT, Laboratory Léon Brillouin, CEA Saclay, France, FRÉDÉRIQUE GIORGIUTTI-DAUPHINÉ, Laboratory FAST, Orsay, France

— Art painting restoration aims to restore and preserve the integrity of a painting. Most of the techniques consist of depositing solvents on the surface of the painting to dissolve the varnish layer. However the sublayers can be damaged by the penetration of the solvent, possibly resulting in swelling or cracking processes. Due to the physical and chemical complexity of the pictorial layer, we propose to study solvent penetration in model nano porous media obtained by controlled drying of aqueous dispersions of silica nanoparticles. We present the dynamics of imbibition of sessile solvent drops on porous media with different pore sizes. Three different regimes in the evolution of the drop size with time are identified. Our experimental set-up provides a precise direct quantification of the different flows outside and through the porous media. Different experimental techniques are used to characterize the porous media : neutron imaging to determine the permeability and microindentation testing to estimate the mechanical properties of the media after the imbibition process.

¹Supported by PALM LabEx grant

11:19AM H40.00004 Dynamics of liquid imbibition through paper fibers with intra-pores¹, SOOYOUNG CHANG, JAEDEOK SEO, SEOKBIN HONG, Sogang Univ, DUCK-GYU LEE, Korea Institute of Machinery & Materials, WONJUNG KIM, Sogang Univ — The accurate control of liquid imbibition in paper is crucial for the applications of paper to microfluidic devices. However, the classical model for capillary flow in porous media, Lucas-Washburn law, has limitations in predicting the flow in a complex fiber network such as paper. We here report that intra-fiber pores in paper are mainly responsible for the limited accuracy of the previous model. From our experiment, we observed that liquid may imbibe through intra-pores in cellulose fibers as through the pores formed by fiber network. We experimentally measured the flow rate through the intra-pores and theoretically developed a hydrodynamic model for liquid imbibition through paper fibers with intra-pores. Our theoretical predictions are shown to agree well with experimental observations, leading to the physical reasons behind the limits of Lucas-Washburn law.

¹This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP) (No.2015R1A2A2A04006181).

11:32AM H40.00005 Wettability control on multiphase flow in patterned microfluidics, RUBEN JUANES, BENZHONG ZHAO, Massachusetts Institute of Technology, CHRISTOPHER MACMINN, University of Oxford — Multiphase flow in porous media is important in many natural and industrial processes, including geologic CO₂ sequestration, enhanced oil recovery, and water infiltration into soil. Although it is well known that the wetting properties of porous media can vary drastically depending on the type of media and pore fluids, the effect of wettability on multiphase flow continues to challenge our microscopic and macroscopic descriptions. Here, we study the impact of wettability on viscously unfavorable fluid-fluid displacement in disordered media by means of high-resolution imaging in microfluidic flow cells patterned with vertical posts. By systematically varying the wettability of the flow cell over a wide range of contact angles, we find that increasing the substrate's affinity to the injected fluid results in more efficient displacement of the defending fluid up to a critical wetting transition, beyond which the trend is reversed. We identify the pore-scale mechanisms—cooperative pore filling (increasing displacement efficiency) and corner flow (decreasing displacement efficiency)—responsible for this macroscale behavior, and show that they rely on the inherent 3D nature of interfacial flows, even in quasi-2D media. Our results demonstrate the powerful control of wettability on multiphase flow in porous media, and show that the markedly different invasion protocols that emerge—from pore-filling to post-bridging—are determined by physical mechanisms that are missing from current pore-scale and continuum-scale descriptions.

11:45AM H40.00006 Wettability effects on fluid-fluid displacement in a capillary tube, BENZHONG ZHAO, AMIR PAHLAVAN, Massachusetts Institute of Technology, LUIS CUETO-FELGUEROZO, Universidad Politcnica de Madrid, RUBEN JUANES, Massachusetts Institute of Technology — Fluid-fluid displacement in a capillary tube is a classical problem in fluid mechanics, and it serves as a simple, but important analogue to multiphase flow in porous media. Despite many experimental and modeling studies of this problem, several key phenomena remain poorly understood. Here we experimentally study the constant-rate displacement of glycerol by air in a capillary tube. By treating the inside of the capillary, we obtain two distinct wetting conditions. We visualize the dynamics of the fluid-fluid interface in high-resolution for a wide range of capillary numbers (Ca). At small Ca, the air/glycerol interface remains spherical, whose curvature varies continuously as a function of Ca. At large Ca, the invading air forms a finger that advances along the center of the tube, leaving behind the contact-line and a macroscopic film of glycerol on the wall. We find that both the critical Ca at which film formation occurs and the speed of the contact-line is strongly controlled by the wettability of the tube. We demonstrate that these salient features of the experiment can be reproduced by a phase-field model of the system.

11:58AM H40.00007 Capillary imbibition in parallel tubes¹, OLIVER MCRAE, Boston University, T. S. RAMAKRISHNAN, Schlumberger-Doll Research, JAMES BIRD, Boston University — In modeling porous media two distinct approaches can be employed; the sample can be examined holistically, using global variables such as porosity, or it can be treated as a network of capillaries connected in series to various intermediate reservoirs. In forced imbibition this series-based description is sufficient to characterize the flow, due to the presence of an externally maintained pressure difference. However, in spontaneous imbibition, flow is driven by an internal capillary pressure, making it unclear whether a series-based model is appropriate. In this talk, we show using numerical simulations the dynamics of spontaneous imbibition in concentrically arranged capillary tubes. This geometry allows both tubes access to a semi-infinite reservoir but with inlets in close enough proximity to allow for interference. We compare and contrast the results of our simulations with theory and previous experiments.

¹Schlumberger-Doll Research

12:11PM H40.00008 Forced imbibition through model porous media, CELESTE ODIER, Univ Lyon, Ens de Lyon, Univ Claude Bernard, CNRS, Laboratoire de Physique, F-69342 Lyon, France. Total SA, BERTRAND LEVACHE, Total SA. Laboratoire Physico-Chimie des Interfaces Complexes, Total- ESPCI ParisTech CNRS - UPMC, DENIS BARTOLO, Univ Lyon, Ens de Lyon, Univ Claude Bernard, CNRS, Laboratoire de Physique, F-69342 Lyon, France — A number of industrial and natural process ultimately rely on two-phase flow in heterogeneous media. One of the most prominent example is oil recovery which has driven fundamental and applied research in this field for decades. Imbibition occurs when a wetting fluid displaces an immiscible fluid e.g. in a porous media. Using model microfluidic experiment we control both the geometry and wetting properties of the heterogenous media, and show that the typical front propagation picture fails when imbibition is forced and the displacing fluid is less viscous than the non-wetting fluid. We identify and quantitatively characterize four different flow regimes at the pore scale yielding markedly different imbibition patterns at large scales. In particular we will discuss the transition from a conventional 2D-front propagation scenario to a regime where the meniscus dynamics is an intrinsically 3D process.

12:24PM H40.00009 On the Examination of Darcy Permeability a Thin Fibrous Porous Layer¹, ZENGHAO ZHU, QIUYUN WANG, QIANHONG WU, Villanova University, VUCBMSS TEAM — In this paper, we report a novel experimental approach to investigate the Darcy permeability of a soft and thin fibrous porous layer. The project is inspired by recent studies involved compression of very thin porous films and the resultant pore fluid flow inside the confined porous structure. The Darcy permeability plays a critical role during the process, which however, is tricky to measure due to the very thin nature of the porous media. In the current study, a special micro-fluidic device is developed that consists of a rectangular flow channel with adjustable gap height ranging from 20 mm to 0.5 mm. Air is forced through the thin gap filled with testing fibrous materials. By measuring the flow rate and the pressure drop, we have successfully obtained the Darcy permeability of different thin porous sheets at different compression ratios. Furthermore, the surface area of the fibers are evaluated using a Micromeritics®ASAP 2020 (Accelerated Surface Area and Porosimetry) system. We found that, although the functions relating the permeability and porosities are different for different fibrous materials, these functions collapse to a single relationship if one express the permeability as a function of the solid phase surface area per unit volume. This finding provides a useful approach to evaluate the permeability of very thin fibrous porous sheet, which otherwise is difficult to measure directly.

¹This research was supported by the National Science Foundation under Award 1511096.

Monday, November 21, 2016 2:30PM - 3:05PM –
Session J32 Near Wall Turbulence - An Experimental View Oregon Ballroom 201-202-203 -
Ivan Marusic, University of Melbourne

2:30PM J32.00001 Near Wall Turbulence: an experimental view.¹, MICHEL STANISLAS, Ecole Centrale de Lille — The aim of this presentation is to summarize the understanding of the near wall turbulence phenomena obtained at Laboratoire de Mécanique de Lille using both hot wire anemometry and PIV. A wind tunnel was built in 1993 specifically designed for these two measurement techniques and aimed at large Reynolds numbers. Several experiments were performed since then in the frame of different PhDs and European projects, all aimed at evidencing turbulence organization in this region. These have fully benefited of the extraordinary development of PIV in that time frame, which has allowed entering visually and quantitatively inside the complex spatial and temporal structure of near wall turbulence. The presentation will try to emphasize the benefit of this approach in terms of understanding and modelling, illustrated by some representative results obtained.

¹M. Stanislas particularly acknowledges the financial support of Region Nord Pas de Calais, unmissing during 25 years.

Monday, November 21, 2016 2:30PM - 3:05PM –
Session J40 Colloids at Curved Fluid Interfaces Portland Ballroom 253-258-254-257 -

2:30PM J40.00001 Colloids at Curved Fluid Interfaces¹, KATHLEEN STEBE, University of Pennsylvania — Fluid interfaces are remarkable sites for colloidal assembly. When a colloid attaches to a fluid interface, it distorts a region around it; this distortion has an associated capillary energy, the product of its area and interfacial tension. The particles capillary energy depends on the local interface curvature. By molding the interface, we can define curvature fields that drive microparticles along pre-determined paths. This example captures the emergent nature of the interactions. We discuss curvature fields as analogues to external electro-magnetic fields, and define curvatures that drive particles to well-defined locations, and to equilibrium sites far from boundaries. Particle-particle and particle-curvature interactions can guide particles into structures via interaction among many particles. This work demonstrates the potential importance of curvature capillary interactions in schemes to make reconfigurable materials, since interfaces and their associated capillary energy landscapes can be readily reconfigured. Analogies in other soft systems will be described.

¹Support acknowledged from NSF DMR 1607878

Monday, November 21, 2016 3:10PM - 3:45PM –
Session K32 Turbulent Flow Physics and Noise in High Reynolds Number Compressible Jets Oregon Ballroom 201-202-203 - Robert Moser, University of Texas, Austin

3:10PM K32.00001 Turbulent Flow Physics and Noise in High Reynolds Number Compressible Jets, MARK GLAUSER, Syracuse University — In this talk I will present a snapshot of our ongoing research in high Reynolds number turbulent compressible jets. The high speed axisymmetric jet work (Mach 0.6 – 1.1) has been jointly performed with Spectral Energies LLC through AFRL support and involves 10 kHz and large window PIV data extracted from the near field jet plume, simultaneously sampled with near field pressure and far field noise. We have learned from the simultaneously sampled 10 kHz PIV near field plume and far field noise data, using POD/OID and Wavelet filtering, that there are certain “loud” velocity modes that have low averaged turbulent kinetic energy content but strongly correlate with the far field noise. From the large window PIV data obtained at Mach 1.0 and 1.1, specific POD modes were found to contain important physics of the problem. For example, the large-scale structure of the jet, shock-related fluctuations, and turbulent mixing regions of the flow were isolated through POD. By computing cross correlations, particular POD modes were found to be related to particular noise spectra. I will conclude with a description of our complex nozzle work which uses the multi-stream supersonic single expansion rectangular nozzle (SERN) recently installed in our large anechoic chamber at SU. This work is funded from both AFOSR (joint with OSU with a primary focus on flow physics) and Spectral Energies LLC (via AFRL funds with a focus on noise). Particular emphasis will be on insight gained into this complex 3D flow field (and its relationship to the far field noise) from applications of POD, Wavelet filtering and DMD to various numerical (LES) and experimental (PIV, high speed schlieren, near and far field pressure) data sets, at a core nozzle Mach number of 1.6 and a second stream Mach number of 1.0.

Monday, November 21, 2016 3:10PM - 3:45PM –
Session K40 Settling and Swimming in Density Stratified Fluids Portland Ballroom 253-258-254-257 - Michael Plesniak, George Washington University

3:10PM K40.00001 Settling and swimming in density stratified fluids , AREZOO ARDEKANI,

Purdue University — Many aquatic environments are stratified, characterized by regions of vertical variation in fluid density caused by gradients in temperature or salinity. In oceans and lakes, intense biological activity and accumulation of particles and organisms are associated with pycnoclines and the occurrence of important environmental and oceanographic processes is correlated with stratification. We explore the effects of stratification on the fundamental hydrodynamics of small organisms, settling particles, and rising drops. These results demonstrate an unexpected effect of buoyancy, potentially affecting a broad range of processes at pycnoclines in oceans and lakes. In particular, stratification has a major effect on the flow field, energy expenditure and nutrient uptake of small organisms. We show that elongation affects both the settling orientation and the settling rate of particles in stratified fluids, which will have direct consequences on the vertical flux of particulate matter and carbon flux in the ocean.

Monday, November 21, 2016 3:45PM - 3:45PM – Session KP1 Poster Session Exhibit Hall B -

KP1.00001 Numerical Study of Laminar Flow over Acoustic Cavities¹ , MATTHEW OWEN,

Union University, GARY CHENG, University of Alabama — Fluid flow over an open cavity often emits acoustic waves with certain natural frequencies dependent on the geometry of the cavity and the properties and flow conditions of the fluid. Numerical studies of this kind, Computational Aeroacoustics (CAA), pose a grave challenge to the accuracy and efficiency of numerical methods. This project examines the Space-Time Conservation Element Solution Element (CESE) method developed by Dr. S.C. Chang at NASA GRC and compares numerical results of two-dimensional flow to previous experimental data found in literature. The conclusion the project reached is that the test data agrees well with one of the modes of the predicted frequencies, and that further testing is needed to be able to match experimental results.

¹Funding from NSF REU site grant EEC 1358991 is greatly appreciated.

KP1.00002 Implementation of CFD to Control Active and Passive Airfoil Propulsion¹ , JAY YOUNG, DANIEL ASSELIN, CHARLES WILLIAMSON, Cornell University — The fluid dynamics of biologically-inspired flapping propulsion provides a fertile testing ground for the field of unsteady aerodynamics, serving as important groundwork for the design and development of fast, mobile underwater vehicles and flapping-wing micro air vehicles (MAVs). There has been a recent surge of interest in these technologies as they provide low cost, compact, and maneuverable means for terrain mapping, search and rescue operations, and reconnaissance. Propulsion by unsteady motions has been fundamentally modeled with an airfoil that heaves and pitches, and previous work has been done to show that actively controlling these motions can generate high thrust and efficiency (Read, Hover & Triantafyllou 2003). In this study, we examine the performance of an airfoil with an actuated heave motion coupled with a passively controlled pitch motion created by simulating the presence of a torsional spring using our cyber-physical fluid dynamics (CPFD) approach (Mackowski & Williamson 2011, 2015, 2016). By using passively controlled pitch, we have effectively eliminated an actuator, decreasing cost and mass, an important step for developing efficient vehicles. In many cases, we have achieved comparable or superior thrust and efficiency values to those obtained using two actively controlled degrees of freedom.

¹This work was supported by the National Science Foundation and the Air Force Office of Scientific Research Grant No. FA9550-15-1-0243, monitored by Dr. Douglas Smith.

KP1.00003 Transient Performance of a Vertical Axis Wind Turbine¹ , AYKUT ONOL, SERHAT YESILYURT, Sabanci University — A coupled CFD/rotor dynamics modeling approach is presented for the analysis of realistic transient behavior of a height-normalized, three-straight-bladed VAWT subject to inertial effects of the rotor and generator load which is manipulated by a feedback control under standardized wind gusts. The model employs the k- ε turbulence model to approximate unsteady Reynolds-averaged Navier-Stokes equations and is validated with data from field measurements. As distinct from related studies, here, the angular velocity is calculated from the rotor's equation of motion; thus, the dynamic response of the rotor is taken into account. Results include the following: First, the rotor's inertia filters large amplitude oscillations in the wind torque owing to the first-order dynamics. Second, the generator and wind torques differ especially during wind transients subject to the conservation of angular momentum of the rotor. Third, oscillations of the power coefficient exceed the Betz limit temporarily due to the energy storage in the rotor, which acts as a temporary buffer that stores the kinetic energy like a flywheel in short durations. Last, average of transient power coefficients peaks at a smaller tip-speed ratio for wind gusts than steady winds.

¹This work was supported by the Sabanci University Internal Research Grant Program (SU-IRG-985).

KP1.00004 Hydrodynamic Disturbances Affect Self-excited Vibrations of Seal Vibrisseae . , CHRISTIN MURPHY, WILLIAM MARTIN, AREN HELLUM , CHARLES HENOCH, Naval Undersea Warfare Center Division Newport — Seals use their vibrisseae, or whiskers, to find prey by hydrodynamic sensing and tracking. As the whiskers move through the water, self-excited vibrations are induced. We hypothesize that the features of these vibrations encode information about the disturbance source. We used laser Doppler vibrometry to study these vibrations in harbor seal whiskers exposed to water flow in a water tunnel with and without an upstream disturbance present, and at various speeds. Whiskers have an elliptical cross-sectional profile, which creates a distinct effect of angle of attack. To examine this effect, experiments were performed with the major axis of the elliptical profile parallel to the flow (0 degrees), and perpendicular to the flow (90 degrees). For the 0 case without a disturbance, the prominent vibration frequency increases as speed increases. When a disturbance is introduced, there is a clear disruption in the prominent vibration frequency. For the 90 case with no disturbance, multiple distinct vibration frequencies are excited. With increase in speed, the relative amplitudes of the excited vibrations change with respect to each other. When a disturbance is introduced, the excited vibration frequencies persist while the energy increases across the computed spectrum.

KP1.00005 Experimental investigation of flow-structure interaction between a model biofilm streamer and water flow , FARZAN KAZEMIFAR, GIANLUCA BLOIS, University of Notre Dame, SUMIT SINHA, RICHARD HARDY, Durham University, JAMES BEST, University of Illinois, GREGORY SAMBROOK SMITH, University of Birmingham, KENNETH CHRISTENSEN, University of Notre Dame — Biofilms are permeable and deformable material whose bulk structure is composed of extracellular polymeric substance (EPS) that houses bacterial colonies. The EPS is responsible for the mechanical properties of the biofilm. In this study we investigate the fluid-structure interaction between a model biofilm streamer and water flow in a closed-loop water channel in the laminar and transitional flow regimes, using the particle image velocimetry (PIV) technique. The model streamer is fabricated from acrylamide polymer hydrogel. The purpose for using this material is twofold: 1) its mechanical properties (i.e. elastic modulus) can be tuned by controlling its chemical composition, 2) the hydrogel is transparent with a refractive index (RI) very close to that of water, thus minimizing the optical distortions for flow visualization. The velocity vector fields obtained from PIV measurements are used to investigate the temporal evolution of the flow structure in the vicinity of the streamer, focusing on the vortex shedding mechanism and the resulting oscillations of the streamer.

KP1.00006 Determining the benefits of Vorticella cell body motion., MATTY C. SPECHT, RACHEL E. PEPPER, University of Puget Sound — Microscopic sessile suspension feeders are single-celled organisms found in aquatic ecosystems. They live attached to underwater surfaces and create a fluid flow in order to feed on bacteria and debris. They participate in the natural degradation of contaminants in water. Understanding the fluid flow they create enhances our knowledge of their environmental impact. One type of suspension feeder, *Vorticella*, have been observed to vary their cell body orientation with respect to their surface, but the benefits of this motion are still unknown. We use simulations to investigate the effect of *Vorticella* body motion on the feeding current and the nutrient flux to the cell body to determine whether or not the motion increases nutrient consumption. We determine the nutrient flux using COMSOL Multiphysics software to solve the advection-diffusion equation with the flow given by a stokeslet model. We use a range of motions similar and dissimilar to that of live *Vorticella*. We find that most patterns of motion do not increase the nutrient flux, since the *Vorticella* feed from regions where they already have depleted the water of nutrients. However, it is possible that their motion could help the *Vorticella* find nutrients that are inhomogeneously distributed in water.

KP1.00007 Effects of red blood cell deformability on the non-uniform distribution of platelet-sized particles in blood flow through microchannels, MASAKO SUGIHARA-SEKI, KEISUKE SAKAMOTO, TOMOAKI ITANO, JUNJI SEKI, Kansai University — In blood flow through microvessels, platelets are known to have enhanced concentrations near the vessel wall, which is the so-called near-wall excess (NWE). This phenomenon is considered to be caused by the interaction with red blood cells (RBCs); since RBCs have a tendency to approach the vessel centerline due to their highly deformability, they may push away less deformable platelets toward the near-wall region. In order to examine this proposition by in vitro experiments, we measured the distribution of platelet-sized particles mixed in intact RBC or hardened RBC suspensions flowing through microchannels of $50\mu\text{m} \times 50\mu\text{m}$ cross-section. Hardened RBCs were prepared by immersing human RBCs in glutaraldehyde solution of 40 – 4000 ppm. Fluorescent observations were conducted with the use of a confocal laser scanning microscope system with a high-speed video camera. It was found that platelet-sized particles exhibited high concentrations near the channel wall, i.e., NWE, when they were mixed in intact RBC suspensions. By contrast, the particles mixed in hardened RBC suspensions showed weak NWE or uniform distribution over the channel cross-section, indicating that deformability of RBCs plays an essential role in the NWE phenomenon.

KP1.00008 On the Normal Force Mechanotransduction of Human Umbilical Vein Endothelial Cells¹, AMIR VAHABIKASHI, QIUYUN WANG, JAMES WILSON, QIANHONG WU, Villanova University, VUCBMSS TEAM — In this paper, we report a cellular biomechanics study to examine the normal force mechanotransduction of Human Umbilical Vein Endothelial Cells (HUEVCs) with their implications on hypertension. Endothelial cells sense mechanical forces and adjust their structure and function accordingly. The mechanotransduction of normal forces plays a vital role in hypertension due to the higher pressure buildup inside blood vessels. Herein, HUEVCs were cultured to full confluence and then exposed to different mechanical loadings using a novel microfluidic flow chamber. One various pressure levels while keeps the shear stress constant inside the flow chamber. Three groups of cells were examined, the control group (neither shear nor normal stresses), the normal pressure group (10 dyne/cm² of shear stress and 95 mmHg of pressure), and the hypertensive group (10 dyne/cm² of shear stress and 142 mmHg of pressure). Cellular response characterized by RT-PCR method indicates that, COX-2 expressed under normal pressure but not high pressure; Mn-SOD expressed under both normal and high pressure while this response was stronger for normal pressure; FOS and e-NOS did not respond under any condition. The differential behavior of COX-2 and Mn-SOD in response to changes in pressure, is instrumental for better understanding the pathogenesis of hypertensive cardiovascular diseases.

¹This research was supported by the National Science Foundation under Award 1511096.

KP1.00009 Investigation of the Left Ventricular Flow Dynamics in the Presence of Severe Mitral Annular Calcification, BATOUL EL - SAYEGH, LYES KADEM, GIUSEPPE DI LABBIO, Concordia Univ, GREGG S. PRESSMAN, EDINRIN OBASARE, Albert Einstein Medical Center — Valvular calcification is frequent with aging and diverse diseases. Mitral annular calcification (MAC) is a degenerative process where the fibrous annulus of the mitral valve degrades. MAC can be found in approximately 40% of people aged over 65. It is associated with increased occurrence of cardiovascular diseases including stroke. This experimental work is aimed to investigate the effects of MAC on the left ventricle (LV) hemodynamics and to develop new clinical parameters. Two patient-specific 3D-printed mitral valves with moderate and severe MACs were placed in a left heart simulator. The velocity fields in the LV were acquired using time-resolved particle image velocimetry (TR-PIV) and compared to normal LV flow. The velocity fields were used to evaluate the temporal evolution of the vorticity fields and viscous energy loss in the LV. The presence of MAC disturbed the flow in the LV leading to markedly increased viscous energy losses. As the severity of MAC increased, the velocity of the inflow jet also increased causing significant perturbations to the normally-occurring vortex in the LV.

KP1.00010 Flow Topology in the Right Ventricle after Tetralogy of Fallot Repair, AMANDA MIKHAIL, LYES KADEM, GIUSEPPE DI LABBIO, Concordia Univ — Among all of the known congenital heart defects, Tetralogy of Fallot (TOF) is the most common cyanotic defect, accounting for 5% of all detected defects. Approximately 1 in 2518 births will result with TOF, leading to about 1657 cases per year in the United States alone. All of those affected will need surgical repair in order to have a relatively normal life and longer life span. Unfortunately, pulmonary regurgitation (PR) has been observed to appear two to three decades after the initial operation in 50% of operated cases. PR results in abnormal flow patterns in the right ventricle, which are currently poorly understood. In this experimental study, several severities of pulmonary regurgitation were simulated on a newly developed right ventricle using a cardiovascular simulator. The interaction between the tricuspid valve inflow and the pulmonary regurgitation was investigated using Time-resolved particle image velocimetry (TR-PIV). PR resulted in a turbulent jet that disturbed the optimal filling of the RV. Energy losses and viscous shear stresses were observed to significantly increase with the severity of PR. This study can contribute towards a better understanding of the suboptimal performance in patients with repaired TOF.

KP1.00011 In vitro evaluation of valve hemodynamics in the pediatric pulmonary outflow tract, NICOLE SCHIAVONE, CHRIS ELKINS, DOFF MCELHINNEY, JOHN EATON, ALISON MARSDEN, Stanford University — Tetralogy of Fallot (ToF) is a congenital heart disease that affects 1 in every 2500 newborns each year and requires surgical repair of the right ventricular outflow tract (RVOT) and subsequent placement of an artificial pulmonary valve. While a wide variety of artificial valves are available, essentially all of them become subject to degradation and dysfunction during the patient's lifetime, which leads to additional interventions. However, there is little understanding about the mechanical function of replacement pulmonary valves and no quantitative placement guidelines to ensure maximum failure-free lifetime. This work aims to experimentally assess the biomechanics of pulmonary valves in realistic RVOT geometries using magnetic resonance velocimetry (MRV), which can measure 3D, three-component phase-averaged velocity fields. The RVOT geometries are constructed using 3D printing, allowing for variation in crucial geometric parameters such as the radius of curvature of the main pulmonary artery (MPA) and the dilation of the artery downstream of the valve. A St. Jude Medical Epic valve is secured inside the RVOT geometry and can be interchanged, allowing for variation of the ratio between valve diameter and MPA diameter. This work will discuss the use of MRV to capture the flow structure in the RVOT and evaluate pulmonary valve performance under different conditions.

KP1.00012 Intraventricular filling in physical models of the left ventricle: influence of aortic pressure, NICHOLAS NELSEN, MILAD SAMAE, ARVIND SANTHANAKRISHNAN, Oklahoma State University — Clinical studies using medical imaging have provided evidence on the formation of an intraventricular vortex in the left ventricle (LV) during diastolic phase of the cardiac cycle. However, the question of how the vortex characteristics are altered with aortic pressure remains unclear. This is of relevance to hypertensive heart disease and heart failure with normal ejection fraction. Using an experimental left heart simulator, we have previously shown that increasing LV wall stiffness results in reduction of the filling vortex circulation. In this study, we investigate the effects of varying aortic pressure in addition to wall stiffness. A series of flexible-walled LV models with varying wall stiffness were tested in a pulsatile flow loop. 2D particle image velocimetry was used to visualize intraventricular flow and calculate filling vortex circulation. The flow circuit was first setup with the least stiffness LV physical model, and tuned to physiological aortic pressure, cardiac output and ejection fraction. We then iteratively tested LV models with increasing stiffness without changing circuit variables. Comparisons of the filling vortex circulation with changing aortic pressure relative to the baseline and increased LV stiffness models will be presented.

KP1.00013 Effect of the bifurcation angle on the flow within a synthetic model of lower human airways¹, ANDRES SANTIAGO ESPINOSA MORENO, CARLOS ALBERTO DUQUE DAZA, Universidad Nacional de Colombia — The effect of the bifurcation angle on the flow pattern developed during respiratory inhalation and exhalation processes was explored numerically using a synthetic model of lower human airways featuring three generations of a dichotomous morphology as described by a Weibel model. Laminar flow simulations were performed for six bifurcation angles and four Reynolds numbers relevant to human respiratory flow. Numerical results of the inhalation process showed a peak displacement trend of the velocity profile towards the inner walls of the model. This displacement exhibited correlation with Dean-type secondary flow patterns, as well as with the onset and location of vortices. High wall shear stress regions on the inner walls were observed for a range of bifurcation angles. Noteworthy, specific bifurcation angles produced higher values of pressure drop, compared to the average behavior, as well as changes in the volumetric flow through the branches. Results of the simulations for exhalation process showed a different picture, mainly the appearance of symmetrical velocity profiles and the change of location of the regions of high wall shear stress. The use of this modelling methodology for biomedical applications is discussed considering the validity of the obtained results.

¹Department of Mechanical and Mechatronics Engineering, Universidad Nacional de Colombia

KP1.00014 Large Eddy Simulation of turbulent-like flow in intracranial aneurysms, MUHAMMAD OWAIS KHAN, CHRISTOPHE CHNAFA, DAVID A. STEINMAN, University of Toronto, SIMON MENDEZ, FRANCK NICLOUD, IMAG, University of Montpellier — Hemodynamic forces are thought to contribute to pathogenesis and rupture of intracranial aneurysms (IA). Recent high-resolution patient-specific computational fluid dynamics (CFD) simulations have highlighted the presence of turbulent-like flow features, characterized by transient high-frequency flow instabilities. In-vitro studies have shown that such turbulent-like flows can lead to lack of endothelial cell orientation and cell depletion, and thus, may also have relevance to IA rupture risk assessment. From a modelling perspective, previous studies have relied on DNS to resolve the small-scale structures in these flows. While accurate, DNS is clinically infeasible due to high computational cost and long simulation times. In this study, we present the applicability of LES for IAs using a LES/blood flow dedicated solver (YALES2BIO) and compare against respective DNS. As a qualitative analysis, we compute time-averaged WSS and OSI maps, as well as, novel frequency-based WSS indices. As a quantitative analysis, we show the differences in POD eigenspectra for LES vs. DNS and wavelet analysis of intra-saccular velocity traces. Differences in two SGS models (i.e. Dynamic Smagorinsky vs. Sigma) are also compared against DNS, and computational gains of LES are discussed.

KP1.00015 A critical velocity of squeezing a droplet through a circular constriction: implications on ischemic stroke¹, ZHIFENG ZHANG, CORINA DRAPACA, The Penn State University — Ischemic stroke accounts for about 87 percent of all stroke cases. In these cases, models of squeezing a droplet through a smaller constriction channel can help better understand the pathology and capillary restoring after a Stroke. In the present research, we analytical expressed the minimum impulse of squeezing a droplet through a circular channel as well as its critical velocity. By comparison with a previously defined critical velocity, we find the difference between these two. Applications of this research in the understanding of ischemic stroke are also discussed.

¹Zhifeng Zhang thanks the support of Robert A. Sebrosky Graduate Fellowship in Engineering Science and Mechanics, the Pennsylvania State University.

KP1.00016 Swimming performance of biomimetic trapezoidal elastic fins, MICHAEL SPADARO, PETER YEH, ALEXANDER ALEXEEV, Woodruff School of Mechanical Engineering, Georgia Institute of Technology — Using three-dimensional computer simulations, we probe the biomimetic free-swimming of trapezoidal elastic plates plunging sinusoidally in a viscous fluid, varying the frequency of oscillations and plate geometry. We choose the elastic trapezoidal plate geometry because it more closely approximates the shape of real caudal fish fins. Indeed, caudal fins are found in nature in a variety of trapezoidal shapes with different aspect ratios. Because of this, we perform our simulations using plates with aspect ratios varying from the cases where the plate has a longer leading edge and to plates with a longer trailing edge. We find that the trapezoidal fins with the longer trailing edge are less efficient than the rectangular fins at the equivalent oscillation frequencies. This is surprising because many fish found in nature have a widening tail. We relate this to the fact that our model considers fins with uniform thickness whereas fish uses tapered fins. Our results will be useful for the design of biomimetic swimming devices as well as understanding more closely the physics of fish swimming.

KP1.00017 Mechanical krill models for studying coordinated swimming, ALICE MONTAGUE, HONG KUAN LAI, MILAD SAMAE, ARVIND SANTHANAKRISHNAN, Oklahoma State University — The global biomass of *Homo sapiens* is about a third of the biomass of *Euphausia superba*, commonly known as the Antarctic krill. Krill participate in organized social behavior. Propulsive jets generated by individual krill in a school have been suggested to be important in providing hydrodynamic sensory cues. The importance of body positions and body angles on the wakes generated is challenging to study in free swimming krill. Our solution to study the flow fields of multiple krill was to develop mechanical krill robots. We designed krillbots using mostly 3D printed parts that are actuated by stepper motors. The krillbot limb lengths, angles, inter-limb spacing and pleopod stroke frequency were dynamically scaled using published data on free-swimming krill kinematics. The vertical and horizontal spacing between krillbots, as well as the body angle, are adjustable. In this study, we conducted particle image velocimetry (PIV) measurements with two tethered krillbots in a flow tank with no background flow. One krillbot was placed above and behind the other. Both krillbots were at a zero-degree body angle. Wake-body interactions visualized from PIV data will be presented.

KP1.00018 Impact of Morphological Changes on the M

SUNITHA SHROFF N, Mount Carmel College, Bangalore — Bio-mechanical properties of cell membrane on the cell locomotion. To explore, morphological changes were induced in *Amoeba proteus* by cholinergic or through KCl mediated membrane depolarization. We observed that, membrane depolarization of pseudopodia in a dose dependent manner, gradually *A. proteus* becomes globular. We also report that (Chilomonas) *A. proteus* transforms them into tube/filament like structure and this transformation may involve signaling mechanisms. Further, we carried out immunostaining of *A. proteus* with P2X2 receptor localization and the extent of expression. The result indicated that in normal *A. proteus* receptors shaped *A. proteus* P2X2-receptor was found to be localized, unlike P2Y2 receptor. As nucleotide changes in the organism, we report corresponding changes in their locomotion.

Mount Carmel College, Bangalore 560 052

KP1.00019 Financial Assets [share, bonds] & Aencylia¹ , WH- MAKSOED², Prodi of Physics UI,

Depok 16415- West Java — Instead Elaine Scarry: "Thermonuclear monarchy" reinvent Carry Nation since Aug 17, 1965 the Republic of Indonesia's President speech: "Reach to the Star" , for "cancellation" usually found in External Debt herewith retrieved from "the Window of theWorld": *Aencylia* , feast in March, a month named after Mars, the god of war. "On March 19 they used to put on their biggest performance of gymnastics in order to "bribe" their god for another good year" , further we have vacancy & "vacuum tube"- Bulat Air karena Pembuluh, Bulat Kata karena Mufakat" proverb from Minangkabau, West Sumatra. Follows March 19, 1984 are first prototype flight of IAI Astra Jet as well as March 19, 2012 invoice accompanies Electric car Kujang-193, Fainancial Assets [share, bonds] are the answer for "infrastructure" & state owned enterprises assets to be hedged first initial debt per capita accordances.

¹Heartfelt gratitudes to HE. Mr. Ir. Sarwono Kusumaatmadja/PT. Smartfren INDONESIA

²Instead trades "the syllabic analytic synthetic method", Windows on the World- 1975 also provides "Which New Year's Day do you keep?": by the year 46 BC Emperor Julius Caesar had to add 2 months APS accomplishes the Twelve Apostle

KP1.00020 Numerical simulation of ultrasound-induced dynamics of a gas bubble neighboring a rigid wall , TATSUYA KOBAYASHI, KEITA ANDO, Department of Mechanical Engineering, Keio University —

Cavitation erosion has been a technical issue in ultrasonic cleaning under which cavitation bubbles appear near target surfaces to be cleaned. In the present study, we numerically study the interaction of ultrasonic standing waves with a gas bubble in the neighborhood of a rigid wall. We solve multicomponent Euler equations that ignore surface tension and phase change at interfaces, by the finite-volume WENO scheme with interface capturing. The pressure amplitude of the ultrasound is set at several atmospheres and the ultrasound wavelength is tuned to obtain the situation near resonance. In the simulation, we observe jetting flow toward the rigid wall at violent bubble collapse that may explain cavitation erosion in ultrasonic cleaning.

KP1.00021 Investigating short-time dynamics of spreading bubbles¹ , MATTHIEU LAURENT, MARK MENESSES, JAMES BIRD, Boston University —

When a bubble comes into contact with a partially wetting surface, the film between the bubble and solid surface rapidly dewets to minimize the free energy of the system. The dynamics of this dewetting is assumed to be dominated by capillary and viscous effects. Yet, when drops rather than bubbles spread, the short-time dynamics are dominated by a balance of capillarity and inertia. Here we revisit spreading bubbles to investigate whether the short-time dynamics is better captured by a viscous or inertial scaling. Counter-intuitively, neither viscous nor inertial effects alone can account for short-time spreading dynamics. Through an experimental approach, we develop a dimensionless scaling relation — incorporating both viscosity and inertia — that successfully collapses the data.

¹Chaire X-ESPCI-Saint Gobain

KP1.00022 High-speed visualization and radiated pressure measurement of a laser-induced gas bubble in glycerin-water solutions , TAKEHIRO NAKAJIMA, TOMOKI KONDO, KEITA ANDO, Department of Mechanical Engineering, Keio University —

We study the dynamics of a spherical gaseous bubble created by focusing a nanosecond laser pulse at 532 nm into a large volume of glycerin-water solutions. Free oscillation of the bubble and shock wave emission from the bubble dynamics are recorded by a high-speed camera together with a pulse laser stroboscope; concurrently, pressure radiated from the oscillating bubble is measured by a hydrophone. The bubble achieves a mechanical equilibrium after free oscillation is damped out; the equilibrium state stays for a while, unlike vapor bubbles. We speculate that the bubble content is mainly gases originally dissolved in the liquid (i.e., air). The bubble dynamics we observed are compared to Rayleigh-Plesset-type calculations that account for diffusive effects; the (unknown) initial pressure just after laser focusing is tuned to obtain agreement between the experiment and the calculation. Moreover, viscous effects on the shock propagation are examined with the aid of compressible Navier-Stokes simulation.

KP1.00023 Diffusion-driven growth of a spherical gas bubble in gelatin gels supersaturated with air¹ , ERIKO SHIROTA, KEITA ANDO, Department of Mechanical Engineering, Keio University —

We experimentally and theoretically study diffusion-driven growth of laser-induced gas bubbles in gelatin gels supersaturated with air. The supersaturation in the gels is realized by using a large separation between heat and mass diffusion rates. An optical system is developed to induce bubble nucleation by laser focusing and visualize the subsequent bubble growth. To evaluate the effect of the gel elasticity on the bubble growth rate, we propose the extended Epstein-Plesset theory that considers bubble pressure modifications due to linear/nonlinear elasticity (in addition to Laplace pressure). From comparisons between the experiments and the proposed theory, the bubble growth rate is found to be hindered by the elasticity.

¹This study is supported by JSPS KAKENHI Grant Number 25709008.

KP1.00024 Epstein-Plesset theory based measurements of concentration of nitrogen gases dissolved in aerated water, MASASHI SASAKI, TATSUYA YAMASHITA, KEITA ANDO, Department of Mechanical Engineering, Keio University — Microbubble aeration is used to dissolved gases into water and is an important technique in agriculture and industry. We can measure concentration of dissolved oxygen (DO) in aerated water by commercial DO meters. However, there do not exist commercially available techniques to measure concentration to dissolved nitrogen (DN). In the present study, we propose the method to measure DN in aerated water with the aid of Epstein-Plesset-type analysis. Gas-supersaturated tap water is produced by applying aeration with micro-sized air bubbles and is then stored in a glass container open to the atmosphere. Diffusion-driven growth of bubbles nucleated at the container surface is recorded with a video camera. The bubble growth rate is compared to the extended Epstein-Plesset theory that models mass transfer of both DO and DN into the surface-attached bubbles base on the diffusion equation. Given the DO measurements, we can obtain the DN level by fitting in the comparison.

KP1.00025 Shock Refraction at Semi-Rigid Interfaces, GABRIELLE MILLER, JAMES REYNOLDS,

Sandia National Laboratories — We consider a strong spherical air shock encountering a planar interface separating the air from a medium of significantly higher impedance, with the goal of obtaining an approximate analytic description. Before encountering the interface, the incident air shock is well described by the Taylor-Sedov solution for a point blast. The behavior of the reflected and transmitted shocks differs depending upon the height of burst. For moderate heights, despite the relatively small amount of energy transferred, the pressure in the second medium may be much higher than that behind the air shock due to the strong impedance mismatch [1]. Near-surface blasts may be further complicated by the entrainment of material from the second medium and/or the deflection of the interface caused by the strong air shock. For the present study, we ignore the effects of entrainment and assume that the deformation of the interface is small compared to the height of burst. We then investigate the relationship between energy loss into the second medium and the reflected air shock. [1] L. Henderson, M. Jia-Huan, S. Akira, T. Kazuyoshi, Fluid Dynamics Research 5(5-6), 337 (1990)

KP1.00026 Inclusion of Separation in Integral Boundary Layer Methods¹, BRODIE WAL-

LACE, CHARLES O'NEILL, The University of Alabama — An integral boundary layer (IBL) method coupled with a potential flow solver quickly allows simulating aerodynamic flows, allowing for aircraft geometries to be rapidly designed and optimized. However, most current IBL methods lack the ability to accurately model three-dimensional separated flows. Various IBL equations and closure relations were investigated in an effort to develop an IBL capable of modeling separation. Solution techniques, including a Newton's method and the inverse matrix solving program GMRES, as well as methods for coupling an IBL with a potential flow solver were also investigated. Results for two-dimensional attached flow as well as methods for expanding an IBL to model three-dimensional separation are presented.

¹Funding from NSF REU site grant EEC 1358991 is greatly appreciated

KP1.00027 Validation of Magnetic Resonance Thermometry by Computational Fluid Dynamics, GRANT RYDQUIST, MARK OWKES, Montana State University, CLAIRE M. VERHULST, MICHAEL J. BENSON, BRETT P. VANPOPPEL, United States Military Academy, West Point, SASCHA BURTON, JOHN K. EATON, CHRISTOPHER P. ELKINS, Stanford University — Magnetic Resonance Thermometry (MRT) is a new experimental technique that can create fully three-dimensional temperature fields in a noninvasive manner. However, validation is still required to determine the accuracy of measured results. One method of examination is to compare data gathered experimentally to data computed with computational fluid dynamics (CFD). In this study, large-eddy simulations have been performed with the NGA computational platform to generate data for a comparison with previously run MRT experiments. The experimental setup consisted of a heated jet inclined at 30° injected into a larger channel. In the simulations, viscosity and density were scaled according to the local temperature to account for differences in buoyant and viscous forces. A mesh-independent study was performed with 5 mil-, 15 mil- and 45 mil-cell meshes. The program Star-CCM+ was used to simulate the complete experimental geometry. This was compared to data generated from NGA. Overall, both programs show good agreement with the experimental data gathered with MRT. With this data, the validity of MRT as a diagnostic tool has been shown and the tool can be used to further our understanding of a range of flows with non-trivial temperature distributions.

KP1.00028 Statistical Inference of a RANS closure for a Jet-in-Crossflow simulation¹, JAN HEYSE, WOUTER EDELING, GIANLUCA IACCARINO, Stanford Univ — The jet-in-crossflow is found in several engineering applications, such as discrete film cooling for turbine blades, where a coolant injected through holes in the blade's surface protects the component from the hot gases leaving the combustion chamber. Experimental measurements using MRI techniques have been completed for a single hole injection into a turbulent crossflow, providing full 3D averaged velocity field. For such flows of engineering interest, Reynolds-Averaged Navier-Stokes (RANS) turbulence closure models are often the only viable computational option. However, RANS models are known to provide poor predictions in the region close to the injection point. Since these models are calibrated on simple canonical flow problems, the obtained closure coefficient estimates are unlikely to extrapolate well to more complex flows. We will therefore calibrate the parameters of a RANS model using statistical inference techniques informed by the experimental jet-in-crossflow data. The obtained probabilistic parameter estimates can in turn be used to compute flow fields with quantified uncertainty.

¹Stanford Graduate Fellowship in Science and Engineering

KP1.00029 A numerical study of natural convection in eccentric spherical annuli¹,

ANGEL GALLEGOS, Utrecht Institute for Theoretical Physics. Universiteit Utrecht. Master's Program., CARLOS MALAGA, Physics Department. School of Science. Universidad Nacional Autonoma de Mexico — A fluid between two spheres, concentric or not, at different temperatures will flow in the presence of a constant gravitational force. Although there is no possible hydrostatic state, energy transport is dominated by diffusion if temperature difference between the spheres is small enough. By the use of a full three-dimensional thermal lattice Boltzmann model we study the transition between the conductive, the steady convective, and the unsteady convective regimes. We use the concentric case to validate the results by comparing with experiments and numerical simulations found in the literature, and then we extend our numerical experiments to the eccentric case to observe the general behavior of the different regimes. We analyze the energy transport characterized by the relation between Nusselt and Rayleigh numbers as well as the arising flow patterns.

¹This work was partially supported by UNAM-DGAPA-PAPIIT grant number IN115216

KP1.00030 ABSTRACT WITHDRAWN —

KP1.00031 The effect of temperature on the impact process of SiO₂ particle onto a planar surface, MING DONG, Associate Professor, SUFEN LI, Professor, YAN SHANG, Doctor — This paper presents the results of a comprehensive program of experiments in which SiO₂ particles were impacted under controlled conditions against a planar steel surface. The overall aim of these experiments was to gain an understanding of the ash deposition process in a pulverized coal boiler system. A continuous nitrogen flow carrying particles was used to simulate the flue gas in boiler, and planer steel surface was used to simulate the heat transfer tube in boiler. The effect of particle incident velocity, particle temperature and planar surface temperature on the normal restitution coefficient was examined. The results show that the normal restitution coefficient increases firstly with increasing incident velocity, and then decreases with increasing incident velocity in the measurement range (ranging from 8m/s to 13m/s). The normal restitution coefficient decreases with increasing particle temperature and surface temperature, and with temperature difference between particle and surface. The experiments are carried out in an atmospheric column, and individual impacts are recorded by a digital camera system. Keywords: Normal restitution coefficient, impaction experiments, particle, rebound characteristics.

KP1.00032 Peruvian perovskite Between Transition-metal to PGM/PlatinumGroupMetal Catalytic Fusion¹, WH- MAKSOED, PT. DAYA SADHANA BHAKTI, tbk, Jl. Kiaradong 204, Bandung 40274-West Java — Strongly correlated electronic materials made of simple building blocks, such as a transition-metal ion in an octahedral oxygen cage forming a perovskite structure- Dagotto & Tokura for examples are the high-temperature superconductivity & the CMR/Colossal Magnetoresistance . Helium-4 denotes from LC Case,ScD: "Catalytic Fusion of Deuterium into Helium-4"- 1998 dealt with gaseous D₂ – "contacted with a supported metallic catalyst at superatmospheric pressure". The catalyst is a platinum-group metal, at about 0.5 % - 1 % by weight, on activated C. Accompanies Stephen J Geier, 2010 quotes "transition metal complexes", the Energy thus produced is enormous, and because the deuterium is very cheap in the form of heavy water (less than US \$ 1/g), the fuel cost is very low (<<1 %/KwH). "The oceans contain enough deuterium to satisfy the Earth's energy needs for many millions of year" to keep "maria"/Latin name of seas &Deuteronomy to be eternally preserves.

¹Heartfelt Gratuities to HE. Mr. Prof. Ir. HANDOJO

KP1.00033 Design Considerations of a Solid State Thermal Energy Storage¹, MO-HAMMAD JANBOZORGHI, SAMMY HOUSAINY, ARIANA THACKER, PEGGY IP, WALID ISMAIL, PIROUZ KAVEHPOUR, University of California, Los Angeles — With the growing governmental restrictions on carbon emission, renewable energies are becoming more prevalent. A reliable use of a renewable source however requires a built-in storage to overcome the inherent intermittent nature of the available energy. Thermal design of a solid state energy storage has been investigated for optimal performance. The impact of flow regime, laminar vs. turbulent, on the design and sizing of the system is also studied. The implications of low thermal conductivity of the storage material are discussed and a design that maximizes the round trip efficiency is presented.

¹This study was supported by award No. EPC-14-027 granted by California Energy Commission (CEC)

KP1.00034 Ghost Particle Velocimetry implementation in millimeters devices and comparison with μ PIV, MARCO RICCOMI, University of Pisa, FEDERICO ALBERINI, University of Birmingham, ELISABETTA BRUNAZZI, University of Pisa, DANIELE VIGOLO, University of Birmingham — Micro/milli-fluidic devices are becoming an important reference for several disciplines and are quickly increasing their applications in scientific, as well as industrial, environment. As a consequence, the development of techniques able to analyse these kinds of systems is required to allow their progress. Here we show the implementation of the Ghost Particle Velocimetry (GPV) for the flow velocity field investigation in milli-fluidic devices. This innovative technique has been recently introduced, and has been already proven to be useful in describing rapid phenomenon at a small scale. In this work, the GPV has been used to characterize the trapping of light suspended material in a branching junction. Experiments have been performed to identify the flow velocity field close to a millimeters scale T-junction, at different Reynolds numbers. Particularly interesting are the complex structures, such as vortices and recirculation zones, induced by the vortex breakdown phenomenon. The results obtained have been deeply validated and compared with the well-established μ PIV, highlighting the differences in terms of qualitative and quantitative parameters. A performance comparison has been designed to underline the strengths and weaknesses of the two experimental techniques.

KP1.00035 Capillary rise in crumpled-sheets of paper, AYAX HERNANDO TORRES VICTORIA, MOISS SALGADO, SALOMN PERALTA, Instituto Politcnico Nacional SEPI ESIME Azcapotzalco, FRANCISO WONG, Instituto Mexicano del Petleo, ABRAHAM MEDINA, Instituto Politcnico Nacional SEPI ESIME Azcapotzalco — In this work we report experiments on the capillary rise of water into crumpled paper, in order to understand how the controlled damage of a soft material, like paper (hand-crumpled paper sheets), improves their capabilities of liquid sorption. We have done a series of experiments where a different number of crumples (from zero up to fifty) were made on different rectangular paper pieces and we found that an increasing number of crumples enhances such a capability. Characteristic power laws for the front of elevation, h, versus the elapsed time to reach such height, t, are reported.

KP1.00036 Effect of Viscosity on Liquid Curtain Stability¹, ALIREZA MOHAMMAD KARIM, WIESLAW SUSZYNSKI, LORRAINE FRANCIS, University of Minnesota, Twin Cities, MARCIO CARVALHO, PUC Rio, DOW CHEMICAL COMPANY COLLABORATION, PUC RIO COLLABORATION, UNIVERSITY OF MINNESOTA, TWIN CITIES COLLABORATION — The effect of viscosity on the stability of Newtonian liquid curtains was explored by high-speed visualization. Glycerol/water solutions with viscosity ranging from 19.1 to 210 mPa.s were used as coating liquids. The experimental set-up used a slide die delivery and steel tube edge guides. The velocity along curtain at different positions was measured by tracking small particles at different flow conditions. The measurements revealed that away from edge guides, velocity is well described by free fall effect. However, close to edge guides, liquid moves slower, revealing formation of a viscous boundary layer. The size of boundary layer and velocity near edge guides are strong function of viscosity. The critical condition was determined by examining flow rate below which curtain broke. Curtain failure was initiated by growth of a hole within liquid curtain, close to edge guides. Visualization results showed that the hole forms in a circular shape then becomes elliptical as it grows faster in vertical direction compared to horizontal direction. As viscosity rises, minimum flow rate for destabilization of curtain increased, indicating connection between interaction with edge guides and curtain stability.

¹We would like to acknowledge the financial support from the Dow Chemical Company.

KP1.00037 Flow and Acoustic Features of a Mach 0.9 Free Jet Using High-Frequency Excitation¹, PUJA UPADHYAY, FARRUKH ALVI, Florida State University — This study focuses on active control of a Mach 0.9 ($Re_D = 6 \times 10^5$) free jet using high-frequency excitation for noise reduction. Eight resonance-enhanced microjet actuators with nominal frequencies of 25 kHz ($St_D \approx 2.2$) are used to excite the shear layer at frequencies that are approximately an order of magnitude higher than the jet preferred frequency. The influence of control on mean and turbulent characteristics of the jet is studied using Particle Image Velocimetry. Additionally, far-field acoustic measurements are acquired to estimate the effect of pulsed injection on noise characteristics of the jet. Flow field measurements revealed that strong streamwise vortex pairs, formed as a result of control, result in a significantly thicker initial shear layer. This excited shear layer is also prominently undulated, resulting in a modified initial velocity profile. Also, the distribution of turbulent kinetic energy revealed that forcing results in increased turbulence levels for near-injection regions, followed by a global reduction for all downstream locations. Far-field acoustic measurements showed noise reductions at low to moderate frequencies. Additionally, an increase in high-frequency noise, mostly dominated by the actuators resonant noise, was observed.

¹AFOSR and ARO

KP1.00038 Effect of noise on Rayleigh-Taylor mixing with time-dependent acceleration¹, NORA SWISHER, ARUN PANDIAN, SNEZHANA ABARZHI, Carnegie Mellon University — We perform a detailed stochastic study of Rayleigh-Taylor (RT) mixing with time-dependent acceleration. A set of nonlinear stochastic differential equations with multiplicative noise is derived on the basis of momentum model and group theory analysis. A broad range of parameters is investigated, and self-similar asymptotic solutions are found. The existence is shown of two sub-regimes of RT mixing dynamics: the acceleration-driven and the dissipation-driven mixing. In each sub-regime, statistic properties of the solutions are investigated, and dynamic invariants are found. Transition between the sub-regimes is studied.

¹The work is supported by the US National Science Foundation

KP1.00039 Effect of pressure fluctuations on Richtmyer-Meshkov coherent structures¹, AKLANT K. BHOWMICK, SNEZHANA ABARZHI, Carnegie Mellon University — We investigate the formation and evolution of Richtmyer Meshkov bubbles after the passage of a shock wave across a two fluid interface in the presence of pressure fluctuations. The fluids are ideal and incompressible and the pressure fluctuations are scale invariant in space and time, and are modeled by a power law time dependent acceleration field with exponent -2. Solutions indicate sensitivity to pressure fluctuations. In the linear regime, the growth of curvature and bubble velocity is linear. The growth rate is dominated by the initial velocity for weak pressure fluctuations, and by the acceleration term for strong pressure fluctuations. In the non-linear regime, the bubble curvature is constant and the solutions form a one parameter family (parametrized by the bubble curvature). The solutions are shown to be convergent and asymptotically stable. The physical solution (stable fastest growing) is a flat bubble for small pressure fluctuations and a curved bubble for large pressure fluctuations. The velocity field (in the frame of references accounting for the background motion) involves intense motion of the fluids in a vicinity of the interface, effectively no motion of the fluids away from the interfaces, and formation of vortical structures at the interface.

¹The work is supported by the US National Science Foundation

KP1.00040 Dimensional crossover in Richtmyer-Meshkov unstable flows in the presence of pressure fluctuations¹, AKLANT K. BHOWMICK, SNEZHANA ABARZHI, Carnegie Mellon University — We analyze Richtmyer-Meshkov unstable interfacial dynamics in the presence of pressure fluctuations. The pressure fluctuations are scale invariant and are modeled by an effective time dependent acceleration field with power law exponent -2. The group theory based analysis is applied to 3D rectangular p2mm, 3D square p4mm and 2D pm1 RM flows. From the symmetry analysis, we find that 3D square and 2D bubbles form a one parameter family and 3D rectangular bubbles form a two parameter family. The families are parametrized by the principal curvature(s). The bubble velocity and Fourier amplitude profiles exhibit RM type behavior for weak accelerations and RT type behavior for strong accelerations. Under the dimensional crossover, the bubbles elongated in one of the directions reduce to the 2D solutions, whereas the bubbles elongated in the other direction flatten. Stability analysis shows that 3D square bubbles are stable with respect to isotropic as well as anisotropic perturbations. 2D bubbles are unstable to 3D perturbations. No continuous transition is possible between 3D square and 2D bubbles and the dimensional crossover is discontinuous for both strong and weak pressure fluctuations.

¹The work is supported by the US National Science Foundation

KP1.00041 Experiments and simulations of Richtmyer-Meshkov Instability with measured volumetric initial conditions., EVEREST SEWELL, KEVIN FERGUSON, JEFFREY JACOBS, The University of Arizona, JEFF GREENOUGH, Lawrence Livermore National Laboratory, VITALIY KRIVETS, The University of Arizona — We describe experiments of single-shock Richtmyer-Meshkov Instability (RMI) performed on the shock tube apparatus at the University of Arizona in which the initial conditions are volumetrically imaged prior to shock wave arrival. Initial perturbations play a major role in the evolution of RMI, and previous experimental efforts only capture a single plane of the initial condition. The method presented uses a rastered laser sheet to capture additional images throughout the depth of the initial condition immediately before the shock arrival time. These images are then used to reconstruct a volumetric approximation of the experimental perturbation. Analysis of the initial perturbations is performed, and then used as initial conditions in simulations using the hydrodynamics code ARES, developed at Lawrence Livermore National Laboratory (LLNL). Experiments are presented and comparisons are made with simulation results.

KP1.00042 Design and Construction of a Shock Tube Experiment for Multiphase Instability Experiments, JOHN MIDDLEBROOKS, Univ of Missouri - Columbia, WOLFGANG BLACK , CONSTANTINE AVOUSTOPOULOS, ROY ALLEN, RAJ KATHAKAPA, QIWEN GUO, JACOB MCFARLAND¹, University of Missouri FLuid Mixing and Shock Tube Laboratory — Hydrodynamic instabilities are important phenomena that have a wide range of practical applications in engineering and physics. One such instability, the shock driven multiphase instability (SDMI), arises when a shockwave accelerates an interface between two particle-gas mixtures with differing multiphase properties. The SDMI is present in high energy explosives, scramjets, and supernovae. A practical way of studying shock wave driven instabilities is through experimentation in a shock tube laboratory. This poster presentation will cover the design and data acquisition process of the University of Missouri's Fluid Mixing Shock Tube Laboratory. In the shock tube, a pressure generated shockwave is passed through a multiphase interface, creating the SDMI instability. This can be photographed for observation using high speed cameras, lasers, and advance imaging techniques. Important experimental parameters such as internal pressure and temperature, and mass flow rates of gases can be set and recorded by remotely controlled devices. The experimental facility provides the University of Missouri's Fluid Mixing Shock Tube Laboratory with the ability to validate simulated experiments and to conduct further inquiry into the field of shock driven multiphase hydrodynamic instabilities.

¹Advisor

KP1.00043 The effect of confinement length on the stability of planar dense wakes, MINQIANG SI, VIKRANT GUPTA, LARRY K.B. LI, The Hong Kong University of Science and Technology — Planar dense wakes can be found in many industrial processes, such as combustion and paper-making. Confinement is known to make such wakes more locally absolutely unstable,¹ but this destabilizing effect has not been comprehensively examined in real wakes bounded by a finite streamwise domain. For example, it is not known (i) how long the confinement walls should be and (ii) what the critical values of the operating parameters should be for global instability to occur. In this experimental study, we try to answer these questions by examining a planar dense wake consisting of a central stream of CO₂ (dense gas) sandwiched by two identical outer streams of air (light gas). The wake is confined by solid walls of variable length, which act as an adjustable confinement. We find that the confinement length has a strong influence on the hydrodynamic stability of the wake: (a) self-excited global oscillations appear only when the confinement length exceeds a critical value and (b) the streamwise location of the wavemaker changes with confinement length. Knowledge of how long the confinement walls should be for global instability to occur under various conditions could be useful for optimizing industrial processes.

¹Juniper, M.P. 2006 *J. Fluid Mech.* 565, 171–195.

KP1.00044 Mobility functions of a spheroidal particle near a planar elastic membrane, ABDALLAH DADDI-MOUSSA-IDER, Biofluid Simulation and Modeling, Universität Bayreuth, Universitätsstraße 30, 95440, Bayreuth, Germany, MACIEJ LISICKI, Department of Applied Mathematics and Theoretical Physics, Wilberforce Rd, Cambridge CB3 0WA, United Kingdom, STEPHAN GEKLE, Biofluid Simulation and Modeling, Universität Bayreuth, Universitätsstraße 30, 95440, Bayreuth, Germany — Using an analytical theory, we compute the leading order corrections to the translational, rotational and translation-rotation coupling mobilities of a prolate spheroid immersed in a Newtonian fluid and moving nearby an elastic cell membrane. The corrections are expressed in terms of the spheroid-to-membrane distance, spheroid orientation and the characteristic frequencies associated with membrane shearing and bending. We find that the corrections to the translation-rotation coupling mobility are primarily determined by bending resistance whereas shearing elasticity manifests itself in a more pronounced way in the rotational mobility. We further demonstrate the validity of the analytical approximation by close comparison with boundary integral simulations of a truly extended spheroidal particle. The analytical calculations are found to be in a good agreement with the numerical simulations over the whole range of the applied frequencies.

KP1.00045 Three Dimensional Plenoptic PIV Measurements of a Turbulent Boundary Layer Overlying a Hemispherical Roughness Element¹, KYLE JOHNSON, BRIAN THUROW, Auburn University, TAEHOON KIM, University of Illinois, Urbana-Champaign, GIANLUCA BLOIS, KENNETH CHRISTENSEN, University of Notre Dame — Three-dimensional, three-component (3D-3C) measurements were made using a plenoptic camera on the flow around a roughness element immersed in a turbulent boundary layer. A refractive index matched approach allowed whole-field optical access from a single camera to a measurement volume that includes transparent solid geometries. In particular, this experiment measures the flow over a single hemispherical roughness element made of acrylic and immersed in a working fluid consisting of Sodium Iodide solution. Our results demonstrate that plenoptic particle image velocimetry (PIV) is a viable technique to obtaining statistically-significant volumetric velocity measurements even in a complex separated flow. The boundary layer to roughness height-ratio of the flow was 4.97 and the Reynolds number (based on roughness height) was 4.5710³. Our measurements reveal key flow features such as spiraling legs of the shear layer, a recirculation region, and shed arch vortices. Proper orthogonal decomposition (POD) analysis was applied to the instantaneous velocity and vorticity data to extract these features.

¹Supported by the National Science Foundation grant no. 1235726

KP1.00046 Experimental Gravity Currents Propagating Downslope Over A Synthetic Topography, ANDREA BURGOS CUEVAS, ANGEL RUIZ-ANGULO, CARLOS PALACIOS-MORALES, Univ Nac Autonoma de Mexico — Lock-release gravity currents are studied experimentally in order to investigate their dynamics and the mixing process between them and the ambient fluid. We produced these currents in a laboratory tank and allow them to propagate downslope first in a flat slope and then in a rough one with a synthetic topography. This topography is similar to the one of a side of a mountain near mexico's valley. Our aim is to investigate the dynamics of gravity currents as similar as possible to the mountain breezes that can develop around this valley. To the best of our knowledge, there are few experimental investigations that take into account the roughness of the slope. For each experiment, we obtain the instantaneous velocity fields using the standard piv technique. From the velocity fields, we estimate the entrainment coefficient time series. We found that this coefficient depends on the roughness of the surface where the current propagates. Besides, pressure time series were obtained in synthetic stations along the rough profile. These series showed a very clear signal of the gravity current propagating along the slope.

KP1.00047 Oceanic Double-Diffusive Layer Thicknesses in the Presence of Turbulence, NICOLE SHIBLEY, MARY-LOUISE TIMMERMANS, Yale University — Double-diffusive stratification in the ocean is characterized by staircase structures consisting of mixed layers separated by high-gradient interfaces in temperature and salinity. Several past studies have examined mechanisms that govern the observed thicknesses of staircase mixed layers. In one formalism, the mixed-layer thickness is set by layer formation that arises when a heat source is applied at the base of water that is stably-stratified in salinity; in another, the equilibrium thickness of mixed layers has been explained as the product of “merging,” where thin layers continue to grow until they reach a thickness determined by a criterion relating the ratio of heat flux to salt flux and the density ratio. We extend the above two theories to consider the influence of turbulence on mixed-layer thicknesses. The study has implications for the Arctic Ocean where double-diffusive staircases are widely present, and mixed-layer thicknesses are well-resolved by ocean measurements. Our theoretical framework provides a means to determine turbulent diffusivities (in regions where microstructure measurements are not available) by considering only observations of density ratio, stratification, and layer thicknesses.

KP1.00048 The relationship between double-diffusive intrusions and staircases in the Arctic Ocean, YANA BEBIEVA, MARY-LOUISE TIMMERMANS, Yale Univ — The origin of double-diffusive staircases in the Arctic Ocean is investigated for the particular background setting in which both temperature and salinity increase with depth. Motivated by observations that show the co-existence of thermohaline intrusions and double-diffusive staircases, a linear stability analysis is performed on the governing equations to determine the conditions under which staircases form. It is shown that a double-diffusive staircase can result from interleaving motions if the observed bulk vertical density ratio is below a critical vertical density ratio estimated for particular lateral and vertical background temperature and salinity gradients. Vertical temperature and salinity gradients dominate over horizontal gradients in determining whether staircases form. Examination of Arctic Ocean temperature and salinity measurements indicates that observations are consistent with the theory for reasonable choices of eddy diffusivity and viscosity.

KP1.00049 Shock Instability and Pattern Emergence in Oscillated Granular Media¹, JUSTIN STUCK, SARAH ANDERSON, BARBARA SKRZYPEK, JON BOUGIE, Loyola University Chicago Department of Physics — We study shocks formed in vertically oscillated layers of granular media and how shock instability relates to resultant pattern formation. Layers of granular media oscillated vertically on a plate at accelerational amplitudes greater than gravity are tossed off the plate, and shocks are formed upon the layers return to the plate. Previous studies have shown that the emergence of standing-wave patterns is dependent on the plates accelerational amplitude and oscillation frequency. We numerically solve continuum equations to Navier-Stokes order using forward-time, centered space (FTCS) differencing on a three-dimensional spatial grid. We employ variable timesteps and parallelization for efficiency. These simulations demonstrate shock instability before and after the onset of patterns. We use data from these simulations to investigate the connection between shock instability and pattern emergence.

¹This research is supported by the Loyola Undergraduate Research Opportunities Program.

KP1.00050 Oscillation Frequency and Pattern Wavelength in Shaken Granular Media¹, SARAH ANDERSON, BARBARA SKRZYPEK, JUSTIN STUCK, JON BOUGIE, Loyola University Chicago Department of Physics — When a layer of grains atop a plate is vertically oscillated at amplitudes greater than that of gravity, the layer of the material leaves the plate at some point in the cycle. Shocks form in the layer upon its return collision with the plate. Standing wave patterns also form at various amplitudes exceeding a critical value for the system. Previous research has examined the relationship between the shock strength and driving frequency at a fixed layer depth and accelerational amplitude. For a given layer depth, a decrease in frequency corresponds to a stronger shock and greater pattern wavelength. We characterize the base state of the system by investigating the shocks just prior to pattern formation in the media, using numerical simulations of continuum equations to Navier-Stokes order. We use this characterization to study the relationship between shock instability and the patterns formed in these layers.

¹This research is supported by the Loyola Undergraduate Research Opportunities Program.

KP1.00051 Magnetohydrodynamic simulations of a magnetized spherical couette experiment¹, ELLIOT KAPLAN, HENRI-CLAUDE NATAF, NATHANIEL SCHAEFFER, Institut des Science de la Terre — Magnetized spherical Couette flow is a common test bed for studying astro- and geophysically relevant magnetohydrodynamics. A magnetic field is applied to an electrically conductive fluid lying between two co- or counterrotating spheres, and the flow and the magnetic fields influence each other in subtle or dramatic ways. One such experiment, the Derviche Tourneur Sodium experiment (DTS-Ω) recently went through a set of upgrades to better characterize the flows and induced magnetic fields. In tandem with the upgrades, a set of direct numerical simulations were run with the XSHELLS code to give a more complete view of the fluid and magnetic dynamics at various rotation rates of the inner and outer spheres. XSHELLS is a highly efficient hybrid finite-difference pseudospectral solver of the coupled Navier-Stokes and magnetic induction equations. These simulations reveal several dynamic regimes determined by the Rossby number ($Ro = \Delta\Omega/\Omega_o$). These include quasigeostrophic flows, saturated hydrodynamic instabilities, and long lived filamentary structures. By comparing the high spatial resolution measurements of the simulation with the long duration measurements of the experiment, we can get a more complete picture of the dynamic system we're exploring.

¹Centre National de la Recherche Scientifique

KP1.00052 Thermoelectrokinetic instability in micro/nanoscales, GEORGY GANCHENKO, NATALIA GANCHENKO, Laboratory of Micro- and Nanoscale Electro- and Hydrodynamics, Financial University — A novel sophisticated type of electro-hydrodynamic instability in an electrolyte solution near ion-selective surfaces in an external electric field is discovered theoretically. The key mechanism of the instability is caused by Joule heating but dramatically differs from the well-known Raleigh-Benard convection. The investigation is based on the Nernst-Planck-Poisson-Navier-Stokes system along with the energy equation and corresponding BCs. The 1D quiescent steady state in microscales can be unstable with respect to either short-wave Rubinstein-Zaltzman or long-wave thermoelectrokinetic instability. The last one prevails in long microchannels and good enough thermal insulation of the system. In addition to the linear stability analysis a direct numerical simulation of the full 3D nonlinear system is fulfilled using a parallel computing. In the final coherent structures salt concentration, temperature and electric current are localized in narrow long fingers normal to the ion-selective surface while space charge forms crown-like micro-patterns. The investigation results can be useful in desalination problem.

KP1.00053 Time-scale estimation of unstirred layer formation in osmotically driven flow., TOMOAKI ITANO, TAISHI INAGAKI, Kansai University, KEITO KONNO, Hitachi Solutions, Ltd., MASAKO SUGIHARA-SEKI, Kansai University — We study the osmotic solvent flow driven by solute concentration difference across a semi-permeable membrane. The concentration difference across the membrane drives the solvent flow penetrating from the low concentration side through pores of the membrane. This spontaneous solvent flow transports solutes away from the membrane in the opposite side, which locally reduces the solute concentration in the vicinity of the membrane. The concentration boundary layer developed locally near the membrane in the case of absence of external stirring process was termed as unstirred layer in the previous studies, which has been recognized as a key of the unfavorable virtual resistance and membrane fouling in the water filtration of the desalination process. In the previous studies, the formation of the unstirred layer was analyzed under the assumption that the thickness of the unstirred layer is steady, which however contradicts the smoothness of the solute concentration at the end of the layer. In the present study, in order to resolve the contradiction, we assume the unsteadiness in the layer development so that the thickness of the unstirred layer may be estimated analytically.

KP1.00054 Deposition of a large area of nanoparticles at an interface onto a substrate¹, DUCK-GYU LEE, THANH-BINH NGUYEN, HYUNUEI LIM, Korea Institute of Machinery and Materials — Inspired by the antireflective function of moth eyes, attributing to the nanostructures on the surface of eyes, it has been a growing interest to fabricate a well-ordered array of nanoparticles. In this study, we demonstrated a simple fabrication method to generate a large area of close-packed nano-particles at a liquid-gas interface for depositing the particles onto a substrate. We experimentally found the optimal concentration of particles with a surfactant which enables the particles float at an interface in the form of a uniform array of particles. Then we gradually attached the array of particles to the surface of inclined substrate with an angle in water by reducing the level of water. It was observed that the flow rate of reducing water level and the inclination angle of the submerged substrate play an important role in determining the uniformity of the deposited monolayer on the substrates. To find the conditions under which the flow rate and the inclination make the uniform monolayer on the substrates, we made a regime map based on dimensionless parameters.

¹This research was supported by Korea Institute of Machinery and Materials under Grant NK196D

KP1.00055 A DC electrophoresis method for determining electrophoretic mobility through the pressure driven negation of electro osmosis, PASCAL KARAM, SUMITA PENNATHUR, University of California, Santa Barbara — Characterization of the electrophoretic mobility and zeta potential of micro and nanoparticles is important for assessing properties such as stability, charge and size. In electrophoretic techniques for such characterization, the bulk fluid motion due to the interaction between the fluid and the charged surface must be accounted for. Unlike current industrial systems which rely on DLS and oscillating potentials to mitigate electroosmotic flow (EOF), we propose a simple alternative electrophoretic method for optically determining electrophoretic mobility using a DC electric fields. Specifically, we create a system where an adverse pressure gradient counters EOF, and design the geometry of the channel so that the flow profile of the pressure driven flow matches that of the EOF in large regions of the channel (ie. where we observe particle flow). Our specific COMSOL-optimized geometry is two large cross sectional areas adjacent to a central, high aspect ratio channel. We show that this effectively removes EOF from a large region of the channel and allows for the accurate optical characterization of electrophoretic particle mobility, no matter the wall charge or particle size.

KP1.00056 Optimal Spatial Scale for Curvature Calculations in Multiphase Flows, JACOB SENECA, MARK OWKES, Montana State University — In gas-liquid flows, the surface tension force often controls the dynamics of the flow and an accurate calculation of this force is necessary for predictive simulations. The surface tension force is directly proportional to the curvature of the gas-liquid interface, making accurate curvature calculations an essential consideration. Multiple methods have been developed to calculate the curvature of volume of fluid (VoF) interface capturing schemes, such as the height function method. These methods have been extensively tested. However, the impact of the scale or size of computational stencil on which the curvature is computed, has not been correlated with the rate at which interface perturbations relax under the surface tension force. In this work, the effect of varying the scale on which the curvature is computed has been tested and quantified. An optimal curvature scale is identified that leads to accurate and converging curvatures, and accurate timescales for surface tension induced, interface dynamics.

KP1.00057 the God Particle & the Delusion of Grandeur¹, WH- MAKSOED², Prodi of Physics UI [ILUNI], Depok 16415- West Java — It had been established that it was crystalline The inner core is isolated from the rest of earth by the low-viscosity fluid outer core, and it can rotate, nod, precess, wobble, oscillate and even flip over, being only loosely constrained by the surrounding shells- Anderson, 2002. Furthers in accordances of PMRI from Dr.Robert K. Sembiring to ASTRANOMICS, herewith Richard Dawkins: “the God delusion” – 2006 ever quotes by the Rector of theUniversity of INDONESIA 2006 HE. Mr. Prof. Dr.derSoz Gumilar Rusliwa SOMANTRI: “Beyond ‘delusion of grandeur’ menuju INDONESIA baru Bebas Kemiskinan” ever retrieves Lester G. Telser- 1994: “the Usefulness of Core Theory in Economics” - “core theory furnishes a useful framework for a wide variety of economic problems. It has an undeserved reputation of being too abstract owing mainly to the manner in which it is employed in the theory of general equilibrium.”

¹Heartfelt Gratuities to HE. Mr. Prof. Ir. Handojo

²in coincidences of ”if the Universe is the answer, what is the question?”-Leon Lederman:”the God particle” herewith Don L. Anderson:”the inner inner core of earth”, PNAS- 2002

KP1.00058 Nanochannel arrays etched into hexagonal boron nitride meso-membranes by focused ion beam., REMY FULCRAND, SÉBASTIEN LINAS, FRANÇOIS CAUWET, BLAISE POINSOT, ARNAUD BRIOUDE, Univ Lyon, Université Claude Bernard Lyon 1, CNRS, Institut Lumière Matière, F-69622, Villeurbanne, France — Meso-membranes with highly ordered nano channel arrays have been fabricated by patterning hexagonal boron nitride (h-BN) films using a focused ion beam. The complete experimental procedure will be given in detail form the chemical vapor deposition for h-BN synthesis to its patterning and the final membrane design for nanofluidic experiments. The membranes obtained are characterized at each experimental step by electron microscopy and Raman spectroscopy. The technique is finally applied to fabricate devices in which the only passage for a fluid is a nano channel array etched into a h-BN film.

KP1.00059 The hydrodynamic interaction of two small freely-moving particles in a Couette flow of a yield stress fluid¹, MOHAMMADHOSSEIN FIROUZNIA, Department of Mechanical Engineering, Ohio University, Athens, Ohio 45701-2979, USA, BLOEN METZGER, Aix-Marseille Universite, CNRS, IUSTI UMR 7343, 13453 Marseille, France, GUILLAUME OVARLEZ, University of Bordeaux, CNRS, Solvay, LOF, UMR 5258, 33608 Pessac, France, SARAH HORMOZI, Department of Mechanical Engineering, Ohio University, Athens, Ohio 45701-2979, USA — The flows of non-Newtonian slurries, often suspensions of noncolloidal particles in yield stress fluids, are ubiquitous in many natural phenomena and industrial processes. Investigating the microstructure is essential allowing the refinement of macroscopic equations for complex suspensions. One important constraint on the dynamics of a Stokesian suspension is reversibility, which is not necessarily valid for complex fluids. The interaction of two particles in a reversing shear flow of complex fluids is a guide to understand the behavior of complex suspensions. We study the hydrodynamic interaction of two small freely-moving spheres in a linear flow field of yield stress fluids. An important point is that non-Newtonian fluid effects can be varied and unusual. Depending on the shear rate, even a yield stress fluid might show hysteresis, shear banding and elasticity at the local scales that need to be taken into account. We study these effects with the aid of conventional rheometry, Particle Image Velocimetry and Particle Tracking Velocimetry in an original apparatus. We show our preliminary experimental results.

¹NSF

KP1.00060 Strangely stable sphere stacking in yield-stress fluids, PIYUSH SINGH, SCOTT ROGMAN, JONATHAN FREUND, RANDY EWOLDT, University of Illinois at Urbana-Champaign — We have previously observed a novel phenomenon whereby spheres sediment and form a single, stable vertical stack in a yield-stress fluid. Individually, the spheres settle to the bottom, since the yield stress is insufficient to suspend a single sphere. However, cooperative effects result in surprisingly stable stacking. Here, we further explore this phenomenon experimentally in a simple yield stress fluid, a carbopol microgel. Depending on the yield stress of the fluid, the sphere density, and the precise alignment of the spheres, a varying number of spheres can be stacked. Although a taller stack is observed for spheres of the same size, smaller stacks are frequently formed for spheres with mixed sizes and offset centers. This stacking phenomenon is not amenable to a simple force balance analysis because of the complex interplay of viscous and yield stresses and the non-trivial deformation zone in a yield-stress fluid. This study provides new insights on the collective flow behavior of objects in structurally complex and widely used yield-stress fluids. Furthermore, this observed phenomenon can be used to check the predictive efficacy of past, present, and future constitutive models for rheologically-complex fluids.

KP1.00061 Dispersed phase effects on boundary layer turbulence¹, DAVID RICHTER, BRIAN HELGANS, University of Notre Dame — In natural and environmental settings, turbulence is often seeded with some sort of dispersed phase: dust, rain, snow, sediment, etc. Depending on the circumstances, elements of the dispersed phase can participate in both dynamic and thermodynamic coupling, thereby altering the turbulent transfer of heat, moisture, and momentum through several complex avenues. In this study, evaporating droplets are two-way coupled to turbulent wall-bounded flow via direct numerical simulation (DNS) and Lagrangian point particle tracking, and we are specifically interested in the wall-normal transport of momentum, heat, and moisture. Our studies show that particles can carry significant portions of all three, and that this is a strong function of the particle Stokes number. These findings are interpreted in the context of environmental flows and the practical implications will be discussed.

¹The authors acknowledge the National Science Foundation for funding under Grant AGS-1429921

KP1.00062 Gravitational collapse of colloidal gels: Origins of the tipping point¹, POORNIMA PADMANABHAN, ROSEANNA ZIA, Cornell University — Reversible colloidal gels are soft viscoelastic solids in which durable but reversible bonds permit on-demand transition from solidlike to liquidlike behavior; these $O(kT)$ bonds also lead to ongoing coarsening and age stiffening, making their rheology inherently time dependent. To wit, such gels may remain stable for an extended time, but then suddenly collapse, sedimenting to the bottom of the container (or creaming to the top) and eliminating any intended functionality of the material. Although this phenomenon has been studied extensively in the experimental literature, the microscopic mechanism underlying the collapse is not well understood. Effects of gel age, interparticle attraction strength, and wall effects all have been shown to affect collapse behavior, but the microstructural transformations underlying the tipping point remain murky. To study this behavior, we conduct large-scale dynamic simulation to model the structural and rheological evolution of colloidal gels subjected to various gravitational stresses, examining the detailed micromechanics in three temporal regimes: slow sedimentation prior to collapse; the tipping point leading to the onset of rapid collapse; and the subsequent compaction of the material as it approaches its final bed height.

¹Acknowledgment for funding and support from the Office of Naval Research; the National Science Foundation; and NSF XSEDE.

KP1.00063 A study of thin-walled Taylor column under the influence of rotation¹, KUAN-RUEI LAI, CHIN-CHOU CHU, CHIEN-CHENG CHANG, Institute of Applied Mechanics, National Taiwan University — An extended study of thin-walled Taylor column under the influence of rotating cylinder is presented with very consistent results in numerical simulations and laboratory experiments. In the previous set-up, the Taylor column effect is produced under the influence of protruded cylinder from the top lid, and the thin-walled Taylor column is formed by draining of the fluid at the bottom. The primary interest of this study is to investigate the influence to thin-walled Taylor column when the cylinder is exerted with a relative rotation rate under very small Rossby number ($Ro = U/fR$) and Ekman number ($Ek = \nu/fR^2$). The flow patterns are performed with different cylinder height ratios (h/H) along with varying relative rotation ratio of cylinder to the background $\alpha = \omega/\Omega$. Steady-state solutions being solved numerically in the rotating frame are shown to have good agreements with experimental flow visualizations on the resulting appearance of deformed thin-walled Taylor columns. As a result, the thin-walled Taylor column is observed to strengthen up with increasing α , and weakens with decreasing α . In addition, the weakening thin-walled Taylor column is observed to experience a break through transition near the bottom, which penetration diverged the recirculating region into two portions.

¹Supported by the Ministry of Science and Technology, TAIWAN ROC, contract no's 103-2221-E002-099-MY3; 105-2221-E002-097-MY3.

KP1.00064 Effects pf Rounding on Flow Characteristics Past an Angulated Cylindrical Object¹, DOOHYUN PARK, KYUNG-SOO YANG, JAEHEE KIM, Inha University — In the present numerical study, we aim at elucidating the effects of rounding on flow topology past an angulated cylindrical object. Change in flow topology significantly affects flow-induced forces on the object. We consider the rounded cylinders ranging from a square cylinder of height D to a circular cylinder of diameter D by rounding the four corners of a square cylinder with a quarter circle of fixed radius (r). An immersed boundary method was adopted for implementation of the cylinder cross-sections in a Cartesian grid system. The key parameters are Reynolds number (Re) and corner radius of curvature (r). A small rounding delays the flow separation towards the trailing rounded edges, resulting in lower lift fluctuation. The minimum mean drag also occurs when the sharp edges of a square cylinder are partially rounded. The optimal edge-radius ratio (r/d) for lift fluctuation or for mean drag depends upon Re , and high- Re flow tends to be more sensitive to small rounding. The main topological changes resulting from four-edge rounding can be obtained by leading-edge rounding alone. Trailing-edge rounding, however, plays a positive role in stabilizing the flow when the two types of rounding are combined.

¹This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP) (No. 2015R1A2A2A01002981).

KP1.00065 Blast wave mitigation by dry aqueous foam: numerical modelling and experimental investigation., DENIS COUNILH¹, FELIX BALLANGER², NICOLAS RAMBERT³, JEAN-FRANCOIS HAAS⁴, CEA,DAM, DIF 91297 ARPAJON, FRANCE, ASCHWIN CHINNAYYA⁵, INSTITUT PPRIME, ENSMA, 86971 CHASSENEUIL, FRANCE, ALEXANDRE LEFRANCOIS⁶, CEA,DAM, Gramat, 46500 GRAMAT, FRANCE — Dry aqueous foams (two-phase media with water liquid fraction lower than 5%) are known to mitigate blast wave effects induced by an explosion. The CEA has calibrated his numerical multiphase code MOUSSACA from shock tube and high-explosive experiments. The shock tube experiments have highlighted the foam fragmentation into droplets and the momentum transfer between the liquid and gas phases of the foam. More recently, experiments with hemispheric explosive charges from 3 g to 120 g have provided more findings about the pressure and impulse mitigation properties of foams. We have also taken into account the heat and mass transfer, as well as the droplets secondary breakup, characterized by the Weber number, ratio of inertia over surface tension. Good agreement is found between the calculation and the experiments.

¹co-supervisor of the Felix Ballanger 's doctoral thesis

²PhD Student

³laboratory manager

⁴engineer

⁵supervisor of the Felix Ballanger 's doctoral thesis

⁶engineer

KP1.00066 3D Lattice Boltzmann Modeling of Nanoparticle Self-Assembly in Evaporating Droplets and Rivulets, MINGFEI ZHAO, XIN YONG, Binghamton University — In this work, a three-dimensional free-energy-based multiphase lattice Boltzmann method-Lagrangian particle tracking hybrid model is presented to simulate nanoparticle-laden droplets and rivulets undergoing evaporation. The 3D model enables the development of the 3D flow structures in the evaporating droplets, as well as allows us to capture the axial flows in the evaporating rivulets. We first model non-evaporating droplets and rivulets loaded with nanoparticles and the effects of particle-fluid interaction parameters on particle dynamics are characterized. By implementing evaporation, we probe the self-assembly of nanoparticles inside the fluid mass or at the liquid-vapor interface. The 3D microstructure of nanoparticle assemblies is quantified through radial distribution functions and structure factors. In particular, the final deposit of evaporating rivulets with oscillatory axial flows is revealed, resembling the flow field in printed rivulets in experiments. Our findings offer a theoretical framework to explore the dynamics of nanoparticle self-assembly in evaporating fluid mass.

KP1.00067 Measuring Surface Tension of a Flowing Soap Film, AAKASH SANE, ILDOO KIM, SHREYAS MANDRE, Brown University — It is well known that surface tension is sensitive to the presence of surfactants and many conventional methods exist to measure it. These techniques measure surface tension either by intruding into the system or by changing its geometry. Use of conventional methods in the case of a flowing soap film is not feasible because intruding the soap film changes surface tension due to Marangoni effect. We present a technique in which we measure the surface tension in situ of a flowing soap film without intruding into the film. A flowing soap film is created by letting soap solution drip between two wires. The interaction of the soap film with the wires causes the wires to deflect which can be measured. Surface tension is calculated using a relation between curvature of the wires and the surface tension. Our measurements indicate that the surface tension of the flowing soap film for our setup is around 0.05 N/m. The nature of this technique makes it favorable for measuring surface tension of flowing soap films whose properties change on intrusion.

KP1.00068 Geometric design of the best performing auto-rotating wing, YUCEN LIU, LIONEL VINCENT, EVA KANSO, Univ of Southern California — Many plants use gravity and aerodynamics to disperse their seeds away from the parent plant. Various seed designs result in different dispersal modes from gliding to auto-rotating. Here, we are interested in understanding the effect of geometric design of auto-rotating seedpods on their aerodynamic performance. As an experimentally tractable surrogate to real seedpods, we investigate auto-rotating paper wings of various shape designs. We compare these designs to a control case consisting of the canonical rectangular wing. Inspired by aerodynamics, we begin by considering the benefit of an elliptical planform, and test the effect of aspect ratio on flight range and descent angle. We find the elliptical planform improves the tumbling rate and the aspect ratio has a positive effect on the flight performance of the wings. We then test two families of more complex shapes: one of tapered planform and one of a planform with sharp tips. We look for an optimal flight performance while constraining either the mass or the maximum length and width of the wing. We find that wings with sharper tips and larger length have higher auto-rotation rates and improved performance. The results imply that both the planform and length of the wing contribute to the wings flight performance.

KP1.00069 Dynamics of a high viscosity layer in response to shear flow¹, EHSAN ESMAILI, ANNE STAPLES, Virginia Tech — We use the Shan-Chen multicomponent Lattice Boltzmann method (LBM) to investigate the time evolution of a thin liquid film (phase I) coating a solid surface under the action of a shearing force imposed by a surrounding fluid (phase II), whose viscosity is significantly lower than that of the film. The goal of this study is to use LBM to capture the contact line motion and interfacial dynamics for an oil-like liquid film which is driven by the upper phase (water) movement as a first approach to modeling thin film dewetting in wave swept marine environments. Lubrication theory is used to validate the results for the driven thin film, and the LBM simulations investigate the effects of the upper phase movement, lower phase thickness, and angle of the imposed shearing force on the thin film profile.

¹This work was supported by the National Science Foundation under grant number 1437387

KP1.00070 Droplet Impact on a Heated Surface under a Depressurized Environment, RYUTA HATAKENAKA, Japan Aerospace Exploration Agency, YOSHIYUKI TAGAWA, Tokyo University of Agriculture and Technology — Behavior of a water droplet of the diameter 1-3mm impacting on a heated surface under depressurized environment (100kPa -1kPa) has been studied. A syringe pump for droplet generation and a heated plate are set into a transparent acrylic vacuum chamber. The internal pressure of the chamber is automatically controlled at a target pressure with a rotary pump, a pressure transducer, and an electrical valve. A silicon wafer of the thickness 0.28 mm is mounted on the heater plate, whose temperature is directly measured by attaching a thermocouple on the backside. The droplet behavior is captured using a high-speed camera in a direction perpendicular to droplet velocity. Some unique behaviors of droplet are observed by decreasing the environmental pressure, which are considered to be due to two basic elements: Enhancement of evaporation due to the lowered saturation temperature, and shortage of pneumatic spring effect between the droplet and heated wall due to the lowered pressure of the air.

KP1.00071 Modeling flow for modified concentric cylinder rheometer geometry, KAREN EKERUCHE, KELLY CONNELLY, H. PIROUZ KAVEHPOUR, University of California Los Angeles — Rheology experiments on biological fluids can be difficult when samples are limited in volume, sensitive to degradation, and delicate to extract from tissues. A probe-like geometry has been developed to perform shear creep experiments on biological fluids and to use the creep response to characterize fluid material properties. This probe geometry is a modified concentric cylinder setup, where the gap is large and we assume the inner cylinder rotates in an infinite fluid. To validate this assumption we perform shear creep tests with the designed probe on Newtonian and non-Newtonian fluids and vary the outer cylinder container diameter. We have also created a numerical model based on the probe geometry setup to compare with experimental results at different outer cylinder diameters. A creep test is modeled by applying rotation to the inner cylinder and solving for the deformation of the fluid throughout the gap. Steady state viscosity values are calculated from creep compliance curves and compared between experimental and numerical results.

KP1.00072 Symbolic dynamics applied to a numerical simulation of a perturbed Hill's spherical vortex., JOSHUA ARENSON, SPENCER SMITH, KEVIN MITCHELL, University of California Merced — In the classic Hill's spherical vortex flow an invariant sphere prevents material inside the vortex from mixing with material outside. Here, we apply an additional shear and rotational flow to break the symmetry of the vortex, thereby allowing mixing of the material inside and outside. The resulting system exhibits fully 3D chaotic advection. We consider the scattering of passive tracers that are drawn into and then ejected from the vortex. Here we focus on the numerical computation of fractal scattering functions—the time trapped within the vortex as a function of two impact parameters. We then compare the fractal self-similarity of these scattering functions to those predicted by 3D homotopic lobe dynamics—a new symbolic method of describing topological dynamics.

KP1.00073 Onset of flow instability in rigid foams, CARLOS ANDRS BARROS OCHOA, Universidad Nacional de Colombia, PETR DENISSENKO, The University of Warwick, CARLOS ALBERTO DUQUE DAZA, Universidad Nacional de Colombia — The flow transition between stationary and time dependent regimes at the exit of a block of open-cell foam has been examined experimentally using Laser Doppler Anemometry. Measurements have been conducted at three points located at a plane located 10 mm downstream from the exit of the foam. The streamwise component of fluid velocity was measured at multiple flow rates. The probability density function of the velocity is two-peaked at Reynolds numbers above 25 based on the average pore size and is a skewed one-peak distribution at lower flow rates. Numerical simulations are being conducted using a computer tomography scanned model of the foam to match the experimental measurements. Obtained results are discussed in the context of using the open-cell foams in catalytic reactors.

KP1.00074 Prediction of Algebraic Instabilities, PAULA ZARETZKY, KRISTINA KING, NICOLE HILL, KIMBERLEE KEITHLEY, NATHANIEL BARLOW, STEVEN WEINSTEIN, MICHAEL CROMER, RIT — A widely unexplored type of hydrodynamic instability is examined - large-time algebraic growth. Such growth occurs on the threshold of (exponentially) neutral stability. A new methodology is provided for predicting the algebraic growth rate of an initial disturbance, when applied to the governing differential equation (or dispersion relation) describing wave propagation in dispersive media. Several types of algebraic instabilities are explored in the context of both linear and nonlinear waves.

KP1.00075 STUDENT POSTER COMPETITION: COMPUTATIONS —

KP1.00076 Harnessing Energy from Arrays of Oscillating Hydrofoils, FILIP SIMESKI, ARIANNE SPAULDING, JENNIFER FRANCK, School of Engineering, Brown University — Computational Fluid Dynamics (CFD) simulations are performed on multiple-hydrofoil systems for the application of energy harvesting. Oscillating hydrofoils generate power through a coupled heaving and pitching motion. Various linear and staggered configurations consisting of three to four hydrofoils are simulated, and the system efficiency of the array is evaluated, as well as the energy density of the system. Of particular interest is the observation that regular vortices from the foils leading and trailing edges develop into a well-structured wake affecting performance of downstream-located hydrofoils in the system, and leading to an optimal phase difference between foils. Simulations are performed at a Reynolds number of 1000, and utilize OpenFOAM with dynamic meshing libraries employed to handle the foil motion.

KP1.00077 Numerical analysis of the inertial migration of a spherical particle in circular tube flows, TAKUYA YABU, TOMOAKI ITANO, MASAKO SUGIHARA-SEKI, Kansai Univ. — In a study of dilute suspension flow through circular tubes, Segré & Silberberg (1961) reported lateral migration of neutrally buoyant spherical particles to an equilibrium radial position at low Reynolds numbers (Re). A later experimental study by Matas et al. (2004) found that another equilibrium annulus (inner annulus) emerges closer to the tube center at elevated Re . Since existing theoretical studies based on the matched asymptotic expansion could not account for the appearance of the inner annulus, the present study aimed to investigate numerically the equilibrium positions of a rigid spherical particle suspended in a Poiseuille flow for $100 < Re < 1,500$. In the case of particle to tube diameter ratios $\tilde{O}1$, the flow field around the particle was computed by the immersed boundary method to calculate the lateral force exerted on the particle. It was found that for $Re < 1,000$, the lateral force vanishes at a single radial position, corresponding to the so-called Segré-Silberberg annulus, whereas beyond this Re , a new equilibrium position appears closer to the tube center, possibly representing the inner annulus. In addition, it was predicted that for $Re > 1,200$, the Segré-Silberberg annulus disappears and only the inner annulus retains.

KP1.00078 On the Structure Orientation in Rotating and Sheared Homogeneous Turbulence¹, JOYLENE C. AGUIRRE, ADAM F. MOREAU, FRANK G. JACOBITZ, University of San Diego — The results of direct numerical simulations are used to study the effect of rotation on the orientation of structures and the evolution of the turbulent kinetic energy in homogeneous sheared turbulence. Shear flows without rotation, with moderate rotation, and with strong rotation are considered and the rotation axis is either parallel or anti-parallel to the mean flow vorticity. In the case of moderate rotation, an anti-parallel configuration increases the growth rate of the turbulent kinetic energy, while a parallel configuration decreases the growth rate as compared to the flow without rotation. The orientation of turbulent structures present in the flows are characterized using the three-dimensional, two-point autocorrelation coefficient of velocity magnitude and vorticity magnitude. An ellipsoid is fitted to the surface defined by a constant autocorrelation coefficient value and the major and minor axes are used to determine the inclination angle of flow structures in the plane of shear. It was found that the inclination angle assumes a maximum value for the anti-parallel configuration with moderate rotation. Again, the inclination angle for the parallel configuration with moderate rotation is reduced as compared to the case without rotation. The smallest inclination angles are found for the strongly rotating cases. Hence, the inclination angle is directly related to the growth rate of the turbulent kinetic energy.

¹University of San Diego Shiley-Marcos School of Engineering and McNair Scholars

KP1.00079 Aerodynamic Analysis of Morphing Blades¹, CALEB HARRIS, The University of Memphis,

DAVID MACPHEE, The University of Alabama, MADELINE CARLISLE, Louisiana Tech University — Interest in morphing blades has grown with applications for wind turbines and other aerodynamic blades. This passive control method has advantages over active control methods such as lower manufacturing and upkeep costs. This study has investigated the lift and drag forces on individual blades with experimental and computational analysis. The goal has been to show that these blades delay stall and provide larger lift-to-drag ratios at various angles of attack. Rigid and flexible airfoils were cast from polyurethane and silicone respectively, then lift and drag forces were collected from a load cell during 2-D testing in a wind tunnel. Experimental data was used to validate computational models in OpenFOAM. A finite volume fluid-structure-interaction solver was used to model the flexible blade in fluid flow. Preliminary results indicate delay in stall and larger lift-to-drag ratios by maintaining more optimal angles of attack when flexing.

¹Funding from NSF REU site grant EEC 1358991 is greatly appreciated.

KP1.00080 Numerical simulation of microlayer formation in nucleate boiling, ALEXANDRE GUION, JACOPO BUONGIORNO, Massachusetts Institute of Technology, SHAHRIAR AFKHAM, New Jersey Institute of Technology, STEPHANE ZALESKI, Sorbonne Universites, UPMC Univ Paris 06, CNRS, UMR 7190, France — Numerical simulations of boiling resolve the macroscopic liquid-vapor interface of the bubble, but resort to subgrid models to account for microscale effects, such as the evaporation of the liquid microlayer underneath the bubble. Realistic time-dependent microlayer evaporation models necessitate initialization of the microlayer profile. In the recent simulations published in the literature [J. Comp. Phys., 300 (2015): 20-52], missing input data on initial microlayer geometry is replaced by estimated values from separate experimental measurements at similar pressure. Yet, the geometry of the initial microlayer not only depends on pressure for a given set of fluids, but also on bubble growth rate and that dependence is not known *a priori*. In this work, the Volume-of-Fluid (VOF) method, implemented in the open-source code Gerris (gfs.sf.net), is used to simulate, with unprecedented accuracy, the dynamics of microlayer formation underneath a growing bubble. A large numerical database is generated, yielding the microlayer thickness during the inertia controlled phase of bubble growth as a function of radial distance from the bubble root, time, contact angle, and capillary number associated with bubble growth. No significant dependence on density or viscosity ratios were found.

KP1.00081 A nonlinear dynamical system for combustion instability in a pulse model combustor, KAZUSHI TAKAGI, HIROSHI GOTODA, Department of Mechanical Engineering, Tokyo University of Science, 6-3-1 Niijuku, Katsushika-ku, Tokyo 125-8585, Japan — We theoretically and numerically study the bifurcation phenomena of nonlinear dynamical system describing combustion instability in a pulse model combustor on the basis of dynamical system theory and complex network theory. The dynamical behavior of pressure fluctuations undergoes a significant transition from steady-state to deterministic chaos via the period-doubling cascade process known as Feigenbaum scenario with decreasing the characteristic flow time. Recurrence plots and recurrence networks analysis we adopted in this study can quantify the significant changes in dynamic behavior of combustion instability that cannot be captured in the bifurcation diagram.

KP1.00082 Is the Stokeslet model sufficient for finding nutrient uptake of microscopic suspension feeders?, ALEXANDER T. LUTTON, RACHEL E. PEPPER, University of Puget Sound — Microscopic sessile suspension feeders are part of many aquatic ecosystems. They are single-celled, vary in size from a few to about 100 microns in length, live attached to substrates, and serve important ecological roles as both food for larger organisms and consumers of bacteria and other small particles. These organisms create currents in order to bring food toward them. Understanding these currents may allow us not only deeper insight into the ecology of aquatic ecosystems, but also may enable innovation in water treatment. Simulations of the feeding currents of these organisms typically use a simple model that places a Stokeslet above an infinite plane boundary representing the surface of attachment. This model produces a useful approximation for the flow field of the organism, but may be of limited accuracy when the organism is near the boundary. We create a different model composed of a Stokeslet and a potential dipole, which form a sphere. This sphere has a $\sin(\theta)$ tangential velocity boundary condition, accounting for the cell body. Using nutrient flux to the organism as our metric, we investigate the discrepancy between the spherical and Stokeslet models in order to determine the efficacy of the Stokeslet model as an approximation of single-celled suspension feeders.

KP1.00083 STUDENT POSTER COMPETITION: EXPERIMENTS —

KP1.00084 An experimental study of low Re cavity vortex formation embedded in a laminar boundary layer¹, SASHANK GAUTAM, AMY LANG, JACOB WILROY, The University of Alabama - Tuscaloosa — Laminar boundary layer flow across a grooved surface leads to the formation of vortices inside rectangular cavities. The nature and stability of the vortex inside any single cavity is determined by the Re and cavity geometry. According to the hypothesis, under low Re and stable vortex conditions a single cavity vortex leads to a roller-bearing effect which results in a decrease in drag as quantified by velocity profiles measured within the boundary layer. At higher Re once the vortex becomes unstable, drag should increase due to the mixing of low-momentum fluid within the cavity and the outer boundary layer flow. The primary objective of this experiment is to document the phenomenon using DPLV in a tow tank facility. This study focuses on the transition of the cavity flow from a steady to an unsteady state as the Re is increased above a critical value. The change in boundary layer momentum and cavity vortex characteristics are documented as a function of Re and boundary layer thickness.

¹Funding from NSF CBET fluid dynamics grant 1335848 is gratefully acknowledged

KP1.00085 An experimental investigation of blast driven turbulence, BENJAMIN MUSCI, DEVESH RANJAN, Georgia Inst of Tech — In the Georgia Tech Shock and Advanced Mixing Lab, a facility is being built to study blast driven turbulence. Motivated by the discrepancies observed between actual and modeled supernovae, this facility aims to resolve the important spatial scales in the extensive mixing of the outer layers. These outer layers will be modeled by subjecting two-three gases of varying density to a blast wave generated by Exploding Bridge Wires. The blast wave's interaction with perturbations at the gaseous, membrane-less, interfaces will induce the Richtmeyer-Meshkov or Rayleigh-Taylor Instability, depending on the acceleration history and perturbation amplitude. Through the use of simultaneous Particle Image Velocimetry, and Planar Laser Induced Fluorescence, this project aims to determine the effect of interface initial conditions on turbulence. A 2D Diverging Wedge and 3D Diverging Conical Tube are being built to enable repeatable blast-wave production, continuous optical viewing of the flow, reproducible multi-layer interface creation, and the collection of simultaneous density-velocity measurements to directly measure turbulent quantities. The preliminary analysis informing the design of this facility, the construction progress, and updates on newly realized design constraints are presented.

KP1.00086 Flexible Blades for Wind Turbines¹, MADELINE CARLISLE COLLINS, Louisiana Tech University, DAVID MACPHEE, The University of Alabama, CALEB HARRIS, The University of Memphis — Previous research has shown that windmills with flexible blades are more efficient than those with rigid blades. Flexibility offers passive pitch control, preferable to active pitch control which is costly and requires maintenance. Flexible blades morph such that the blade more closely resembles its design point at part load and over load. The lift-to-drag ratios on individual blades was investigated. A mold was designed and machined from an acrylic slab for the casting of blades with a NACA 0012 cross section. A flexible blade was cast from silicone and a rigid blade was cast from polyurethane. Each of these blades was tested in a wind tunnel, cantilever mounted, spanning the whole test section. The angle of attack was varied by rotating the mount. All tests were performed at the same wind speed. A load cell within the mount measured forces on the blade, from which the lift and drag forces were calculated. The stall point for the flexible blade occurred later than for the rigid blade, which agrees with previous research. Lift-to-drag ratios were larger for the flexible blade at all angles of attack tested. Flexible blades seem to be a viable option for passive pitch control. Future research will include different airfoil cross sections, wind speeds, and blade materials.

¹Funding from NSF REU site grant EEC 1358991 is greatly appreciated.

KP1.00087 CLOUD-MAP Field Campaign Measurements of the Earth's Lower Boundary Layer, NICHOLAS FOSTER, ALYSSA AVERY, JAMEY JACOB, Oklahoma State University — CLOUD-MAP (Collaboration Leading Operational UAS Development for Meteorology and Atmospheric Physics) is a 4 year, 4 university collaboration to develop capabilities that will allow meteorologists and atmospheric scientists to use unmanned aircraft as a common, useful everyday tool. Currently, we know that systems can be used for meteorological measurements, but they are far from being practical or robust for everyday field diagnostics by the average meteorologist or scientist. In particular, UAS are well suited for the lower atmosphere, namely the lower boundary layer that has a large impact on the atmosphere and where much of the weather phenomena begin. A sensor set called MDASS (Meteorological Data Acquisition Sonde System) was developed and used to collect and transmit live data necessary for developing such forecasts as well as be usable on multiple platforms ranging from fixed-wing and multi-rotor UAVs to rockets. The data transmitted from MDASS is viewed and stored on a ground control station via LabVIEW in a program developed for real-time data analysis. Results from the first CLOUD-MAP are presented. The campaign resulted in nearly 250 unmanned aircraft flights of 12 separate platforms over a 3 day period, collecting meteorological data at 3 different sites.

KP1.00088 STUDENT POSTER COMPETITION: MICROFLUIDICS AND INTERFACES —

KP1.00089 High-speed ethanol micro-droplet impact on a solid surface¹, YUTA FUJITA, AKIHITO KIYAMA, YOSHIOUKI TAGAWA, Tokyo Univ of Agri & Tech — Recently, droplet impact draws great attention in the fluid mechanics. In previous work, micro-droplet impact on a solid surface at velocities up to 100 m s^{-1} was studied. However the study was only on water micro-droplets. In this study, we experimentally investigate high-speed impact of ethanol micro-droplets in order to confirm the feature about maximum spreading radius with another liquid. A droplet is generated from a laser-induced high-speed liquid jet. The diameter of droplets is around $80 \mu\text{m}$ and the velocity is larger than 30 m s^{-1} . The surface tension of ethanol is 22.4 mNm^{-1} and density is 789 kgm^{-3} . Weber number ranges $\text{We} > 1000$. By using a high-speed camera, we investigate the deformation of droplets as a function of Weber number.

¹This work was supported by JSPS KAKENHI Grant Number JP26709007

KP1.00090 The injection of a highly focused microjet into a soft target¹, NANAMI ENDO, SENNO SUKE KAWAMOTO, YOSHIOUKI TAGAWA, Tokyo Univ of Agri & Tech — Needle-free drug injection systems have been developed in order to supersede traditional syringe injection system with needles. However, in spite of its great potential, these systems are not commonly used. One of the main reasons is to use diffusive jets, which results in severe deceleration of the jets and causes insufficient penetration. Recently, a highly focused microjet generated by irradiating a laser pulse to a point inside a liquid filled in a capillary tube is gathering attention as a method to solve these problems. Although the microjet injection phenomena into a model material of the skin have been studied, the effect of the distance R (R is a distance between a gas-liquid interface and a target) on injection phenomena have not been researched. The distance R is not a parameter which controls the jet generation. However, considering the practical use of the needle-free injection, it is necessary to know appropriate value of the distance R. In this study, we change the distance R in a range of 0.3 mm to 5 mm to investigate its influence on the injection depth Di. As a target, we used 5 wt% gelatin. We show relationship between injection depth Di and distance R and rationalize it in this presentation.

¹This work was supported by JSPS KAKENHI Grant Number JP26709007

KP1.00091 How does the micro-splashing threshold change with drop size?¹, SAM J. BOOS, RACHEL E. PEPPER, Univ of Puget Sound — Micro-splashing is a newly discovered type of splashing that appears within microseconds of first contact between liquid drop and surface, producing tiny droplets with diameters of approximately $10\text{-}50 \mu\text{m}$. The droplets are ejected outwards at speeds over ten times that of the parent drop. Previously discovered splashing phenomena, like prompt or corona splashing, happen much later in the drop impact and produce larger, slower droplets compared to micro-splashing. A greater understanding of micro-splashing may be important in industry and global health because micro-splashes may, for example, affect the quality of ink printing or contribute to atmospheric aerosolization of particles and toxins. An initial study (Thoroddsen ST, Takehara K, Etoh TG. J. Fluid Mech. 706 (2012)) discovered this new type of splashing, described the nature of the micro-splashes, and proposed a mechanism behind their generation. However, micro-splashing is yet to be fully understood. We use high-speed video to determine how drop size affects the threshold velocity for micro-splashing, as a step towards further understanding this phenomenon.

KP1.00092 Separating Particles Using Tangential Flow Filtration and Inertial Microfluidics, AMANDA SINGLETON, MIKE GARCIA, SUMITA PENNATHUR, Univ of California - Santa Barbara — The separation of micron-sized particles is a crucial component in a myriad of applications. Recently researchers have attempted to use inertial microfluidics to separate particles because the technique requires smaller sample volume, has a high throughput, and is inherently robust. Unfortunately, inertial microfluidics lacks versatility: geometric considerations limit variation of particle size. To overcome this limitation, we experimentally investigate the effect of adding permeate flow to refocus particles into tunable equilibrium locations. Specifically, we experimentally investigate the effect of permeate flow on the equilibrium location of 5, 10, and 15-micron polystyrene particles in a MEMS fabricated tangential flow filtration device. We see that contrary to inertial focusing in straight microfluidic channels, smaller particles focus closer to the center than larger particles. Furthermore, the particle equilibrium location is a function of streamwise distance, and equilibrium location at the exit is a function of the ratio of outlet to inlet flow. Taking advantage of this data, we aim to create in-situ control of particle equilibrium locations resulting in real time separations of particles of unknown size distribution. This method can be combined with on-chip devices for diagnostic applications, benefitting the fluids and separations community.

KP1.00093 Three-dimensional wave evolution on electrified falling films. , RUBEN TOMLIN,

DEMETRIOS PAPAGEORGIOU, GREG PAVLIOTIS, Imperial College London — We consider the full three-dimensional model for a thin viscous liquid film completely wetting a flat infinite solid substrate at some non-zero angle to the horizontal, with an electric field normal to the substrate far from the flow. Thin film flows have applications in cooling processes. Many studies have shown that the presence of interfacial waves increases heat transfer by orders of magnitude due to film thinning and convection effects. A long-wave asymptotics procedure yields a Kuramoto-Sivashinsky equation with a non-local term to model the weakly nonlinear evolution of the interface dynamics for overlying film arrangements, with a restriction on the electric field strength. The non-local term is always linearly destabilising and produces growth rates proportional to the cube of the magnitude of the wavenumber vector. A sufficiently strong electric field is able to promote non-trivial dynamics for subcritical Reynolds number flows where the flat interface is stable in the absence of an electric field. We present numerical simulations where we observe rich dynamical behavior with competing attractors, including “snaking” travelling waves and other fully three-dimensional wave formations.

KP1.00094 Towards designing miniature surfing robots , SAEED JAFARI KANG, VAHID VANDADI,

HASSAN MASOUD, University of Nevada, Reno — We theoretically study the surfing motion of chemically and thermally active particles located at a flat liquid-gas interface that sits above a liquid layer of finite depth. The particles' activity creates and maintains a surface tension gradient resulting in the auto-surfing. It is intuitively perceived that Marangoni surfers propel towards the direction with a higher surface tension. Remarkably, we find that the surfers may propel in the lower surface tension direction depending on their geometry and proximity to the bottom of the liquid layer. In particular, our analytical calculations for Stokes flow and diffusion-dominated scalar (i.e. chemical concentration and temperature) fields indicate that spherical particles undergo reverse Marangoni propulsion under confinement whereas disk-shaped surfers always move in the expected direction. We extend our results by proposing an approximate formula for the propulsion speed of oblate spheroidal particles based on the speeds of spheres and disks. Overall, our findings pave the way for designing microsurfers capable of operating in bounded environments.

KP1.00095 The use of micro-/milli- fluidics to better understand the mechanisms behind deep venous thrombosis¹ , ZOE SCHOFIELD, ALESSIO ALEXIADIS, ALEXANDER BRILL, GERARD NASH, DANIELE VIGOLO, University of Birmingham — Deep venous thrombosis (DVT) is a dangerous and painful condition in which blood clots form in deep veins (e.g., femoral vein). If these clots become unstable and detach from the thrombus they can be delivered to the lungs resulting in a life threatening complication called pulmonary embolism (PE). Mechanisms of clot development in veins remain unclear but researchers suspect that the specific flow patterns in veins, especially around the valve flaps, play a fundamental role. Here we show how it is now possible to mimic the current murine model by developing micro-/milli-fluidic experiments. We exploited a novel detection technique, ghost particle velocimetry (GPV), to analyse the velocity profiles for various geometries. These vary from regular microfluidics with a rectangular cross section with a range of geometries (mimicking the presence of side and back branches in veins, closed side branch and flexible valves) to a more accurate venous representation with a 3D cylindrical geometry obtained by 3D printing. In addition to the GPV experiments, we analysed the flow field developing in these geometries by using computational fluid dynamic simulations to develop a better understanding of the mechanisms behind DVT.

¹ZS gratefully acknowledges financial support from the EPSRC through a studentship from the Sci-Phy-4-Health Centre for Doctoral Training (EP/L016346/1).

KP1.00096 POSTDEADLINE —

KP1.00097 Communication of Information with Sub-particles (Sub-strings) from Fifth Dimension of the Universe (Information) as the “Fundamental Symmetry” in the Nature , HASSAN GHOLIBEIGIAN¹, Retired, GHSEM GHOLIBEIGIAN, None, ABDOLAZIM AMIRSHAHKARAMI², Retired, KAZEM GHOLIBEIGIAN³, None — All fundamental particles (strings) getting information from their four animated sub-particles (sub-strings) after processing by them for motion [Gholibeigian, APS April Meeting 2015, abstract #L1.027]. It seems that the source of information which particles and dark matter/energy are floating in it and whispering to its communication may be “fifth dimension” of the nature after space-time dimensions. In other words, the space-time can be the universe’s hardware and information’s dimension can be dynamic software of the universe which has always become up to date. Communication of information has a vital role in creation and evolution of the universe, may be as the “fundamental symmetry” in the nature, which began before the spark to B.B. (Convection Bang), and leads other symmetries and supersymmetry as well as other phenomena. Duration of the before Planck time, from $0 \rightarrow 10^{-44}$ second, and its correspondence space which its result was generation of the very hot and energetic point for the B.B. / C.B. needed to communication of information. It seems that this fifth dimension has appeared for leading the processes before and after Planck time. How this dimension of the nature appeared and has always become up to date?

¹AmirKabir University of Technology, Tehran, Iran

²AmirKabir University of Technology, Tehran, Iran

³Technische Universität, Vienna, Austria

KP1.00098 Embedding of electrodes within a microchannel interfacing a permselective medium for sensing and active control of the concentration-polarization layer , GILAD YOSSIFON, SINWOOK PARK, Technion - Israel Institute of Technology — Previously, it has been shown that for a prescribed system, the diffusion length may be affected by any number of mechanisms including natural and forced convection, electroosmotic flow of the second kind and electro-convective instability. In all of the above mentioned cases the length of the diffusion layer is indirectly prescribed by the complicated competition between several mechanisms which are primarily dictated by the various system parameters and applied voltage. In contrast, we suggest that by embedding electrodes/heaters within a microchannel interfacing a permselective medium, the diffusion layer length may be controlled regardless of the dominating overlimiting current mechanism and system parameters. As well as demonstrating that the simple presence of electrodes can enhance mixing via induced-charge electrokinetic effects, we also offer a means of externally activating embedded electrodes and heaters to maintain external, dynamic control of the diffusion length. Such control is particularly important in applications requiring intense ion transport, such as electrodialysis. At the same time, we will also investigate means of suppressing these mechanisms which is of fundamental importance for sensing applications.

KP1.00099 Experimental study of the submerged jet flow from a circular orifice for low values of Reynolds (Re) number., AYAX HERNANDO TORRES VICTORIA, SEPI ESIME Azcapotzalco Instituto Politcnico Nacional, MARIO ALBERTO SNCHEZ ROSAS, JUAN CASILLAS NAVARRETE, FERNANDO ARAGN RIVERA, JOS ALFREDO JIMNEZ BERNAL, ESIME Zacatenco Instituto Politcnico Nacional SEPI, ABRAHAM MEDINA OVANDO, SEPI ESIME Azcapotzalco Instituto Politcnico Nacional — The results of the experimental study of the submerged jet flow emerging from a circular orifice, where the fluid injected and the fluid in the receiving volume are the same, are presented in this work. Velocity vector fields for Reynolds (Re) 2, 4, 6, 8, 10 & 20 were obtained by means of the PIV technique. Similarly, results for the inward flow for the same geometry and Reynolds (Re) numbers are presented. Velocity profile plots and streamlines, for their corresponding Reynolds (Re) value, are also presented.

KP1.00100 Theoretical study of instability observed inside a precessing cylinder, WALEED MOUHALI, ECE Paris, THIERRY LEHNER, Lth Observatoire de Paris, AZIZ SALHI, Universit de Tunis, ATER COLLABORATION, ATER COLLABORATION — Cyclones have been observed in our experiment involving water in a both rotating and precessing cylinder. The following mechanism can explain their generation: 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 numbers) 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 inflection point leading to a localized inflectional/shear secondary instability. We show that when the parameter ε is increased from low values the mode $m=0$ becomes the most unstable one in this induced differential rotation at a reproducible threshold in ε , which can induce further the observed cyclones. In addition radial jets coming from the lateral boundary layers have been also observed which can drive additional cyclones by another instability developing in the boundary shear layer in presence of radial flow.

KP1.00101 The effect of changing topography on coastal tides and storm surge: a historical perspective¹, STEFAN TALKE, DAVID JAY, LUMAS HELAIRE, RAMIN FAMILKHALILI, Portland State University — Over decadal and century time scales, the topography of coastal harbors changes due to natural and anthropogenic factors. These changes alter the mass and momentum balances of incoming waves, producing measurable changes to tides and surge. Here we use recently recovered archival data, historic bathymetric charts, and numerical models to assess changes in multiple estuaries. In the Columbia River estuary, Ems estuary, and Cape Fear Estuary, channel deepening has increased the M2 tide between 10 to 100% since the 19th century, due to both reduced frictional effects and altered resonance. The bathymetric perturbations also affect the propagation of other long-period waves: in Wilmington (NC), the worst-case scenario CAT-5 storm surge is modeled to increase by 50% since 19th century conditions. Similarly, in New York harbor, the 10 year storm-tide level has outpaced sea-level rise by nearly 30 cm since 1850. In the Columbia River, reduced friction has decreased the river slope (reducing water levels), but also led to amplification of both tides and flood waves. Going forward, historical bathymetric change may provide a clue to the future effects of climate change and continued anthropogenic development.

¹National Science Foundation; US Army Corp of Engineers

KP1.00102 The control effect in a detached laminar boundary layer of an array of normal synthetic jets.¹, FERNANDO VALENZUELA CALVA, RUBEN AVILA RODRIGUEZ, Universidad Nacional Autonoma de Mexico — In this work, 3D numerical simulations of an array of three normal circular synthetic jets embedded in an attached laminar boundary layer that separates under the influence of an inclined flap are performed for flow separation control. At the beginning of the present study, three cases are used to validate the numerical simulation with data obtained from experiments. The experimental data is chosen based on the cases which presented higher repeatability and reliability. Simulations showed reasonable agreement when compared with experiments. The simulations are undertaken at three synthetic jet operating conditions, i.e. Case A: $L = 2$, $VR = 0.32$; Case B: $L = 4$, $VR = 0.64$ and Case C: $L = 6$, $VR = 0.96$. The vortical structures produced for each synthetic jet operating condition are hairpin vortices for Case A and tilted vortices for Case B and C, respectively. By examining the spatial wall shear stress variations, the effect on the boundary layer prior to separation of the middle synthetic jet is evaluated. For effective flow control, produced at a relatively low the finding from this study suggests that hairpin vortical structures are more desirable structures.

¹Universidad Nacional Autonoma de Mexico

KP1.00103 Defects at the Nanoscale Impact Contact Line Motion at all Scales¹, HUGO PERRIN, Univ Paris Diderot APC, BRUNO ANDREOTTI, Univ. Paris Diderot - PMMH ESPCI - CNRS, ROMAIN LHERMEROUT, KRISTINA DAVITT, ETIENNE ROLLEY, LPS ENS - UPMC - Univ. Paris Diderot - CNRS, WETTING AND NUCULATION TEAM, PMMH TEAM — The contact angle of a liquid drop moving on a real solid surface depends on the speed and direction of motion of the three-phase contact line. Many experiments have demonstrated that pinning on surface defects, thermal activation and viscous dissipation impact contact line dynamics, but so far efforts have failed to disentangle the role of each of these dissipation channels. Here, we propose a unifying multi-scale approach that provides a single quantitative framework. We use this approach to successfully account for the dynamics measured in a classic dip-coating experiment performed over a unprecedentedly wide range of velocity. We show that the full contact line dynamics up to the liquid film entrainment threshold can be parametrized by the size, amplitude and density of nanometer-scale defects. This leads us to reinterpret the contact angle hysteresis as a dynamical cross-over rather than a depinning transition.

¹ANR SMART and REALWET

KP1.00104 Numerical simulation of convective evaporation of a droplet on a porous surface¹, MOONHYEOK CHOI, GIHUN SON, Sogang Univ — Numerical simulation is performed for droplet evaporation on a porous surface under an external flow condition. The droplet interface is tracked by a level-set (LS) method, which is modified to include the effects of porosity and evaporation coupled to heat and mass transfer. The conservation equations of mass, momentum, energy and vapor fraction for the external fluid region are combined with the local volume averaged conservation equations for the porous region through the matching conditions of velocity, pressure, temperature and vapor fraction at the fluid-solid interface. The temperature and the vapor fraction at the liquid-gas 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 numerical simulation demonstrates the droplet penetration into the porous region and the evaporation to the porous and external flow regions. The effects of external flow velocity, porosity and porous particle size on the droplet deformation and evaporation are investigated.

¹This research was supported by the Agency for Defense Development.

KP1.00105 Numerical analysis of bubble-cluster formation in an ultrasonic field¹

, DONGHYUN KIM, GHIN SON, Sogang Univ — Bubble-cluster formation in an ultrasonic field is investigated numerically solving the conservation equations of mass, momentum and energy. The liquid-gas interface is calculated using the volume-of-fluid method with variable gas density to consider the bubble compressibility. The effect of liquid-gas phase change is also included as the interface source terms of the mass and energy equations. The numerical approach is tested through the simulation of the expansion and contraction motion of a compressed bubble adjacent to a wall. When the bubble is placed in an ultrasonic field, it oscillates radially and then collapses violently. Numerical simulation is also performed for bubble-cluster formation induced by an ultrasonic generator, where the generated bubbles are merged into a macrostructure along the acoustic flow field. The effects of ultrasonic power and frequency, liquid properties and pool temperature on the bubble-cluster formation are investigated.

¹This work was supported by the Korea Institute of Energy Research

KP1.00106 Macromolecular Origins of Harmonics Higher than the Third in Large-Amplitude Oscillatory Shear Flow¹ , ALAN GIACOMIN, LAYAL JBARA, PETER GILBERT, Queen's University, CHEMICAL ENGINEERING DEPARTMENT TEAM — In 1935, Andrew Gemant conceived of the complex viscosity, a rheological material function measured by "jiggling" an elastic liquid in oscillatory shear [*Rheol. Acta*, **51**, 481 (2012)]. This test reveals information about both the viscous and elastic properties of the liquid, and about how these properties depend on frequency. The test gained popularity with chemists when John Ferry perfected instruments for measuring both the real and imaginary parts of the complex viscosity [*Mem. Trib., NAE*, **17**, 96 (2013)]. In 1958, Cox and Merz discovered that the steady shear viscosity curve was easily deduced from the magnitude of the complex viscosity, and today oscillatory shear is the single most popular rheological property measurement. With oscillatory shear, we can control two things: the frequency (Deborah number) and the shear rate amplitude (Weissenberg number). When the Weissenberg number is large, the elastic liquids respond with a shear stress over a series of odd-multiples of the test frequency. In this lecture we will explore recent attempts to deepen our understand of the physics of these higher harmonics, including especially harmonics higher than the third.

¹Canada Research Chairs program of the Government of Canada for the Natural Sciences and Engineering Research Council of Canada (NSERC) Tier 1 Canada Research Chair in Rheology

KP1.00107 Flow Characteristics of Lid-Driven Cavities with Particle Suspensions using an Eulerian-Lagrangian Modeling Approach , MORAKINYO ADESEMOWO, JOHN SHELTON, Northern Illinois University — Previous experimental and numerical investigations involving lid-driven cavity flows with particle suspensions have primarily focused on particle tracking and the visualization of complex three-dimensional structures that compose the flow field. However, these particle suspensions and their resulting particle-particle interactions could also be viewed as a system of time-dependent perturbation equations to the steady-state Navier-Stokes equations and could affect both the stability and steady-state characteristics of the two-dimensional lid-driven cavity system. In this investigation, an Eulerian-Lagrangian approach to modeling particle suspensions in the lid-driven cavity is utilized in FV-CFD simulations to investigate the effect particle density, area fraction, and Reynolds number have on the two-dimensional flow characteristics of a laminar fluid. Observations have indicated that the development of the primary vortex in the lid-driven cavity varies according to the area fraction of particle suspensions in the system; transitioning from development via an adverse pressure gradient at the top-right corner of the cavity towards particle-laden behavior where particle-particle interactions dominate the development of the primary vortex. Dynamic responses were also observed for particle systems of less dense particles. Finally, a comparison between flows perturbed using disturbance velocities and from particle interactions was performed.

KP1.00108 Interactions between mean flow and turbulence in the 2D condensate , CORENTIN HERBERT, Ecole Normale Supérieure de Lyon, GREGORY FALKOVICH, ANNA FRISHMAN, Weizmann Institute of Science — Understanding the interaction of a mean flow with turbulent fluctuations is a central problem in turbulence theory. Here, we shall tackle this issue in the framework of incompressible 2D turbulence in a finite box. In the presence of small-scale energy injection and small large-scale friction, the inverse cascade of energy leads to a stationary state made of a pair of coherent vortices, upon which incoherent turbulent fluctuations are superimposed. Due to the time scale separation between the mean-flow and turbulence, an asymptotic expansion of the hierarchy of moments can be carried out to obtain closed equations describing both the mean flow and the fluctuations profiles. Using extensive numerical simulations, we will test the validity of these analytical predictions. In particular, we will discuss how the components of the Reynolds stress tensor scale with both distance from vortex core and large scale friction, which is the small parameter in the theory.

KP1.00109 Unifying Rules for Aquatic Locomotion , MEHDI SAADAT, Harvard University / University of South Carolina, AUGUST DOMEL, VALENTINA DI SANTO, GEORGE LAUDER, Harvard University, HOSSEIN HAJ-HARIRI, University of South Carolina — Strouhal number, $St (=fA/U)$, a scaling parameter that relates speed, U , to the tail-beat frequency, f , and tail-beat amplitude, A , has been used many times to describe animal locomotion. It has been observed that swimming animals cruise at $0.2 \leq St \leq 0.4$. Using simple dimensional and scaling analyses supported by new experimental evidence of a self-propelled fish-like swimmer, we show that when cruising at minimum hydrodynamic input power, St is predetermined, and is only a function of the shape, i.e. drag coefficient and area. The narrow range for St , 0.2-0.4, has been previously associated with optimal propulsive efficiency. However, St alone is insufficient for deciding optimal motion. We show that hydrodynamic input power (energy usage to propel over a unit distance) in fish locomotion is minimized at all cruising speeds when $A^* (=A/L)$, a scaling parameter that relates tail-beat amplitude, A , to the length of the swimmer, L , is constrained to a narrow range of 0.15-0.25. Our analysis proposes a constraint on A^* , in addition to the previously found constraint on St , to fully describe the optimal swimming gait for fast swimmers. A survey of kinematics for dolphin, as well as new data for trout, show that the range of St and A^* for fast swimmers indeed are constrained to 0.2-0.4 and 0.15-0.25, respectively. Our findings provide physical explanation as to why fast aquatic swimmers cruise with relatively constant tail-beat amplitude at approximately 20 percent of body length, while their swimming speed is linearly correlated with their tail-beat frequency.

KP1.00110 Unsteady adjoint for large eddy simulation of a coupled turbine stator-rotor system , CHAITANYA TALNIKAR, QIQI WANG, Massachusetts Inst of Tech-MIT, GREGORY LASKOWSKI, GE Aviation — Unsteady fluid flow simulations like large eddy simulation are crucial in capturing key physics in turbomachinery applications like separation and wake formation in flow over a turbine vane with a downstream blade. To determine how sensitive the design objectives of the coupled system are to control parameters, an unsteady adjoint is needed. It enables the computation of the gradient of an objective with respect to a large number of inputs in a computationally efficient manner. In this paper we present unsteady adjoint solutions for a coupled turbine stator-rotor system. As the transonic fluid flows over the stator vane, the boundary layer transitions to turbulence. The turbulent wake then impinges on the rotor blades, causing early separation. This coupled system exhibits chaotic dynamics which causes conventional adjoint solutions to diverge exponentially, resulting in the corruption of the sensitivities obtained from the adjoint solutions for long-time simulations. In this presentation, adjoint solutions for aerothermal objectives are obtained through a localized adjoint viscosity injection method which aims to stabilize the adjoint solution and maintain accurate sensitivities. Preliminary results obtained from the supercomputer Mira will be shown in the presentation.

KP1.00111 High order discontinuous Galerkin discretizations with discontinuity resolution within the cell, JOHN EKATERINARIS, KONSTANTINOS PANOURGIAS, Embry Riddle Aeronautical University — The nonlinear filter of Yee et al. and used for low dissipative well-balanced high order accurate finite-difference schemes is adapted to the finite element context of discontinuous Galerkin (DG) discretizations. The performance of the proposed nonlinear filter for DG discretizations is demonstrated for different orders of expansions for one- and multi-dimensional problems with exact solutions. It is shown that for higher order discretizations discontinuity resolution within the cell is achieved and the design order of accuracy is preserved. The filter is applied for inviscid and viscous flow test problems including strong shocks interactions to demonstrate that the proposed dissipative mechanism for DG discretizations yields superior results compared to the results obtained with the TVB limiter and high-order hierarchical limiting [2]. The proposed approach is suitable for p-adaptivity in order to locally enhance resolution of three-dimensional flow simulations. [1] H.C. Yee, N.D. Sandham, M.J. Djomehri, J. Comput. Phys., 150 (1999) 199-238. [2] K. Panourgiyas, J.A. Ekaterinaris, Comput. Methods Appl. Mech. Engrg., 299 (2016) 254-282.

KP1.00112 Simultaneous fingering, double-diffusive convection, and thermal plumes derived from autocatalytic exothermic reaction fronts¹, MATTHEW W. ESKEW, JASON HARRISON, REUBEN H. SIMOYI², Portland State University — Oxidation reactions of thiourea by chlorite in a Hele-Shaw cell are excitable, autocatalytic, exothermic, and generate a lateral instability upon being triggered by the autocatalyst. Reagent concentrations used to develop convective instabilities delivered a temperature jump at the wave front of 2.1 K. The reaction zone was 2 mm and due to normal cooling after the wave front, this generated a spike rather than the standard well-studied front propagation. The reaction front has solutal and thermal contributions to density changes that act in opposite directions due to the existence of a positive isothermal density change in the reaction. The competition between these effects generates thermal plumes. The fascinating feature of this system is the coexistence of plumes and fingering in the same solution which alternate in frequency as the front propagates, generating hot and cold spots within the Hele-Shaw cell, and subsequently spatiotemporal inhomogeneities. The small ΔT at the wave front generated thermocapillary convection which competed effectively with thermogravitational forces at low Eötvös Numbers. A simplified reaction-diffusion- convection model was derived for the system. Plume formation is heavily dependent on boundary effects from the cell dimensions.

¹This work was supported by Grant no. CHE-1056366 from the NSF and a Research Professor grant from the University of KwaZulu-Natal.

²University of KwaZulu-Natal, Westville Campus

KP1.00113 Number of Packages of Information which are processed in a Second by the Fundamental Particles (strings) of a Human Body, HASSAN GHOLIBEIGIAN¹, Retired, GHASEM GHOLIBEIGIAN, KAZEM GHOLIBEIGIAN², None — The fundamental particle (string) gets a package of complete information of its quantum state via inside of its sub-particle (sub-string) from dimension of information. This package is processed by sub-particle in each Planck time [Gholibegian, APS 2015, abstract #L1.027]. On the other hand, a 70 kg human's body would have approximately 7×10^{27} atoms. Of that, 4.7×10^{27} would be hydrogen atoms. Another 1.8×10^{27} would be oxygen and there are 7.0×10^{26} carbon atoms. If we add that all up, total is 2.3×10^{28} protons, 1.8×10^{28} neutrons, and 2.3×10^{28} electrons. Each proton and neutron has 6 fundamental particles. So the total number of packages of information which are processed by each of us in a second becomes: $I = [6 \times (2.3 + 1.8) \times 10^{28} + 2.3 \times 10^{28}] \times 10^{44} = 2.69 \times 10^{73}$ packages. The processed information carry by fundamental particles. Based on Shanon equation, $I = -S$, this number can be equal to the increased entropy of each of us per second too.

¹AmirKabir University of Technology, Tehran, Iran.

²Technische Universitat (TU), Vienna, Austria.

KP1.00114 The Generalized Onsager Model and DSMC Simulations of High-Speed Rotating Flow with Swirling Feed, DR. SAHADEV PRADHAN, Department of Chemical Engineering, Indian Institute of Science, Bangalore- 560 012, India — The generalized Onsager model for the radial boundary layer and of the generalized Carrier-Maslen model for the axial boundary layer at the end-caps in a high-speed rotating cylinder ((S. Pradhan & V. Kumaran, J. Fluid Mech., 2011, vol. 686, pp. 109-159); (V. Kumaran & S. Pradhan, J. Fluid Mech., 2014, vol. 753, pp. 307-359)), are extended to incorporate the angular momentum of the feed gas for a swirling feed for single component gas and binary gas mixture. For a single component gas, the analytical solutions are obtained for the sixth-order generalized Onsager equations for the master potential, and for the fourth-order generalized Carrier-Maslen equation for the velocity potential. In both cases, the equations are linearized in the perturbation to the base flow, which is a solid-body rotation. The equations are restricted to the limit of high Reynolds number and (length/radius) ratio, but there is no limitation on the stratification parameter. The linear operators in the generalized Onsager and generalized Carrier-Maslen equations with swirling feed are still self-adjoint, and so the eigenfunctions form a complete orthogonal basis set. The analytical solutions are compared with direct simulation Monte Carlo (DSMC) simulations. The comparison reveals that the boundary conditions in the simulations and analysis have to be matched with care. When these precautions are taken, there is excellent agreement between analysis and simulations, to within 15%.

KP1.00115 The scaling transition of Nu number and boundary layer thickness in RB convection, HONG-YUE ZOU, SKLTCS, COE, Peking Univ., XI CHEN, Department of Mechanical Engineering, Texas Tech Univ., ZHEN-SU SHE, SKLTCS, COE, Peking Univ. — A quantitative theory is developed for the vertical mean temperature profile (MTP) and mean velocity profile (MVP) in turbulent Rayleigh-Benard Convection (RBC), which explains the experimental and numerical observations of logarithmic law in MTP and the Rayleigh number (Ra)-dependence of its coefficient A. The theory extends a symmetry analysis of canonical wall-bounded turbulent flows, which allows to extract accurate Ra scaling of the sub-layer, buffer layer and log-layer thicknesses from the empirical data over a wide range of Ra. In particular, the scaling of the multi-layer thicknesses predicts that the log-law coefficient A follows a -0.121 scaling, which agrees well with the experimental data. More interestingly, a scaling transition is discovered for the kinetic sublayer thickness around Ra of 1010, which yields a scaling transition of Nu from 1/3 to 0.38. We also develop a new explanation for mean temperature logarithmic law: the effect of inverse pressure gradient drives plumes upwards near the side wall, and yields a similarity between temperature and momentum transport in the vertical direction.

KP1.00116 An immersed boundary method for two-phase fluids and gels and the swimming of *Caenorhabditis elegans* through viscoelastic fluids¹, PILHWA LEE, University of Michigan, CHARLES WOLGEMUTH, University of Arizona — While swimming in Newtonian fluids has been examined extensively, only recently have investigations into microorganism swimming through non-Newtonian fluids and gels been explored. The equations that govern these more complex media are often nonlinear and require computational algorithms to study moderate to large amplitude motions of the swimmer. Here we develop an immersed boundary method for handling fluid-structure interactions in a general two-phase medium, where one phase is a Newtonian fluid and the other phase is viscoelastic. We use this algorithm to investigate the swimming of an undulating, filamentary swimmer in 2D. A novel aspect of our method is that it allows one to specify how forces produced by the swimmer are distributed between the two phases of the fluid. The algorithm is validated by comparison to theoretical predictions for small amplitude swimming in gels and viscoelastic fluids. We show how the swimming velocity depends on material parameters of the fluid and the interaction between the fluid and swimmer. In addition, we simulate the swimming of *Caenorhabditis elegans* in viscoelastic fluids and find good agreement between the swimming speeds and fluid flows in our simulations and previous experimental measurements.

¹NIH R01 GM072004, NIH P50GM094503

KP1.00117 Inherently Unstable Internal Gravity Waves, REZA ALAM¹, University of California, Berkeley — Here we show that there exist internal gravity waves that are inherently unstable, that is, they cannot exist in nature for a long time. The instability mechanism is a one-way (irreversible) harmonic-generation resonance that permanently transfers the energy of an internal wave to its higher harmonics. We show that, in fact, there are countably infinite number of such unstable waves. For the harmonic-generation resonance to take place, nonlinear terms in the free surface boundary condition play a pivotal role, and the instability does not obtain for a linearly-stratified fluid if a simplified boundary condition such as rigid lid or linear form is employed. Harmonic-generation resonance discussed here also provides a mechanism for the transfer of the energy of the internal waves to the higher-frequency part of the spectrum where internal waves are more prone to breaking, hence losing energy to turbulence and heat and contributing to oceanic mixing.

¹Yong Liang (yong.liang@berkeley.edu)

KP1.00118 Braids in a two-body micro swimming, MEHDI MIRZAKHANLOO, MIR ABBAS JALALI, M.-REZA ALAM, University of California, Berkeley — Here we show that microswimmers' trajectories may get entangled as a result of their mutual hydrodynamic interactions, resulting in a group behavior that is significantly different from individual swimmers' trajectories. Specifically, we consider a two-swimmer motion of "Quadroar", a newly proposed swimmer consists of two axles of rotating disks connected through a linear reciprocating actuator. In the absence of hydrodynamic interaction, each microswimmer moves along a straight path. When hydrodynamic interaction is introduced, the two swimmers move along tightly woven trajectories whose properties depend on the swimmers' initial conditions. We also show that if swimmers are sent toward each other they may reach an equilibrium at which while they are swimming (i.e. spending energy) no net motion is achieved. We further discuss that since the streamlines of the flow induced by the Quadroar closely resemble the oscillatory flow field of the green alga Chlamydomonas reinhardtii, our findings can thus be utilized to understand the interactions of microorganisms with each other.

KP1.00119 Removal of bio-aerosols by water flow on surfaces in health-care settings, HAN YU, YUGUO LI, The Univ of Hong Kong — Hand hygiene is one of the most important and efficient measures to prevent infections, however the compliance with hand hygiene remains poor especially for health-care workers.¹ To improve this situation, the mechanisms of hand cleansing need to be explored and a detailed study on the adhesion interactions for bio-aerosols on hand surfaces and the process during particles removal by flow is significant for more efficient methods to decrease infections. The first part of presentation will focus on modelling adhesion interactions between particles, like bacteria and virus, and hand surfaces with roughness in water environment. The model presented is based on the DLVO and its extended theories. The removal process comes next, which will put forward a new model to describe the removal of particles by water flow. In this model, molecular dynamics is combined with particle motion and the results by the model will be compared with experiment results and existed models (RnR, Rock & Roll). Finally, possible improvement of the study and future design of experiments will be discussed.

¹World Health Organization. WHO Guidelines on Hand Hygiene in Health Care. 2009

KP1.00120 Prediction of local losses of low Re flows in elastic porous media, SID BECKER, University of Canterbury, STEFAN GASOW, Technical University of Hamburg — An isotropic elastic porous structure whose pore scale geometry is regular (periodically uniform) will experience non-uniform deformation when a viscous fluid flows through the matrix under the influence of an externally applied pressure difference. In such a case, the flow field will experience a non uniform pressure gradient whose magnitude increases in the direction of bulk flow. In this study, a method is presented that predicts local losses of the flow through a porous matrix whose geometry varies in the direction of flow. Employing an asymptotic expansion about the deformation provides an expression relating local hydraulic permeability to local pore geometry. In this way the pressure field is able to be determined without requiring the explicit solution of the flow field. In this study a test case is presented showing that the local pressure losses are predicted to be within 0.5% those of the solution to the Navier-Stokes Equations. The approach can be used to simplify the coupled fluid-solid problem of flow through elastic porous media by replacing the need to explicitly solve the flow field.

KP1.00121 Experimental study of filament break-off of dense suspensions, GUSTAF MRTENSSON, Chalmers University of Technology, Sweden, Mycronic AB, Sweden, FABIAN CARSON, Mycronic AB, Sweden — 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 establish a deeper understanding of the break-off process of filaments of dense suspensions. The experimental set-up consists of a filament break-off device (FiBO) developed in-house. The suspension samples consist of a resin-based flux and spherical particles with diameters of $d_p = 10 - 25\mu\text{m}$. A cylindrical sample ($d_{\text{cyl}} = 1 \text{ mm}$ and $h_{\text{cyl}} = 1 \text{ mm}$) of the suspension is extended using a cylindrical probe travelling between $v_{\text{cyl}} = 100 - 800 \text{ mm/s}$ in the vertical direction. A decrease in particle size correlates with increasing break-off length. Further results relating break-off length and rate versus particle diameter, volume fraction and probe speed will be presented. Comparisons of the filament break-off experiments with practical jetting of the suspensions will be presented.

KP1.00122 A Quasi-Dynamic Approach to modelling Hydrodynamic Focusing, ADITYA

KOMMAJOSULA, SONGZHE XU, Iowa State Univ, CHUEH-YU WU, DINO DI CARLO, University of California, Los Angeles, BASKAR GANAPATHYSUBRAMANIAN, Iowa State Univ, COMPM LAB TEAM, DI CARLO LAB COLLABORATION — We examine a particle's tendency at different spatial locations to shift/rotate towards the equilibrium location, by constrained simulation. Although studies in the past have used this procedure in conjunction with FSI methods to great effect, the current work in 2D explores an alternative approach by utilizing a modified trust-region-based root-finding algorithm to solve for particle position and velocities at equilibrium, using "snapshots" of finite-element solutions to the steady-state Navier-Stokes equations iteratively over a computational domain attached to the particle reference frame. Through an assortment of test cases comprising circular and non-circular particle geometries, an incorporation of stability theory as applicable to dynamical systems is demonstrated, to locate the final focusing location and velocities. The results are compared with previous experimental/numerical reports, and found to be in close agreement. A thousand-fold increase is observed in computational time for the current workflow from its transient counterpart, for an illustrative case. The current framework is formulated in 2D for 3 Degrees-of-Freedom, and will be extended to 3D. This framework potentially allows for quick, high-throughput parametric space studies of equilibrium scaling laws.

KP1.00123 The Effect of Elasticityon the Extrudate Swell of Molten Polymers, SAVVAS HATZIKIRIAKOS, VINOD KUMAR KONAGANTI, The University of British Columbia, UBC TEAM

The extrudate swell of an industrial grade high molecular weight high-density polyethylene (HDPE) in capillary dies is studied using the integral K-BKZ constitutive model. The non-linear viscoelastic flow properties of the polymer resin are studied for a broad range of large step shear strains and high shear rates using the cone partitioned plate (CPP) geometry of the stress/strain controlled rotational rheometer. This allowed the determination of the rheological parameters accurately, in particular the damping function, which is proven to be the most important in simulating transient flows such as extrudate swell. A series of simulations performed using the integral K-BKZ Wagner model with different values of the Wagner exponent n , ranging from $n = 0.15$ to 0.5 , demonstrates that the extrudate swell predictions are extremely sensitive to the Wagner damping function exponent. Using the correct n -value resulted in extrudate swell predictions that are in excellent agreement with experimental measurements.

KP1.00124 Performance of a reduced-order FSI model for flow-induced vocal fold vibration¹, SIYUAN CHANG, HAOXIANG LUO, Vanderbilt University, LUO'S LAB TEAM — Vocal fold vibration during speech production involves a three-dimensional unsteady glottal jet flow and three-dimensional nonlinear tissue mechanics. A full 3D fluid-structure interaction (FSI) model is computationally expensive even though it provides most accurate information about the system. On the other hand, an efficient reduced-order FSI model is useful for fast simulation and analysis of the vocal fold dynamics, which is often needed in procedures such as optimization and parameter estimation. In this work, we study the performance of a reduced-order model as compared with the corresponding full 3D model in terms of its accuracy in predicting the vibration frequency and deformation mode. In the reduced-order model, we use a 1D flow model coupled with a 3D tissue model. Two different hyperelastic tissue behaviors are assumed. In addition, the vocal fold thickness and subglottal pressure are varied for systematic comparison. The result shows that the reduced-order model provides consistent predictions as the full 3D model across different tissue material assumptions and subglottal pressures. However, the vocal fold thickness has most effect on the model accuracy, especially when the vocal fold is thin.

¹Supported by the NSF.

KP1.00125 Bringing first-hand experience one step closer to theoretical understanding¹, HAOXIANG LUO, Vanderbilt University, MIKE MYERS, Oregon Institute of Technology, LUO'S LAB TEAM — Many theoretical concepts and analytical approaches in fluid mechanics are challenging to teach. Classroom demos are very useful to engage and motivate students, but they do not necessarily lead straightforwardly to higher level understanding of model abstraction that is expressed with mathematical equations. To facilitate the process, we have designed a few demos and integrated them with quantitative measurements and theoretical analysis. These demos, usually generated from daily life examples, are of low cost and simple to implement, and the experimental procedures do not take significant time in a 50-min lecture. When combining them with classroom interactions, problem solving, and discussions, we found that these modules are effective in helping students in the learning process.

¹Supported by the NSF.

KP1.00126 Electrorotation instability of a liquid bridge¹, GERARDO PRADILLO, PETIA VLAHOVSKA, Brown University — We experimentally study the behavior of an oil droplet sandwiched between two ITO coated glass electrodes in a DC electric field. Above a threshold electric field, the contact line contour changes from circular to a steady flower-like one. As the field strength increases, the wave becomes unsteady and begins to rotate. It is observed that the stable mode of the wavy contact line is different if the field is applied as a pulse or gradually increased in small steps, which suggests that the charge relaxation time is important. The amplitude of the wave is related to the conductivity of the drop fluid, which is controlled by adding organic electrolytes. The physical mechanism of the instability is yet to be explained.

¹supported by NSF award 1437545

replacing DFD16-2016-020106.

KP1.00127 Tunable Superomniphobic Surfaces for Sorting Droplets by Surface Tension, SANLI MOVAFAGHI, WEI WANG, ARI METZGER, DESIREE WILLIAMS, JOHN WILLIAMS, ARUN KOTA, Colorado State University — Manipulation of liquid droplets on super-repellent surfaces (i.e., surfaces that are extremely repellent to liquids) has been widely studied because droplets exhibit high mobility on these surfaces due to the ultra-low adhesion, which leads to minimal sample loss and contamination. Although droplet manipulation has been demonstrated using electric fields, magnetic fields, guiding tracks and wettability gradients, to the best of our knowledge, there are no reports of droplet manipulation methods that can sort droplets by surface tension on super-repellent surfaces. In this work, we utilized tunable superomniphobic surfaces (i.e., surfaces that are extremely repellent to virtually all liquids) to develop a simple device with precisely tailored solid surface energy domains that, for the first time, can sort droplets by surface tension. Droplet sorting occurs on our device entirely due to a balance between the work done by gravity and the work expended due to adhesion, without the need for any external energy input. Our device can be fabricated easily in a short time and is particularly useful for in-the-field and on-the-go operations, where complex analysis equipment is unavailable. We envision that our methodology for droplet sorting will enable inexpensive and energy-efficient analytical devices for personalized point-of-care diagnostic platforms and lab-on-a-chip systems.

KP1.00128 Free-standing, Flexible Superomniphobic Films¹, HAMED VAHABI, WEI WANG, SANLI MOVAFAGHI, ARUN K.KOTA, Colorado State Univ — Fabrication of most superomniphobic surfaces requires complex process conditions or specialized and expensive equipment or skilled personnel. In order to circumvent these issues and make them end-user friendly, we developed the free-standing, flexible superomniphobic films. These films can be stored and delivered to the end-users, who can readily attach them to virtually any surface (even irregular shapes) and impart superomniphobicity. The hierarchical structure, the re-entrant texture and the low solid surface energy render our films superomniphobic for a wide variety of liquids. We demonstrate that our free-standing, flexible, superomniphobic films have applications in enhanced chemical resistance and enhanced weight bearing.

¹We thank Colorado Office of Economic Development and International Trade for financial support under award EDA 14-246.

KP1.00129 Effect of Fluid Structure Interaction on the Wake Structure of a Thin Flexible Cylinder, HARIBA GURRAM, CHELAKARA SUBRAMANIAN, PRIYANKA KANHERKAR, Florida Institute of Technology — Previous studies by the authors of the drag coefficient for thin flexible cylinders (diameter ~0(mm)) in a cross flow for Reynolds's numbers range between 100-1000 showed about 20 – 30 percent reduction compared to literature values. At free stream low Reynolds number around 100 the spectral analysis of the hotwire signals in the wake showed tonal and broadband frequencies suggesting features similar to transition flows. To better understand the flow behind the cylinder and wake structure interaction with boundary layer for above range of Reynolds number DNS simulations were conducted. The computational study is performed for two cases: (1) flow on rigid thin cylinder, and (2) flow with 3-D fluid structure interaction for the thin cylinder. It is observed the coefficient of drag values computed for the rigid wire were 8-12 percent lower compared to the experimental results, while simulation with the fluid structure interaction gave results within 4 percent of the experimental values. The wake structure results based on the experiment and computational study will be discussed.

KP1.00130 Nonlinear Acoustics at the Air-Water Free Surface¹, SETH PREE, BRIAN NARANJO, SETH PUTTERMAN, Univ of California - Los Angeles — According to linear acoustics, airborne sound incident on a water surface transmits only a tenth of a percent of its energy. This difficulty of transmitting energy across the water surface limits the feasibility of standoff ultrasound imaging. We propose to overcome this long standing problem by developing new methods of coupling into the medium at standoff. In particular, we believe that the acoustic nonlinearity of both the air and the medium may yield a range of effects in the vicinity of the surface permitting an efficient transmission of ultrasound from the air into the medium. The recent commercial availability of parametric speakers that deliver modulated 100kHz ultrasound at 135dB to nonlinearly generate music at 95dB provides an interesting platform with which to revisit the transmission of sound across acoustic impedance mismatches. We show results of experimental studies of the behavior of the air-water free surface when subjected to large amplitude acoustic pressures from the air.

¹This work was supported by the ARO STIR program.

KP1.00131 Effect of angle of attack on the flow past a harbor seal vibrissa shaped cylinder¹, HYO JU KIM, HYUN SIK YOON, Pusan National University, Korea — The present study considered the geometric disturbance inspired by a harbor seal vibrissa of which undulated surface structures are known as a detecting device to capture the water movement induced by prey fish. In addition, this vibrissa plays an important role to suppress vortex-induced vibration, which has been reported by the previous researches. The present study aims at finding the effect of the angle of attack (AOA) on flow characteristics around the harbor seal vibrissa shaped cylinder, since the flow direction facing the harbor seal vibrissa with the elliptic shape can be changed during the harbor seal's movements and surrounding conditions. Therefore, we considered a wide range of AOA varying from 0 to 90 degree. We carried out large eddy simulation (LES) to investigate the flow around inclined vibrissa shaped cylinder for the Reynolds number (Re) of 500 based hydraulic diameter of a harbor seal vibrissa shape. For comparison, the flow over the elliptic cylinder was also simulated according to AOA at the same Re. The vortical structures of both vibrissa shaped and elliptic cylinders have been compared to identify the fundamental mechanism making the difference flow quantities.

¹This subject is supported by Korea Ministry of Environment (MOE) as the Chemical Accident Prevention Technology Development Project., National Research Foundation of Korea (NRF) grant through GCRCSOP (No.20110030013) and (NRF-2015R1D1A3A01020867).

KP1.00132 Elasticity-Driven Backflow of Fluid-Driven Cracks¹, CHING-YAO LAI, ZHONG ZHENG, Department of Mechanical and Aerospace Engineering, Princeton University, USA, EMILIE DRESSAIRE, Department of Mechanical and Aerospace Engineering, New York University Tandon School of Engineering, USA, GUY RAMON, Department of Civil and Environmental Engineering, Technion - Israel Institute of Technology, Israel, HERBERT E. HUPPERT, Institute of Theoretical Geophysics, Department of Applied Mathematics and Theoretical Physics, University of Cambridge, UK, HOWARD A. STONE, Department of Mechanical and Aerospace Engineering, Princeton University, USA — Fluid-driven cracks are generated by the injection of pressurized fluid into an elastic medium. Once the injection pressure is released, the crack closes up due to elasticity and the fluid in the crack drains out of the crack through an outlet, which we refer to as backflow. We experimentally study the effects of crack size, elasticity of the matrix, and fluid viscosity on the backflow dynamics. During backflow, the volume of liquid remaining in the crack as a function of time exhibits a transition from a fast decay at early times to a power law behavior at late times. Our results at late times can be explained by scaling arguments balancing elastic and viscous stresses in the crack. This work may relate to the environmental issue of flowback in hydraulic fracturing.

¹This work is supported by National Science Foundation via grant CBET-1509347 and partially supported by Andlinger Center for Energy and the Environment at Princeton University.

KP1.00133 Time-Reversal of Nonlinear Water Waves, AMIN CHABCHOUB, Department of Mechanical Engineering, School of Engineering, 02150 Espoo, Finland, GUILLAUME DUCROZET, LHEEA, cole Centrale Nantes, UMR CNRS No. 6598, 1 rue de la No, 44321 Nantes, France, MATHIAS FINK, Institut Langevin, ESPCI Paris, PSL University, CNRS, UMR CNRS No. 7587, 10 rue Vauquelin, 75005 Paris, France — Time-reversal (TR) refocusing of hydrodynamic nonlinear waves can be discussed within the framework of the nonlinear Schrödinger equation (NLS). Indeed, exact solutions of the latter weakly nonlinear evolution equation can be used to study the applicability and limitations of wave refocusing using TR mirrors in hydrodynamics. Recent laboratory experiments confirmed the applicability of TR approach to breathers, known to model extreme and doubly-localized wave configurations. In order to study the range of validity of the TR approach to nonlinear waves, a numerical study using a unidirectional numerical water wave tank, implemented by the higher-order spectral method, reveals new insights to the problem. The validity of the TR approach is assessed over a diversity of NLS configurations, ranging from stationary envelope and breathing solutions, pointing out the importance of higher-order dispersive and particularly nonlinear effects in the refocusing of these hydrodynamic localized structures. Due to the interdisciplinary nature of the approach several applications in other nonlinear dispersive physical media may result in addition to evident usage in the field of ocean engineering.

KP1.00134 Reducing spin-up time for DNS and LES of turbulent channel flow¹, KURT NELSON, OLIVER FRINGER, Stanford Univ — In DNS or LES of turbulent channel flow, significant computational resources are wasted on simulation of flow evolution as it approaches statistical equilibrium. Although the driving pressure gradient that produces the desired time-averaged bottom stress is known a-priori, during flow spin-up this pressure gradient is typically not in balance with the time-averaged bottom stress, leading to flow acceleration beyond the target velocity which can significantly prolong the time to reach statistical equilibrium. Through DNS of turbulent channel flow with $Re_\tau = 500$, we present a method that ensures a time invariant volume-averaged streamwise velocity. While the method eliminates spin-up time related to approaching the target volume-averaged velocity, spin-up time is still needed for the turbulence to reach statistical equilibrium. To this end, we study the evolution of the turbulence in response to different initial velocity profiles and initial random perturbations and show that initialization with a laminar velocity profile significantly reduces spin-up time because the linear distribution of vertical shear triggers turbulence faster than it would with a log-law velocity profile.

¹We gratefully acknowledge ONR Grant N00014-15-1-2287

KP1.00135 An engineering closure for heavily under-resolved coarse-grid CFD in large applications, ANDREAS G CLASS, FUJIANG YU, THOMAS JORDAN, Karlsruhe Institute of Technology — Even though high performance computation allows very detailed description of a wide range of scales in scientific computations, engineering simulations used for design studies commonly merely resolve the large scales thus speeding up simulation time. The coarse-grid CFD (CGCFD) methodology is developed for flows with repeated flow patterns as often observed in heat exchangers or porous structures. It is proposed to use inviscid Euler equations on a very coarse numerical mesh. This coarse mesh needs not to conform to the geometry in all details. To reinstall physics on all smaller scales cheap subgrid models are employed. Subgrid models are systematically constructed by analyzing well-resolved generic representative simulations. By varying the flow conditions in these simulations correlations are obtained. These comprehend for each individual coarse mesh cell a volume force vector and volume porosity. Moreover, for all vertices, surface porosities are derived. CGCFD is related to the immersed boundary method as both exploit volume forces and non-body conformal meshes. Yet, CGCFD differs with respect to the coarser mesh and the use of Euler equations. We will describe the methodology based on a simple test case and the application of the method to a 127 pin wire-wrap fuel bundle.

KP1.00136 Pulling rigid bodies through granular material, RYAN KUBIK, EMILIE DRESSAIRE, NYU Tandon Sch of Engr — The need for anchoring systems in granular materials such as sand is present in the marine transportation industry, e.g. to layout moorings, keep vessels and docks fixed in bodies of water, build oil rigs, etc. The holding power of an anchor is associated with the force exerted by the granular media. Empirical evidence indicates that the holding power depends on the size and shape of the anchoring structure. In this model study, we use a two-dimensional geometry in which a rigid body is pulled through a granular media at constant velocity to determine the drag and lift forces exerted by a granular medium on a moving object. The method allows measuring the drag force and recording the trajectory of the rigid object through the sand. We systematically vary the size and geometry of the rigid body, the properties of the granular medium and the extraction speed. For different initial positions of a cylindrical object pulled horizontally through the medium, we record large variations in magnitude of the drag and a significant lift force that pulls the object out of the sand.

KP1.00137 Similarity Solution for High Weissenberg Number Flow of Upper-Convection Maxwell Fluid on a Linearly Stretching Sheet, NARIMAN ASHRAFI, MEYSAM MOHAMADALI, Department of Mechanical Engineering, Payame Noor University, 19395-3697, Tehran, Iran — High Weissenberg boundary layer flow of viscoelastic fluids on a stretching surface has been studied. The flow is considered to be steady, low inertial, and two-dimensional. Upon proper scaling and by means of an exact similarity transformation, the nonlinear momentum and constitutive equations of each layer transform into the respective system of highly nonlinear and coupled ordinary differential equations. Numerical solutions to the resulting boundary value problem are obtained using an efficient shooting technique in conjunction with a variable stepping method for different values of pressure gradients. It is observed that, unlike the Newtonian flows, in order to maintain a potential flow, normal stresses must inevitably develop. The velocity field and stresses distributions over plate are presented for difference values of pressure gradient and Weissenberg numbers.

KP1.00138 Viscoelastic Squeeze Flow, NARIMAN ASHRAFI¹, Department of Mechanical Engineering, Payame Noor University, 19395-3697, Tehran, Iran, MEHDI SHAFAH², Science and Research Branch, Islamic Azad University, Tehran, Iran — The squeeze flow of a nonlinear viscoelastic flow is studied. In particular the flow of an upper-convection Maxwell fluid between two approaching disks is analyzed. The momentum and continuity equations together with constitutive relations are solved by a low-order method. Both no-slip and slip boundary conditions are considered. Next, stress components are evaluated and flow stability is investigated. It is observed that as the disks approach velocity is increased the developed stresses, which are interrelated to velocity gradients through the constitutive relation, are altered exponentially. This analysis is applicable to many industrial instances of such as lubrication as well as natural joints.

¹nariman ashrafi

²Mehdi Shafahi

Monday, November 21, 2016 4:30PM - 6:40PM –
Session L1 Disgust: The Fluid Dynamics of the Gross A105 - Megan Leftwich, George Washington University

4:30PM L1.00001 A numerical investigation of a simplified human birth model, ROSEANNA GOSSMANN, Tulane University, ALEXA BAUMER, The George Washington University, LISA FAUCI, Tulane University, MEGAN C. LEFTWICH, The George Washington University — This work uses a simplified model to explore the forces experienced by the fetus during human birth. Numerical results are compared with the results of a physical 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 a highly viscous fluid. Numerical simulations are run at low and zero Reynolds numbers. In each case, the pulling force necessary to move the rigid inner cylinder at a constant velocity through the tube is measured. The discrete elastic tube through which the rigid cylinder passes has macroscopic elasticity matched to the tube used in the physical experiment. The buckling behavior of the elastic tube is explored by varying velocity, length, and diameter of the rigid cylinder, and length of the elastic tube. 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.

4:43PM L1.00002 An experimental study of human birth models, ALEXA BAUMER, The George Washington University, ROSEANNA GOSSMANN, LISA J FAUCI, Tulane University, MEGAN C. LEFTWICH, The George Washington University — The laboring uterus is a complex and dynamic fluid system. Relatively little is known about the fluid properties in this system.

However, the two primary fluids of interest, amniotic fluid and vernix caseosa, likely play integral roles in the force transferred to the fetus during the final stages of parturition. This investigation probes the role of fluid in the force transfer during delivery by considering physical models that determine the role of various components of the full system. The first experimental model represents the fetus passing through the birth canal as concentric cylinders with a fluid filled gap. The rigid, inner cylinder moves through the highly flexible outer cylinder at a prescribed velocity. The geometry of the inner cylinder is varied by aspect ratio and length. A total of five different inner geometries are used to fully investigate the parameter space. As the inner cylinder moves through the outer cylinder, strain measurements are taken. These measurements are converted to force measurements as a function of time and position in the outer cylinder. The results of these experiments are compared with numerical results to form a more complete picture of force transfer. This model can be used as the foundation for predicting the force needed to deliver a fetus in the final stages of parturition. Additionally, more complex models, that incorporate uterine contraction forces, are being developed.

4:56PM L1.00003 Urethral anatomy and semen flow during ejaculation, DIANE KELLY,

Ronin Institute; University of Massachusetts, Amherst — Ejaculation is critical for reproductive success in many animals, but little is known about its hydrodynamics. In mammals, ejaculation pushes semen along the length of the penis through the urethra. Although the urethra also carries urine during micturition, the flow dynamics of micturition and ejaculation differ: semen is more viscous than urine, and the pressure that drives its flow is derived primarily from the rhythmic contractions of muscles at the base of the penis, which produce pulsatile rather than steady flow. In contrast, Johnston et al. (2014) describe a steady flow of semen through the crocodilian urethral groove during ejaculation. Anatomical differences of tissues associated with mammalian and crocodilian urethral structures may underlie these differences in flow behavior. Ref.Johnston SD et al. Aquaculture 422: 25-35. (2014)

5:09PM L1.00004 Sperm navigation in complex environments¹, SARAH OLSON, Worcester Polytechnic Institute — Sperm can swim in a variety of environments, interacting with chemicals and other proteins in the fluid. Some of these extra proteins or cells may act as friction, possibly preventing or enhancing forward progression of swimmers. The homogenized fluid flow is assumed to be governed by the incompressible Brinkman equation, where a friction term with a resistance parameter represents a sparse array of obstacles. Representing the swimmers with a centerline approximation, we employ regularized fundamental solutions to investigate swimming speeds, trajectories, and interactions of swimmers. Asymmetric waveforms due to an increase in flagellar calcium is known to be important for sperm to reach and fertilize the egg. The trajectories of hyperactivated swimmers are found to have a decreased path curvature. Although attraction of two swimmers is more efficient in the Stokes regime, we find that attraction does not occur for larger resistance. Additionally, we study interactions of swimmers in a channel.

¹NSF DMS 1413110

5:22PM L1.00005 On the need for a biomimetic breast device, NICOLE DANOS, University of San Diego, REBECCA GERMAN, Northeast Ohio Medical University — The function of the mammary gland, a key anatomical innovation that led to the rise of mammals, is governed by solid-fluid mechanics. There is strong evidence that these mechanical interactions regulate the production of milk and the transport of milk through the lactiferous ducts and into the infants mouth. Solid-fluid mechanics determine the rate of milk flow and therefore may affect the safe coordination of sucking, swallowing and breathing in the infant. Additionally, links between breastfeeding, the material properties of the gland and breast cancer have been shown repeatedly. However, there is to date no direct way of characterizing breast mechanics during the physiological function for which it has evolved: infant feeding. We are developing an engineered biomimetic breast in which we can experimentally manipulate both structural and material properties of the gland. The device will be tested with an animal model of infant feeding, the pig, to measure the direct effect of gland mechanics on infant feeding. Data from these studies may lead to better designed feeding bottles for infants, milk pumps for both humans and agricultural mammals, and will provide the control mechanical environmental for studies of breast cancer mechanobiology.

5:35PM L1.00006 Flow and active mixing have a strong impact on bacterial growth dynamics in the proximal large intestine, JONAS CREMER, IGOR SEGOTA, CHIH-YU YANG, MARKUS ARNOLDINI, ALEX GROISMAN, TERENCE HWA, University of California, San Diego — More than half of fecal dry weight is bacterial mass with bacterial densities reaching up to 10^{12} cells per gram. Mostly, these bacteria grow in the proximal large intestine where lateral flow along the intestine is strong: flow can in principle lead to a washout of bacteria from the proximal large intestine. Active mixing by contractions of the intestinal wall together with bacterial growth might counteract such a washout and allow high bacterial densities to occur. As a step towards understanding bacterial growth in the presence of mixing and flow, we constructed an in-vitro setup where controlled wall-deformations of a channel emulate contractions. We investigate growth along the channel under a steady nutrient inflow. Depending on mixing and flow, we observe varying spatial gradients in bacterial density along the channel. Active mixing by deformations of the channel wall is shown to be crucial in maintaining a steady-state bacterial population in the presence of flow. The growth-dynamics is quantitatively captured by a simple mathematical model, with the effect of mixing described by an effective diffusion term. Based on this model, we discuss bacterial growth dynamics in the human large intestine using flow- and mixing-behavior having been observed for humans.

5:48PM L1.00007 Sticky Saliva, LOUISE MCCARROLL, MICHAEL SOLOMON, WILLIAM SCHULTZ, University of Michigan — Oral and even systemic health begins with healthy saliva by maintaining antibacterial activity, lubricating hard and soft oral tissues, healing, tasting, chewing, and swallowing. Saliva functionality is intimately linked to its rheology. Alterations in saliva rheology may indicate or cause unhealthy biological function. One imprecise pathological designation is sticky saliva, usually self-reported or qualitatively described by health professionals. Saliva is 99% water and therefore behaves like water in shear. Saliva also contains mucins, electrolytes, enzymes, hormones, and antibodies. These additional constituents enable saliva to form a long-lasting filament with a beads-on-a-string morphology in extension. Therefore, the main kinematic feature that distinguishes the coupling between the oral cavity and saliva elongational mechanics. We investigate the effect of pH and salinity on saliva filament formation with preliminary experiments and compare to 1D unsteady viscoelastic models. We discuss the results in the context of saliva functionality and in generating more satisfactory saliva substitutes for those suffering from xerostomia. We will discuss when beads-on-a-string are likely to occur.

6:01PM L1.00008 Cat tongue Velcro, ALEXIS NOEL, ANDREA MARTINEZ, HYEWON JUNG, TING-WEN TSAI, DAVID HU, Georgia Institute of Technology, Mechanical Engineering — A cat's tongue is covered in an array of spines called papillae. These spines are thought to be used in grooming and rasping meat from bones of prey, although no mechanism has been given. We use high-speed video to film a cat removing cat food deeply wedged into a 3-D printed fur mat. We show that the spines on the tongue act as Velcro for particles. The tongue itself is highly elastic. As the cat presses it against a substrate, the tongue flattens and the spines separate. When the tongue is removed from the substrate the spines come together, wedging particles between them. This elasticity-driven entrapment permits the surface of the tongue to act as a carrier for hard to reach particles, and to increase the efficacy of grooming and feeding.

6:14PM L1.00009 Self-mixing of fly larvae during feeding, OLGA SHISHKOV, CHRISTOPHER JOHNSON, BRYAN ZHANG, DAVID HU, Georgia Institute of Technology, Mechanical Engineering — How do we sustainably feed a growing world population? One solution of increasing interest is the use of black soldier fly larvae, pea-sized grubs envisioned to transform hundreds of tons of food waste into a sustainable protein source. Although startups across the world are raising these larvae, a physical understanding of how they should be raised and fed remains missing. In this study, we present experiments measuring their feeding rate as a function of number of larvae. We show that larger groups of larvae have greater mixing which entrains hungry larvae around the food, increasing feeding rate. Feeding of larvae thus differs from feeding of cattle or other livestock which exhibit less self-mixing.

6:27PM L1.00010 Boys who pee the farthest have a large hollow head, a thin skin, and medium-size manhood., DANIEL ATTINGER, Iowa State University, VINCENT LEE, Georgia Tech — Following a recent trend of scientific studies on artwork, we study here the thermodynamics of a jetting thermometer made of ceramic, related to the Chinese tea culture. The thermometer represents a boy who “urinates” shortly after hot water is poured onto his head. Long jetting distance indicates if the water temperature is hot enough to brew tea. Here, a thermofluid model describes the jetting phenomenon of that pee-pee boy. The study demonstrates how thermal expansion of an interior air pocket causes jetting. The validity of assumptions underlying the Hagen-Poiseuille flow is discussed for urethra of finite length. A thermodynamic potential is shown to define maximum jetting velocity. Seven optimization criteria to maximize jetting distance are provided, including two dimensionless numbers. The dimensionless numbers are obtained by comparing the time scales of the internal pressure buildup due to heating, with that of pressure relief due to jetting. Optimization results show that longer jets are produced by large individuals, with low body mass index, with a boyhood of medium size inclined at an angle $\pi/4$. Analogies are drawn with peeing contests among humans and lobsters. The study ends by noting similitudes of working principle between that politically incorrect thermometer and Galileo Galilei’s thermoscope. [1] V. Lee and D. Attinger, “Thermodynamics and historical relevance of a jetting thermometer made of Chinese zisha ceramic,” Sci Rep, vol. 6, p. 28609, 2016.

Monday, November 21, 2016 4:30PM - 6:40PM – Session L2 Energy: Wind Power A106 - Mahesh Bandi, OIST Graduate University

4:30PM L2.00001 The Spectrum of Wind Power Fluctuations¹, MAHESH BANDI, OIST Graduate University — Wind is a variable energy source whose fluctuations threaten electrical grid stability and complicate dynamical load balancing. The power generated by a wind turbine fluctuates due to the variable wind speed that blows past the turbine. Indeed, the spectrum of wind power fluctuations is widely believed to reflect the Kolmogorov spectrum; both vary with frequency f as $f^{-5/3}$. This variability decreases when aggregate power fluctuations from geographically distributed wind farms are averaged at the grid via a mechanism known as geographic smoothing. Neither the $f^{-5/3}$ wind power fluctuation spectrum nor the mechanism of geographic smoothing are understood. In this work, we explain the wind power fluctuation spectrum from the turbine through grid scales. The $f^{-5/3}$ wind power fluctuation spectrum results from the largest length scales of atmospheric turbulence of order 200 km influencing the small scales where individual turbines operate. This long-range influence spatially couples geographically distributed wind farms and synchronizes farm outputs over a range of frequencies and decreases with increasing inter-farm distance. Consequently, aggregate grid-scale power fluctuations remain correlated, and are smoothed until they reach a limiting $f^{-7/3}$ spectrum.

¹This work was funded by the Collective Interactions Unit, OIST Graduate University, Japan.

4:43PM L2.00002 Coordinated Control of Cross-Flow Turbines, BENJAMIN STROM, STEVEN BRUNTON, BRIAN POLAGYE, University of Washington — Cross-flow turbines, also known as vertical-axis turbines, have several advantages over axial-flow turbines for a number of applications including urban wind power, high-density arrays, and marine or fluvial currents. By controlling the angular velocity applied to the turbine as a function of angular blade position, we have demonstrated a 79 percent increase in cross-flow turbine efficiency over constant-velocity control. This strategy uses the downhill simplex method to optimize control parameter profiles during operation of a model turbine in a recirculating water flume. This optimization method is extended to a set of two turbines, where the blade motions and position of the downstream turbine are optimized to beneficially interact with the coherent structures in the wake of the upstream turbine. This control scheme has the potential to enable high-density arrays of cross-flow turbines to operate at cost-effective efficiency. Turbine wake and force measurements are analyzed for insight into the effect of a coordinated control strategy.

4:56PM L2.00003 Multiscale simulations of a horizontal axis tidal turbine in a tidal site, ARTURO FERNANDEZ, North Carolina AT State University, ANDRES TEJADA-MARTINEZ, University of South Florida, MARIO JUHA, Rensselaer Polytechnic Institute — We present multiscale simulations of a three-bladed horizontal axis tidal turbine (HATT) in a tidal site. The power and wake generated by the HATT are computed using an Overset Large Eddy Simulation (OLES) technique, where the velocity field is split into a background and a perturbation field. The background flow is a tidal boundary layer with a depth of 10 m, which exhibits a logarithmic distribution for the averaged streamwise velocity. This flow is computed via Large Eddy Simulation for an open channel flow with a time varying pressure gradient. The interaction between this background flow and the HATT is captured using a line actuator model, which generates the perturbation flow representing the turbulent wake. These simulations assume the turbine to have a diameter of 2.5 m and the hub to be located 7.5 m from the bottom floor. The results show the power coefficient to exhibit a cyclical behavior and the power spectral density to have peaks at multiples of one third of the rotational frequency. The turbulent wake exhibits the classical behavior with the near wake dominated by helicoidal vortical structures, and the far wake characterized by diffusion phenomena.

5:09PM L2.00004 ABSTRACT WITHDRAWN –

5:22PM L2.00005 ABSTRACT WITHDRAWN –

5:35PM L2.00006 Leading Edge Vortex Formation of an Oscillating Foil in Energy Harvest Modes, JAMES LIBURDY, FIRAS SIALA, Oregon State University, ALEXANDER TOTPAL, Oregonstate University — The vortex formation number, represents a non-dimensional time scale to achieve its maximum circulation, defined as Γ/cU_{SL} , where Γ is circulation, c is foil chord length, and U_{SL} is the feeding shear-layer velocity. It has been observed that the formation number may be universal, with an optimum value of 4. In this study, phase-locked particle image velocimetry measurements are used to investigate the effect of foil flexibility on the vortex formation number of the leading edge vortex (LEV) of an oscillatory foil in the energy harvesting mode. The goal is to provide insights as to how surface flexibility can enhance energy harvesting efficiencies. Experiments are conducted using three airfoil configurations: fully rigid, flexible trailing edge and flexibility leading edge. The objective is to identify the ability of surface flexibility to enhance the LEV circulation and its associated shear-layer. Results are presented in terms of the vortex formation number for a range of operating reduced frequencies, $k = 0.05 - 0.2$, which reportedly yield the most efficient energy harvesting conditions.

5:48PM L2.00007 The effects of passive foil flexibility on the energy extraction performance of an oscillating foil operating at low reduced frequencies, ALEXANDER D. TOTPAL, FIRAS F. SIALA, JAMES A. LIBURDY, Oregon State University — With a goal to improve energy extraction efficiency from an oscillating foil, direct aerodynamic force measurements are used to study the effect of surface flexibility of an oscillating foil operating in the energy harvesting regime. The experiments are conducted in a closed-loop wind tunnel at a low reduced frequencies range of 0.04 -0.06. The pitching amplitude was varied from 45 to 90 degrees and the phase shift between pitching and heaving motions was varied from 30 to 120 degrees. Three different airfoil configurations were tested: fully rigid, flexible leading edge and flexible trailing edge. In addition, phase-locked particle image velocimetry (PIV) measurements were taken at the higher efficiency cases, and are used to help interpret trends seen in the force measurement data. The timing and position of the leading edge vortex along the foil, which has been shown to be crucial to energy extraction, is investigated in order to help explain why certain operating conditions yield larger efficiencies.

6:01PM L2.00008 Optimal control of wind turbines in a turbulent boundary layer¹, ALI EMRE YILMAZ, JOHAN MEYERS, KU Leuven, Mechanical Engineering, Celestijnenlaan 300A, B3001 Leuven, Belgium — In recent years, optimal control theory was combined with large-eddy simulations to study the optimal control of wind farms and their interaction with the atmospheric boundary layer [1,2]. The individual turbine's induction factors were dynamically controlled in time with the aim of increasing overall power extraction. In these studies, wind turbines were represented using an actuator disk method. In the current work, we focus on optimal control on a much finer mesh (and a smaller computational domain), representing turbines with an actuator line method. Similar to Refs. [1,2], optimization is performed using a gradient-based method, and gradients are obtained employing an adjoint formulation. Different cases are investigated, that include a single and a double turbine case both with uniform inflow, and with turbulent-boundary-layer inflow. [1] Goit Jay, Meyers Johan (2015). Optimal control of energy extraction in wind-farm boundary layers. Journal of Fluid Mechanics 768, 5-50. [2] Goit Jay, Munters Wim, Meyers Johan (2016). Optimal coordinated control of power extraction in LES of a wind farm with entrance effects. Energies 9 (1), art.nr. 29

¹The authors acknowledge support from the European Research Council (FP7-Ideas, grant no. 306471).

6:14PM L2.00009 Optimal control of wind-farm boundary layers: effect of turbine response time¹, WIM MUNTERS, JOHAN MEYERS, KU Leuven — Complex turbine wake interactions play an important role in overall energy extraction in large wind farms. Current control strategies optimize individual turbine power, and lead to significant energy losses in wind farms compared to lone-standing turbines. In recent work, an optimal control framework for dynamic induction control of wind farms and their interaction with the atmospheric boundary layer (ABL) was introduced, with the aim of mitigating such losses. The framework applies a receding horizon methodology, in which the ABL state is modeled through large-eddy simulations. Previously, the framework was applied to both fully-developed (Goit and Meyers 2015, J Fluid Mech, 768, 5–50) and spatially developing wind farms (Goit et al. 2016, Energies, 9, 29), for which respective energy gains of 16% and 7% were obtained, albeit at the cost of additional turbine loading variability. Here, we quantify the trade-off between increased power extraction and smoothed turbine dynamics by varying the turbine response time in the control framework. We consider simulation cases restricted to underinduction compared to Betz-optimal induction, as well as cases that also allow overinduction. In addition, efforts on replicating optimized power gains with practical controllers are presented.

¹The authors are supported by the ERC (ActiveWindFarms, grant no.: 306471)

6:27PM L2.00010 Resonance wave pumping: wave mass transport pumping¹, REMI CARMIGNIANI, ENPC, DAMIEN VIOLEAU, EDF-ENPC, MORTEZA GHARIB, Caltech — It has been previously reported that pinching at intrinsic resonance frequencies a valveless pump (or Liebau pump) results in a strong pulsating flow. A free-surface version of the Liebau pump is presented. The experiment consists of a closed tank with a submerged plate separating the water into a free-surface and a recirculation section connected through two openings at each end of the tank. A paddle is placed at an off-centre position at the free-surface and controlled in a heaving motion with different frequencies and amplitudes. Near certain frequencies identified as resonance frequencies through a linear potential theory analysis, the system behaves like a pump. Particle Image Velocimetry (PIV) is performed in the near free surface region and compared with simulations using Volume of Fluid (VOF) method. The mean eulerian mass flux field (ρu) is extracted. It is observed that the flow is located in the vicinity of the surface layer suggesting Stokes Drift (or Wave Mass Transport) is the source of the pumping. A model is developed to extend the linear potential theory to the second order to take into account these observations.

¹The authors would like to acknowledge the Gordon and Betty Moore Foundation for their generous support

Monday, November 21, 2016 4:30PM - 6:01PM –
Session L3 Vortex Dynamics: Mixed Applications B110-111 - John Farnsworth, University of Colorado, Boulder

4:30PM L3.00001 Application of Biot-Savart Solver to Predict Axis Switching Phenomena in Finite-Span Vortices Expelled from a Synthetic Jet¹, JOSEPH STRACCIA, JOHN FARNSWORTH, University of Colorado, Boulder — The Biot-Savart law is a simple yet powerful inviscid and incompressible relationship between the velocity induced at a point and the circulation, orientation and distance of separation of a vortex line. The authors have developed an algorithm for obtaining numerical solutions of the Biot-Savart relationship to predict the self-induced velocity on a vortex line of arbitrary shape. In this work the Biot-Savart solver was used to predict the self-induced propagation of non-circular, finite-span vortex rings expelled from synthetic jets with rectangular orifices of varying aspect ratios. The solver's prediction of the time varying shape of the vortex ring and frequency of axis switching was then compared with Particle Image Velocimetry (PIV) data from a synthetic jet expelled into a quiescent flow i.e. zero cross flow condition. Conclusions about the effectiveness and limitations of this simple, inviscid relationship are drawn from this experimental data.

¹This material is based upon work supported by the National Science Foundation Graduate Research Fellowship under Grant No. DGE 1144083.

4:43PM L3.00002 Enhancement of heat transfer by clamped flags in a Poiseuille channel flow., JAE BOK LEE, SUNG GOON PARK, BOYOUNG KIM, HYUNG JIN SUNG, KAIST — A pair of flexible flags clamped vertically in a heated channel was numerically modeled to study an enhancement of heat transfer by the clamped flags in a Poiseuille channel flow. The penalty immersed boundary method was adopted to analyze the fluid–structure–thermal interaction between the surrounding fluid and the clamped flags. The dynamics of the clamped flags was categorized into three distinctive modes: a flapping mode, a fully deflected mode, and an irregular mode. The distinctive modes that depended on the relationship between the hydrodynamic force and the restoring force displayed different movement patterns. The flapping mode provided superior thermal performance to the other modes. Vortices generated from the flapping flags swept out the thermal boundary layer and entrained the fluid near channel walls into the channel core flow while passing through the wake periodically. Compared to rigid flags, the flapping flags significantly improved the thermal efficiency. In addition, the effects of channel height and Reynolds number on the thermal efficiency were explored to obtain an optimal parameter set, which presented the highest thermal performance in present study. The flexible flags regarding the optimal parameter set showed an increase of up to 230% in net heat flux, compared to the baseline flow. Dynamic modes decomposition (DMD) method was adopted to examine the correlation between the vorticity and temperature fields.

4:56PM L3.00003 Rotation-triggered path instabilities of rising spheres and cylinder

, VARGHESE MATHAI, XIAOJUE ZHU, Physics of Fluids Group, University of Twente, The Netherlands, CHAO SUN, Center for Combustion Energy & Department of Thermal Engineering, Tsinghua University, DETLEF LOHSE, Physics of Fluids Group, University of Twente, The Netherlands — Path-instabilities are a common observation in the dynamics of buoyant particles in flows. However, the factors leading to the onset of oscillatory motion have remained difficult to predict even for simple bodies such as bubbles, spheres and cylinders. In literature, two quantities are considered to control the buoyancy-driven dynamics for isotropic bodies (spheres and cylinders); they are the particle's density relative to the fluid ($\Gamma \equiv \rho_p / \rho_f$) and its Galileo number (Ga). In contrast to this picture, we show that buoyant spheres (as well as cylinders) can exhibit dramatically different modes of vibration and wake-shedding patterns under seemingly identical conditions (Γ and Ga fixed). These effects stem from the simplest of changes in the mass distribution of the particle (hollow to solid sphere), which changes its rotational inertia. We show that rotation can couple with the particle's translational motion and trigger distinctly different wake-induced oscillatory motions. The present findings also provide an explanation for the wide variation that is witnessed in the dynamics of buoyant isotropic bodies.

5:09PM L3.00004 Particle motion in a periodic driving flow. The role of added mass force and the finite size of particles.¹

, GERARDO RUIZ CHAVARRIA, ERICK JAVIER LOPEZ SANCHEZ, Facultad de Ciencias, Universidad Nacional Autonoma de Mexico — The motion of particles in a fluid is an open problem. The main difficulty arises from the fact that hydrodynamical forces acting on a particle depend on the flow properties. In addition, the form and the size of particles must be taken into account. In this work we present numerical results of the particle transport in a periodic driving flow in a channel flushing into an open domain. To study the transport of particles we solve the equation of motion for a spherical particle in which we include the drag, the gravity, the buoyancy, the added mass and the history force. Additionally we include the corrections for a particle of finite size. For solving this equation a knowledge of the velocity field is required. To obtain the velocity field we solve the Navier Stokes and the continuity equations with a finite volume method. In the flow under study a vorticity dipole and a spanwise vortex are present, both have an important influence on the motion of particles. The dipole enhances displacement of particles because flow between vortices behaves like a jet and the spanwise vortex produces the lifting and deposition of particles from/to the bottom. We observe clustering of particles both into the channel and in the open domain as observed in coastal systems.

¹The authors acknowledge DGAPA-UNAM by support under project PAPIIT IN115315 Ondas y estructuras coherentes en dinmica de fluidos.

5:22PM L3.00005 Stability and dynamics of electron plasma vortex under external strain

, N. C. HURST, J. R. DANIELSON, D. H. E. DUBIN, C. M. SURKO, University of California - San Diego — The behavior of two-dimensional vortex structures is of key interest in a number of important physical systems, including geophysical fluids¹ and strongly magnetized plasmas.² Studied here is the case of an initially axisymmetric vortex subjected to a simple strain flow. Experiments are performed using pure electron plasmas confined in a Penning-Malmberg trap to model the dynamics of an ideal two-dimensional fluid.³ Vortex-In-Cell simulations are also conducted to complement the laboratory results. The dynamical behavior and stability threshold of the strained vortex are measured, showing good agreement with Kida's elliptical patch model for relatively flat vorticity profiles⁴. However, non-flat profiles feature a reduced stability threshold, apparently due to filamentation at the vortex periphery.

¹D. G. Dritschel and B. Legras, *Phys. Today* **46**, 44 (1993).

²P. W. Terry, *Rev. Mod. Phys.* **72**, 1 (2000).

³C. F. Driscoll *et. al.*, *Physica C* **369**, 21 (2002)

⁴S. Kida, *J. Phys. Soc. Japan* **50**, 3517 (1981).

5:35PM L3.00006 Rotating Wheel Wake

, JEAN-ELOI LOMBARD, HUI XU, DAVE MOXEY, SPENCER SHERWIN, Imperial College — For open wheel race-cars, such as Formula One, or IndyCar, the wheels are responsible for 40% of the total drag. For road cars, drag associated to the wheels and under-carriage can represent 20 – 60% of total drag at highway cruise speeds. Experimental observations have reported two, three or more pairs of counter rotating vortices, the relative strength of which still remains an open question. The near wake of an unsteady rotating wheel. The numerical investigation by means of direct numerical simulation at $Re_D=400\text{--}1000$ is presented here to further the understanding of bifurcations the flow undergoes as the Reynolds number is increased. Direct numerical simulation is performed using Nektar++, the results of which are compared to those of Pirozzoli *et al.*(2012). Both proper orthogonal decomposition and dynamic mode decomposition, as well as spectral analysis are leveraged to gain unprecedented insight into the bifurcations and subsequent topological differences of the wake as the Reynolds number is increased.

5:48PM L3.00007 Physics of rowing

, JEAN-PHILIPPE BOUCHER, ROMAIN LABBE, TIMOTHEE MOUTERDE, CHRISTOPHE CLANET, LadHyx - Ecole Polytechnique — Synchronization in rowing seems like a crucial condition for those who aim at winning top-level rowing races. However, in nature, one can observe animals with many legs, such as krill, swimming in a desynchronized manner which is nearly metachronal. From a physicist point of view, rowing by following a metachronal wave also seems like a great idea because, at high Reynolds number, the metachronal gait has one big advantage over the synchronized gait: it reduces the fluctuations of speed and thus the drag on the body. In this experimental study, we have built a scale model of a rowing boat to deal with the question of the effect of synchronization on the boat performance.

Monday, November 21, 2016 4:30PM - 6:27PM –

Session L4 Industrial Application: Power Generation B112 - Jennifer Franck, Brown University

4:30PM L4.00001 Simulations of Oscillating Hydrofoils in Array Configurations¹,

JENNIFER FRANCK, FILIP SIMESKI, ARIANNE SPAULDING, Brown University — The vortex and wake interactions of multiple oscillating foils are investigated computationally for energy harvesting applications. Oscillating with high pitch and heave amplitudes to maximize power production, the elliptical-shaped foils generate large coherent vortices at the leading and trailing edge, which are shed downstream to create a large highly structured wake of vortices with alternating sign. Downstream foils oscillate within the large organized wake at a relative phase angle to the lead foil such that power efficiency is optimized. When placed directly downstream of one another, the optimal phase of a second foil is to avoid interactions with the first foil's wake, generating less than half of the total power of the first foil. However, when placed in a staggered configuration the downstream foil has an increase in efficiency through constructive vortex-foil interactions.

¹Funded by ARPAE

4:43PM L4.00002 Effect of Elevated Free Stream Turbulence on the Hydrodynamic Performance of a Tidal Turbine Blade Section , ASHWIN VINOD, ANGELA LAWRENCE, ARINDAM BANERJEE,

Lehigh University — The effects of elevated freestream turbulence (FST) on the performance of a tidal turbine blade is studied using laboratory experiments. Of interest for the current investigation is elevated levels of FST in the range of 6-24% that is prevalent in deployment sites of tidal turbines. A constant chord, no twist blade section (SG6043) is tested at an operating Reynolds number of 1.5×10^5 and at angles of attack ranging from -90° to $+90^\circ$. The parameter space encompasses the entire operational range of a tidal turbine that includes flow reversal. Multiple levels of controlled FST are achieved using an active grid type turbulence generator placed at the entrance to the water tunnel test section. The hydrodynamic loads experienced by the blade section are measured using a 3-axis load cell; a Stereo-PIV technique is used to analyze the flow field around the blade. The results indicate that elevated levels of FST cause a delay in flow separation when compared to the case of a laminar freestream. Furthermore, the lift to drag ratio of the blade is considerably altered depending on the level of FST and angle of attack tested.

4:56PM L4.00003 The effect of blade pitch in the rotor hydrodynamics of a cross-flow turbine.¹ , MIGUEL SOMOANO, FRANCISCO HUERA-HUARTE, Universitat Rovira i Virgili — In this work we will show how

the hydrodynamics of the rotor of a straight-bladed Cross-Flow Turbine (CFT) are affected by the Tip Speed Ratio (TSR), and the blade pitch angle imposed to the rotor. The CFT model used in experiments consists of a three-bladed (NACA-0015) vertical axis turbine with a chord (c) to rotor diameter (D) ratio of 0.16. Planar Digital Particle Image Velocimetry (DPIV) was used, with the laser sheet aiming at the mid-span of the blades, illuminating the inner part of the rotor and the near wake of the turbine. Tests were made by forcing the rotation of the turbine with a DC motor, which provided precise control of the TSR, while being towed in a still-water tank at a constant Reynolds number of 61000. A range of TSRs from 0.7 to 2.3 were covered for different blade pitches, ranging from 8° toe-in to 16° toe-out. The interaction between the blades in the rotor will be discussed by examining dimensionless phase-averaged vorticity fields in the inner part of the rotor and mean velocity fields in the near wake of the turbine.

¹Supported by the Spanish Ministry of Economy and Competitiveness, grant BES-2013-065366 and project DPI2015-71645-P.

5:09PM L4.00004 Shape Optimization of A Turbine-99 Draft Tube Using Design-by-Morphing , SAHUCK OH, CHUNG-HSIANG JIANG, PHILIP MARCUS, University of California, Berkeley, DAVID GUTZWILLER, ALAIN DEMEULENAERE, NUMECA USA, Inc., CHIYU JIANG, University of California, Berkeley — We have found the optimal shape of a turbine-99

draft tube that maximizes its pressure recovery factor using a new design method called design-by- morphing. In design-by- morphing, new draft tubes are created by morphing multiple baseline draft tubes with different weights. The surfaces of baseline draft tubes are approximated by a summation of spectral coefficients multiplied by spectral basis functions. Then, a morphed draft tube is produced by computing a new set of spectral coefficients which are a weighted average of the spectral coefficients of the baseline draft tubes. The optimal draft tube is obtained by finding the weights such that the mean pressure recovery factor is maximized. After optimization is carried out using design-by- morphing, the high static pressure region is significantly reduced, and the flow is smoother and more uniform than it was in any of the baseline turbine-99 draft tubes. The optimal draft tube shows a 10.9% improvement over the turbine-99 draft tube. We have applied this method to trains and to aircrafts, and have reduced the drag and the drag-to-lift ratio by 13.2% and 23.1%, respectively. We believe that this optimization method is applicable to many engineering applications in which the performance of an object depends on its shape.

5:22PM L4.00005 Effects of free surface on flow energy harvesting system based on flapping foils¹ , LUBAO TENG, JIAN DENG, XUEMING SHAO, Department of Mechanics, Zhejiang University, Hangzhou 310027, People's Republic of China — Here, we consider a flapping foil based energy harvester, which is modelled by a 2D NACA0015 foil performing coupled motions of pitching and heaving. Volume of fraction(VOF) method is employed to capture the free surface. We fix the Reynolds number at $Re = 900$, and the Froude number at $Fr = 0.32$. We fix the non-dimensional flapping frequency at $f = 0.16$, the pitching amplitude at $\theta_0 = 75^\circ$, and the heaving amplitude at $h_0 = 1c$, where c is the chord length. With these parameters, the harvester has been proved to reach its highest efficiency of $\eta = 0.34$ in a single phase flow. By varying the submergence d , which is defined as the distance between the calm free surface and the highest position of the pitching pivot of the flapping foil, we find that the free surface affects pronouncedly the energy harvesting efficiency η . As d decreases from $24c$ to $0.5c$, η increases from 0.34 to 0.41, getting a 20% promotion of the efficiency. To reveal the underlying physical mechanism of the effects of free surface, we examine the time histories of hydrodynamic forces on the foil. We find that due to the existence of the the free surface, the lift force and pitching moment experience asymmetric time histories during the upstroke and downstroke of the foil.

¹This research is supported by the National Natural Science Foundation of China (Grant No: 11272283) and the Public Projects of Zhejiang Province (Grant No: 2015C31126) to conduct this research.

5:35PM L4.00006 Vortex wake interactions and energy harvesting from tandem pitching and heaving hydrofoils¹, YUNXING SU, JENNIFER CARDONA, MICHAEL MILLER, SHREYAS MANDRE, KENNETH BREUER, Brown University — Measurements of flow structure and power extraction by tandem pitching and heaving hydrofoils are conducted in a flume. The leading and trailing hydrofoils are synchronized and aligned parallel to the oncoming flow. Force measurements and time-resolved PIV are used to characterize the system. The system efficiency of tandem foils with the same kinematics is quantified as a function of the phase difference between the foils and there exist favorable and unfavorable phase angles and that system efficiencies can be as large as 0.45. For unfavorable phase angles, PIV indicates that the leading edge vortex generated by the trailing foil, which is critical to good energy harvesting, is weakened by the oncoming wake from the leading foil. Conversely, at a favorable phase, the vortex shed from the leading foil enhances the performance of the trailing foil, compensating for the otherwise negative aspects of operating in the wake. A model, combining frequency, separation distance and a characteristic convection velocity, is introduced to predict the optimal phase region and is validated over a range of parameters. By changing the pitching amplitude and phase angle in trailing foil we show that relatively larger pitching amplitudes can further improve the system efficiency.

¹ARPA-e

5:48PM L4.00007 The Effects of Inducer Inlet Diffusion on Backflow, TATE FANNING, Brigham Young University, RYAN LUNDGREEN, The Ohio State University, STEVEN GORRELL, DANIEL MAYNES, Brigham Young University, 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 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 the backflow of inducer geometries with a tip clearance of 0.4 mm to allow tip leakage flow and a tip clearance of 0 mm to remove tip leakage flow at varying flow coefficients under both single phase and cavitating conditions. Despite removal of the tip leakage flow, backflow persists, and upstream propagation 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 for single phase conditions. 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.

6:01PM L4.00008 Comparison of atomization characteristics of drop-in and conventional jet fuels¹, KUMARAN KANNAIYAN, REZA SADR, Texas A and M University at Qatar, MICRO SCALE THERMO-FLUIDS LAB TEAM — Surge in energy demand and stringent emission norms have been driving the interest on alternative drop-in fuels in aviation industry. The gas-to-liquid (GTL), synthetic paraffinic kerosene fuel derived from natural gas, has drawn significant attention as drop-in fuel due to its cleaner combustion characteristics when compared to other alternative fuels derived from various feedstocks. The fuel specifications such as chemical and physical properties of drop-in fuels are different from those of the conventional jet fuels, which can affect their atomization characteristics and in turn the combustion performance. The near nozzle liquid sheet dynamics of the drop-in fuel, GTL, is studied at different nozzle operating conditions and compared with that of the conventional Jet A-1 fuel. The statistical analysis of the near nozzle sheet dynamics shows that the drop-in fuel atomization characteristics are comparable to those of the conventional fuel. Furthermore, the microscopic spray characteristics measured using phase Doppler anemometry at downstream locations are slightly different between the fuels.

¹Authors acknowledge the support by National Priorities Research Program (NPRP) of Qatar National Research Fund through the grant NPRP-7-1449-2-523

6:14PM L4.00009 Design of an anemometer to characterize the flow in the ducts of a hydrogenerator rotor rim¹, KEVIN VENNE, LAURENT MYDLARSKI, McGill University, FEDERICO TORRIANO, JEAN-PHILIPPE CHAREST-FOURNIER, CLAUDE HUDON, JEAN-FRANCOIS MORISSETTE, IREQ (Hydro-Quebec) — Due to its complex geometry, the airflow within hydrogenerators is difficult to characterize. And although CFD can be a reliable engineering tool, its application to the field of hydrogenerators is very recent and has certain inherent limitations, which are due in part to geometrical and flow complexities, including the coexistence of moving (rotor) and stationary (stator) components. For this reason, experimental measurements are required to validate the CFD simulations of such complex flows. To this end, a 1:4 scale model of a hydrogenerator was constructed at the IREQ (Hydro-Québec Research Institute) to better understand the flow dynamics in the rotor and stator components, and to help benchmark its CFD simulations. However, new flow sensors must be developed to quantify the flow in the confined and harsh regions of hydrogenerators. Of particular interest is the flow within the rotor rim ducts, since it is directly responsible for cooling one of the most critical components, the poles. This rather complex task required the design of an anemometer that had to be accurate, durable, cost-effective, easy to install, and able to withstand the extreme conditions (temperatures of 50C, centrifugal forces of 300g, etc.) found in hydrogenerators. This paper presents two preliminary designs of such sensors and a series of tests that were performed to calibrate and test them.

¹Funding graciously provided by the NSERC and FRQNT

Monday, November 21, 2016 4:30PM - 6:14PM – Session L5 Compressible Flow: Shock Waves B113 - Michael Hargather, New Mexico Tech

4:30PM L5.00001 Optical diagnostics of turbulent mixing in explosively-driven shock tube¹, JAMES ANDERSON, MICHAEL HARGATHER, New Mexico Tech — Explosively-driven shock tube experiments were performed to investigate the turbulent mixing of explosive product gases and ambient air. A small detonator initiated Al / I₂O₅ thermite, which produced a shock wave and expanding product gases. Schlieren and imaging spectroscopy were applied simultaneously along a common optical path to identify correlations between turbulent structures and spatially-resolved absorbance. The schlieren imaging identifies flow features including shock waves and turbulent structures while the imaging spectroscopy identifies regions of iodine gas presence in the product gases. Pressure transducers located before and after the optical diagnostic section measure time-resolved pressure. Shock speed is measured from tracking the leading edge of the shockwave in the schlieren images and from the pressure transducers. The turbulent mixing characteristics were determined using digital image processing. Results show changes in shock speed, product gas propagation, and species concentrations for varied explosive charge mass.

¹Funded by DTRA Grant HDTRA1-14-1-0070

4:43PM L5.00002 An experimental study of shock wave propagation through a polyester film¹, VERONICA ELIASSEN, University of Southern California and University of California, San Diego, HONGJOO JEON, University of Southern California — A polyester film is available in a variety of uses such as packaging, protective overlay, barrier protection, and other industrial applications. In the current study, shock tube experiments are performed to study the influence of a polyester film on the propagation of a planar shock wave. A conventional shock tube is used to create incident shock Mach numbers of $M_s = 1.34$ and 1.46 . A test section of the shock tube is designed to hold a 0.009 mm, 0.127 mm, 0.254 mm, or 0.508 mm thick polyester film (Dura-Lar). High-temporal resolution schlieren photography is used to visualize the shock wave mitigation caused by the polyester film. In addition, four pressure transducers are used to measure the elapsed time of arrival and overpressure of the shock wave both upstream and downstream of the test section. Results show that the transmitted shock wave in the polyester film is clearly observed and the transmitted shock Mach number is decreased by increasing film thickness.

¹This study is supported by the National Science Foundation under grant No. CBET-1437412.

4:56PM L5.00003 Direct Numerical Simulation of a normal shock train with thermal nonequilibrium, ROMAIN FIVET¹, VENKAT RAMAN², University of Michigan — The role of a normal shock train in a supersonic engine is to convert a sufficient amount of the incoming kinetic energy into internal energy by the entrance of the combustor, in order to guarantee flame ignition. It comprises a succession of compression and expansion waves attached to a turbulent boundary layer. When the molecular collisional process is not fast enough compared to convective and turbulent timescales, thermal nonequilibrium becomes important, which could alter the energy conversion process. By changing the local thermophysical properties and density, nonequilibrium can change the shock structures leading to changes in the energy conversion process. Here, direct numerical simulations are used to study the effect of such nonequilibrium on a Mach 2.0 rectangular isolator. A one-dimensional time-averaged analysis is used to quantify this effect on the pressure work and turbulent kinetic energy evolution.

¹PhD Candidate

²Associate Professor

5:09PM L5.00004 Numerical Study of Shock Wave Attenuation Using Logarithmic Spiral Liquid Sheet., QIAN WAN, University of Southern California, RALF DETERDING, University of Southampton, VERONICA ELIASSEN, University of California, San Diego — Research of shock wave attenuation has drawn much attention due to its military and civilian applications. One method to attenuate shock waves is to use water to block the shock wave propagation path and allow the shock wave to lose energy by breaking up the water sheet. We propose a way by holding a water sheet in logarithmic spiral shape, which has the ability of focusing the incident shock wave to its focal region. In addition, the shock wave will break up the bulk water and thus lose energy. The process of shock wave reflecting off and transmitting through the water sheet is numerically modeled using Euler equations and stiffened gas equation of state. In this study, the shock focusing ability of a logarithmic spiral water sheet is compared for various logarithmic spiral sheets. Further, the attenuation effect is quantified by the measurement of pressure impulse and peak pressure behind the transmitted and reflected shock waves.

5:22PM L5.00005 Material Point Methods for Shock Waves¹, DUAN ZHANG, TILAK DHAKAL, Los Alamos National Laboratory — Particle methods are often the choice for problems involving large material deformation with history dependent material models. Often large deformation of a material is caused by shock loading, therefore accurate calculation of shock waves is important for particle methods. In this work, we study four major versions (original MPM, GIMP, CPDI, and DDMP) of material point methods, using a weak one-dimensional isothermal shock of ideal gas as an example. The original MPM fails. With a small number of particles, the GIMP and the CPDI methods produce reasonable results. However, as the number of particles increases these methods do not converge and produce pressure spikes. With sparse particles, DDMP results are unsatisfactory. As the number of particles increases, DDMP results converge to correct solutions, but the large number of particles needed for an accurate result makes the method very expensive to use in shock wave problems. To improve the numerical accuracy while preserving the convergence, conservation, and smoothness of the DDMP method, a new numerical integration scheme is introduced. The improved DDMP method is only slightly more expensive than the original DDMP method, but accuracy improvements are significant as shown by numerical examples.

¹This work was performed under the auspices of the United States Department of Energy

5:35PM L5.00006 The role of statistical fluctuations on the stability of shockwaves through gases with activated inelastic collisions¹, NICK SIRMAS, MATEI RADULESCU, University of Ottawa — The present study addresses the stability of piston driven shock waves through a system of hard particles subject to activated inelastic collisions. Molecular Dynamics (MD) simulations have previously revealed an unstable structure for such a system in the form of high density non-uniformities and convective rolls within the shock structure. The work has now been extended to the continuum level by considering the Euler and Navier-Stokes equations for granular gases with a modified cooling rate to include an impact threshold necessary for inelastic collisions. We find that the pattern formations produced in MD can be reproduced at the continuum level by continually perturbing the incoming density field. By varying the perturbation amplitude and wavelength, we find that fluctuations consistent with the statistical fluctuations seen in MD yield similar instabilities to those previously observed. While the inviscid model predicts a highly chaotic structure from these perturbations, the inclusion of viscosity and heat conductivity yields equivalent wavelengths of pattern formations to those seen in MD, which is equal to the relaxation length scale of the dissipative shock structure.

¹The authors acknowledged funding through the Alexander Graham Bell Canada Graduate Scholarship (NSERC) and Ontario Graduate Scholarship

5:48PM L5.00007 Regularized Moment Equations and Shock Waves for Rarefied Granular Gas, LAKSHMINARAYANA REDDY, MEHEBOOB ALAM, Jawaharlal Nehru Centre for Advanced Scientific Research, Bangalore, India — It is well-known that the shock structures predicted by extended hydrodynamic models are more accurate than the standard Navier-Stokes model in the rarefied regime, but they fail to predict continuous shock structures when the Mach number exceeds a critical value. Regularization or parabolization is one method to obtain smooth shock profiles at all Mach numbers. Following a Chapman-Enskog-like method [H. Struchtrup, 2004, Phys. Fluids], we have derived the “regularized” version 10-moment equations (“R10” moment equations) for inelastic hard-spheres. In order to show the advantage of R10 moment equations over standard 10-moment equations, the R10 moment equations have been employed to solve the Riemann problem of plane shock waves for both molecular and granular gases. The numerical results are compared between the 10-moment and R10-moment models and it is found that the 10-moment model fails to produce continuous shock structures beyond an upstream Mach number of 1.34, while the R10-moment model predicts smooth shock profiles beyond the upstream Mach number of 1.34. The density and granular temperature profiles are found to be asymmetric, with their maxima occurring within the shock-layer [Reddy & Alam, 2015, J Fluid Mech, vol. 779, R2].

6:01PM L5.00008 Existence of solutions to the Guderley implosion problem in arbitrary media, ZACHARY BOYD, SCOTT RAMSEY, ROY BATY, Los Alamos National Laboratory — It is known classically that in an ideal gas, there exist self-similar, spherical, converging shock solutions, but much less is understood about the existence of such solutions in compressible flow of real materials. On the other hand, it has recently been pointed out that there exist self-similar solutions for the Euler equations regardless of the equation of state closure model, which suggests the possibility that the Guderley problem might be solvable in general. In this work, we rigorously determine what properties are required of an equation of state in order for an exact, self-similar Guderley flow to be realized, including a generic solution procedure in the cases where existence holds. Among other contexts, this result is of great practical interest for the verification of codes intended to treat shock propagation in a wide variety of real materials.

Monday, November 21, 2016 4:30PM - 6:40PM –

Session L6 Aerodynamics: General Application B114 - Claire Verhulst, United States Military Academy

4:30PM L6.00001 Numerical analysis of the Magnus moment on a spin-stabilized projectile¹, MICHAEL CREMINS, United States Military Academy, GREGORY RODEBAUGH, Picatinny Arsenal, CLAIRE VERHULST, MICHAEL BENSON, BRET VAN POPPEL, United States Military Academy — The Magnus moment is a result of an uneven pressure distribution that occurs when an object rotates in a crossflow. Unlike the Magnus force, which is often small for spin-stabilized projectiles, the Magnus moment can have a strong detrimental effect on flight stability. According to one source, most transonic and subsonic flight instabilities are caused by the Magnus moment [Modern Exterior Ballistics, McCoy], and yet simulations often fail to accurately predict the Magnus moment in the subsonic regime. In this study, we present hybrid Reynolds Averaged Navier Stokes (RANS) and Large Eddy Simulation (LES) predictions of the Magnus moment for a spin-stabilized projectile. Velocity, pressure, and Magnus moment predictions are presented for multiple Reynolds numbers and spin rates. We also consider the effect of a sting mount, which is commonly used when conducting flow measurements in a wind tunnel or water channel. Finally, we present the initial designs for a novel Magnetic Resonance Velocimetry (MRV) experiment to measure three-dimensional flow around a spinning projectile.

¹This work was supported by the Department of Defense High Performance Computing Modernization Program (DoD HPCMP)

4:43PM L6.00002 Characterizing the flow field around ballutes of various geometries¹, JEFFREY PANKO, MARIA-ISABEL CARNASCIALI, University of New Haven — A ballute combines the performance of large parachutes with the rigidity and design flexibility of aeroshells. Such designs, when optimized, could drastically increase the allowable payload for interplanetary missions associated with high reentry velocities, for which, the current capabilities of thermal protection systems are being reached. Using commercially available software, a CFD investigation into the flow phenomena and performance characteristics of various such designs was conducted in order to determine features which may prove conducive for use in aerocapture missions, a primary application of such technology. Concerns around current ballute designs stem from the aerodynamic heating loads and flow instabilities at reentry velocities and as such, the study revolved around geometries which would provide favorable performance under such environments. Design parameters included: blunt versus sharp bodies, boundary layer control, and turbulence model. Results were monitored for changes in lift to drag ratios (L/D), separation point, vortex shedding, and control authority.

¹Funding for this work, in part, provided by the CT Space Grant Consortium.

4:56PM L6.00003 Aerodynamic Design of a Locomotive Fairing, CHAD STUCKI, DANIEL MAYNES, Brigham Young Univ - Provo — Rising fuel cost has motivated increased fuel efficiency of freight trains. At cruising speed, the largest contributing factor to the fuel consumption is the aerodynamic drag. As a result of air stagnation at the front of the train and substantial flow separation behind, the leading locomotive and trailing railcar experience greater drag than intermediate cars. This work introduces the design of streamlined nose fairings to be attached to freight locomotives as a means of reducing the leading locomotive drag. The aerodynamic performance of each fairing design is modeled using a commercial CFD software package. The K-epsilon turbulence model is used, and fluid properties are equivalent to atmospheric air at standard conditions. A selection of isolated screening studies are performed, and a multidimensional regression is used to predict optimal-performing fairing designs. Between screening studies, careful examination of the flow field is performed to inspire subsequent fairing designs. Results are presented for 250 different nose fairings. The best performing fairing geometry predicts a nominal drag reduction of 17% on the lead locomotive in a train set. This drag reduction is expected to result in nearly 1% fuel savings for the entire train.

5:09PM L6.00004 ABSTRACT WITHDRAWN –

5:22PM L6.00005 Spectral Analysis of the Wake behind a Helicopter Rotor Hub, CHRISTOPHER PETRIN, Oklahoma State Univ, DAVID REICH, SVEN SCHMITZ, Pennsylvania State Univ, BRIAN ELBING, Oklahoma State Univ — 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. LDV and PIV measurements in the far-wake consistently showed a six-per-revolution flow structure, in addition to stronger two- and four-per-revolution structures. These six-per-revolution structures persisted into the far-field, and have no direct geometric counterpart on the hub model. The current study will examine the Reynolds number dependence of these structures and present higher-order statistics of the turbulence within the wake. In addition, current activity using the EFPL Large Water Tunnel at Oklahoma State University will be presented. This effort uses a more canonical configuration to identify the source for these six-per-revolution structures, which are assumed to be a non-linear interaction between the two- and four-per-revolution structures.

5:35PM L6.00006 Unsteady Sail Dynamics in Olympic Class Sailboats, CHARLES WILLIAMSON, RILEY SCHUTT, Cornell University — Unsteady sailing techniques have evolved in competitive sailboat fleets, in cases where the relative weight of the sailor is sufficient to impart unsteady motions to the boat and sails. We will discuss three types of motion that are used by athletes to propel their boats on an Olympic race course faster than using the wind alone. In all of our cases, body weight movements induce unsteady sail motion, increasing driving force and speed through the water. In this research, we explore the dynamics of an Olympic class Laser sailboat equipped with a GPS, IMU, wind sensor, and a 6-GoPro camera array. We shall briefly discuss "sail flicking", whereby the helmsman periodically rolls the sail into the apparent wind, at an angle which is distinct from classical heave (in our case, the oscillations are not normal to the apparent flow). We also demonstrate "roll tacking", where there are considerable advantages to rolling the boat during such a maneuver, especially in light wind. In both of the above examples from on-the-water studies, corresponding experiments using a towing tank exhibit increases in the driving force, associated with the formation of strong vortex pairs into the flow. Finally, we focus on a technique known as "S-curving" in the case where the boat sails downwind. In contrast to the previous cases, it is drag force rather than lift force that the sailor is trying to maximise as the boat follows a zig-zag trajectory. The augmented apparent wind strength due to the oscillatory sail motion, and the growth of strong synchronised low-pressure wake vortices on the low-pressure side of the sail, contribute to the increase in driving force, and velocity-made-good downwind.

5:48PM L6.00007 Topology of Vortex-Wing Interaction¹, CHRIS MCKENNA, DONALD ROCKWELL, Lehigh University — Aircraft flying together in an echelon or V formation experience aerodynamic advantages. Impingement of the tip vortex from the leader (upstream) wing on the follower wing can yield an increase of lift to drag ratio. This enhancement is known to depend on the location of vortex impingement on the follower wing. Particle image velocimetry is employed to determine streamline topology in successive crossflow planes, which characterize the streamwise evolution of the vortex structure along the chord of the follower wing and into its wake. Different modes of vortex-follower wing interaction are created by varying both the spanwise and vertical locations of the leader wing. These modes are defined by differences in the number and locations of critical points of the flow topology, and involve bifurcation, attenuation, and mutual induction. The bifurcation and attenuation modes decrease the strength of the tip vortex from the follower wing. In contrast, the mutual induction mode increases the strength of the follower tip vortex.

¹AFOSR

6:01PM L6.00008 Tip vortex core pressure estimates derived from velocity field measurements, KYLE SINDING, MICHAEL KRANE, Penn State University — We present estimates of tip vortex core pressure derived from velocity field measurements of a high Reynolds number flow over a lifting surface. Tip vortex cavitation decreases propulsor efficiency and contributes to both unwanted noise and surface damage. Coordinated load cell, pressure, and velocity measurements were performed in the 12-inch tunnel at the Applied Research Laboratory at Penn State University, over a range of angles of attack and flow speeds. Stereo PIV imaging planes were oriented normal to the tunnel axis. Pressure estimates in each measurement plane were estimated from the velocity field. Visual cavitation calls were performed over the same range of conditions as the optical velocity measurements, by varying the tunnel pressure until tip vortex cavitation was observed to initiate. The pressure differences between the tip vortex and the tunnel ambient pressure obtained with these two methods were then compared.

6:14PM L6.00009 Calculating forces on thin flat plates with incomplete vorticity-field data, ERIC LIMACHER, CHRIS MORTON, DAVID WOOD, University of Calgary — Optical experimental techniques such as particle image velocimetry (PIV) permit detailed quantification of velocities in the wakes of bluff bodies. Patterns in the wake development are significant to force generation, but it is not trivial to quantitatively relate changes in the wake to changes in measured forces. Key difficulties in this regard include: (i) accurate quantification of velocities close to the body, and (ii) the effect of missing velocity or vorticity data in regions where optical access is obscured. In the present work, we consider force formulations based on the vorticity field, wherein mathematical manipulation eliminates the need for accurate near-body velocity information. Attention is restricted to nominally two dimensional problems, namely (i) a linearly accelerating flat plate, investigated using PIV in a water tunnel, and (ii) a pitching plate in a freestream flow, as investigated numerically by Wang & Eldredge (2013). The effect of missing vorticity data on the pressure side of the plate has a significant impact on the calculation of force for the pitching plate test case. Fortunately, if the vorticity on the pressure side remains confined to a thin boundary layer, simple corrections can be applied to recover a force estimate.

6:27PM L6.00010 Large Angle Unsteady Aerodynamic Theory of a Flat Plate, FIELD MANAR, ANYA JONES, Univ. of Maryland - College Park — A purely analytical approach is taken for the evaluation of the unsteady loads on a flat plate. This allows for an extremely low cost theoretical prediction of the plate loads in the style of Wagner and Theodorsen, without making the assumption of small angle of attack or small disturbance flow. The forces and moments are evaluated using the time rate of change of fluid momentum, expressed as an integral of the vorticity field. The flow is taken as inviscid and incompressible with isolated vorticity bound to the plate and in the shed wake. The bound vorticity distribution on the plate is solved exactly using conformal mapping of the plate to a cylinder. In keeping with the original assumption of Wagner, the wake vorticity is assumed to remain stationary in an inertial reference frame and convection is disregarded. Formulation in this manner allows for a closed form solution of Wagner's problem valid at all angles of attack. Separation from the leading edge of the plate can also be included to further increase the fidelity of the model at high angles.

Monday, November 21, 2016 4:30PM - 6:40PM –
Session L7 Flow Control: Feedback, System and Model Identification B115 - Jeff Borggaard, Virginia Tech

4:30PM L7.00001 The Stability Region for Feedback Control of the Wake Behind Twin Oscillating Cylinders¹, JEFF BORGGAARD, SERKAN GUGERCIN, LIZETTE ZIETSMAN, Virginia Tech — Linear feedback control has the ability to stabilize vortex shedding behind twin cylinders where cylinder rotation is the actuation mechanism. Complete elimination of the wake is only possible for certain Reynolds numbers and cylinder spacing. This is related to the presence of asymmetric unstable modes in the linearized system. We investigate this region of parameter space using a number of closed-loop simulations that bound this region. We then consider the practical issue of designing feedback controls based on limited state measurements by building a nonlinear compensator using linear robust control theory with and incorporating the nonlinear terms in the compensator (e.g., using the extended Kalman filter). Interpolatory model reduction methods are applied to the large discretized, linearized Navier-Stokes system and used for computing the control laws and compensators. Preliminary closed-loop simulations of a three-dimensional version of this problem will also be presented.

¹Supported in part by the National Science Foundation

4:43PM L7.00002 Modelling and Feedback Control of Bistability in a Turbulent Bluff Body Wake, ROWAN BRACKSTON, ANDREW WYNN, JUAN MARCOS GARCIA DE LA CRUZ, Imperial College London, GEORGIOS RIGAS, California Institute of Technology, JONATHAN MORRISON, Imperial College London — The turbulent wake behind many three-dimensional bluff bodies exhibits a bistable behaviour, the properties of which has been the subject of significant recent interest. This feature of the wake is known to contribute to the pressure drag on the body and is relevant for geometries representative of many road vehicles. Furthermore, due to its high visibility from surface mounted pressure measurements, it is a feature that may be observed and controlled in real-time. In Brackston et al (*J. Fluid Mech.*, 2016) we have recently demonstrated such a feedback control strategy that aims to suppress the bistable feature of the wake. Starting from a stochastic modelling approach, 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 the feedback controller, with the aim of restoring the flow to the unstable, symmetric state. The controller is implemented experimentally at $Re \sim 2.3 \times 10^5$ 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 wake at turbulent Reynolds numbers.

4:56PM L7.00003 ABSTRACT WITHDRAWN —

5:09PM L7.00004 Towards DMD-Based Estimation and Control of Flow Separation using an Array of Surface Pressure Sensors¹, ERIC DEEM, LOUIS CATTAFESTA, Florida State University, HAO ZHANG, CLANCY ROWLEY, Princeton University — Closed-loop control of flow separation requires the spatio-temporal states of the flow to be fed back through the controller in real time. Previously, static and dynamic estimation methods have been employed that provide reduced-order model estimates of the POD-coefficients of the flow velocity using surface pressure measurements. However, this requires a learning dataset a priori. This approach is effective as long as the dynamics during control do not stray from the learning dataset. Since only a few dynamical features are required for feedback control of flow separation, many of the details provided by full-field snapshots are superfluous. This motivates a state-observation technique that extracts key dynamical features directly from surface pressure, without requiring PIV snapshots. The results of identifying DMD modes of separated flow through an array of surface pressure sensors in real-time are presented. This is accomplished by employing streaming DMD on the fly to surface pressure snapshots. These modal characteristics exhibit striking similarities to those extracted from PIV data and the pressure field obtained via solving Poissons equation. Progress towards closed-loop separation control based on the dynamic modes of surface pressure will be discussed.

¹Supported by AFOSR grant FA9550-14-1-0289

5:22PM L7.00005 Active flow control insight gained from a modified integral boundary layer equation, AVRAHAM SEIFERT, Tel Aviv Univ — Active Flow Control (AFC) can alter the development of boundary layers with applications (e.g., reducing drag by separation delay or separating the boundary layers and enhancing vortex shedding to increase drag). Historically, significant effects of steady AFC methods were observed. Unsteady actuation is significantly more efficient than steady. Full-scale AFC tests were conducted with varying levels of success. While clearly relevant to industry, AFC implementation relies on expert knowledge with proven intuition and or costly and lengthy computational efforts. This situation hinders the use of AFC while simple, quick and reliable design method is absent. An updated form of the unsteady integral boundary layer (UIBL) equations, that include AFC terms (unsteady wall transpiration and body forces) can be used to assist in AFC analysis and design. With these equations and given a family of suitable velocity profiles, the momentum thickness can be calculated and matched with an outer, potential flow solution in 2D and 3D manner to create an AFC design tool, parallel to proven tools for airfoil design. Limiting cases of the UIBL equation can be used to analyze candidate AFC concepts in terms of their capability to modify the boundary layers development and system performance.

5:35PM L7.00006 Localized modelling and feedback control of linear instabilities in 2-D wall bounded shear flows, HENRY TOL, MARIOS KOTSONIS, COEN DE VISSER, Delft Univ of Tech — A new approach is presented for control of instabilities in 2-D wall bounded shear flows described by the linearized Navier-Stokes equations (LNSE). The control design accounts both for spatially localized actuators/sensors and the dominant perturbation dynamics in an optimal control framework. An inflow disturbance model is proposed for streamwise instabilities that drive laminar-turbulent transition. The perturbation modes that contribute to the transition process can be selected and are included in the control design. A reduced order model is derived from the LNSE that captures the input-output behavior and the dominant perturbation dynamics. This model is used to design an optimal controller for suppressing the instability growth. A 2-D channel flow and a 2-D boundary layer flow over a flat plate are considered as application cases. Disturbances are generated upstream of the control domain and the resulting flow perturbations are estimated/controlled using wall shear measurements and localized unsteady blowing and suction at the wall. It will be shown that the controller is able to cancel the perturbations and is robust to unmodelled disturbances.

5:48PM L7.00007 Sensitivity analysis of unstable periodic orbits in a weakly chaotic Kuramoto-Sivashinsky system, DAVIDE LASAGNA, University of Southampton — Unstable periodic orbits (UPOs) often explain to a remarkable degree of accuracy global statistical features of the turbulent flow in which they are found. In other words, orbital averages, even for short-period UPOs, are good approximation of long-time averages computed over a chaotic, turbulent realisation. Here, we re-examine this property from a design perspective: Does the same degree of approximation exists between the *sensitivity* of orbital averages with respect to design parameters and the sensitivity of the long-time average itself? Knowledge of this quantity is key in many fundamental design problems involving turbulent flows, most notably in control. In this work, we present an efficient, well conditioned adjoint algorithm derived from specialising well-known variational techniques to the inherent temporal periodicity of UPOs. Once an UPO is available, this algorithm computes the sensitivity of orbital averages with respect to many design parameters at once, regardless of the orbital stability properties. As a demonstration, we analyse the sensitivity to in-domain linear feedback of UPOs found for the Kuramoto-Sivashinsky system in a weakly chaotic regime.

6:01PM L7.00008 Adjoint-optimization algorithm for spatial reconstruction of a scalar source¹, QI WANG, Johns Hopkins University, YOSUKE HASEGAWA, The University of Tokyo, CHARLES MENEVEAU, TAMER ZAKI, Johns Hopkins University — Identifying the location of the source of passive scalar transported in a turbulent environment based on remote measurements is an ill-posed problem. A conjugate-gradient algorithm is proposed, and relies on eddy-resolving simulations of both the forward and adjoint scalar transport equations to reconstruct the spatial distribution of the source. The formulation can naturally accommodate measurements from multiple sensors. The algorithm is evaluated for scalar dispersion in turbulent channel flow ($Re_T = 180$). As the distance between the source and sensor increases, the accuracy of the source recovery deteriorates due to diffusive effects. Improvement in performance is demonstrated for higher Prantl numbers and also with increasing number of sensors.

¹This study is supported by the National Science Foundation (grant CNS 1461870)

6:14PM L7.00009 What is the 'correct' formulation of the linearised Navier-Stokes equations for designing feedback flow control systems?, OLIVER DELLAR, BRYN JONES, The University of Sheffield, ACSE COLLABORATION — The use of feedback control is looking increasingly attractive as a means of reducing the pressure drag which acts upon bluff body vehicles such as heavy goods vehicles, and thus reducing both fuel consumption and CO₂ emissions. Motivated by the need to efficiently obtain low-order models of such flows in order to utilise model based control theory, we consider the effect on system dynamics of basing the plant model on different formulations of the linearised Navier-Stokes equations. The dynamics of a single computational node's subsystem which arises upon spatial discretisation of the governing equations in both primitive variables and pressure Poisson equation formulations are considered, revealing fundamental differences at the nodal level. The effects of these differences on system dynamics at the full fluid flow system level are exemplified by considering the corresponding formulations of a two-dimensional channel flow, subjected to a number different of boundary conditions. This ultimately reveals which formulations of the governing equations are suitable for feedback control design, and which should be avoided.

6:27PM L7.00010 Delaunay-based derivative-free optimization for efficient minimization of time-averaged statistics of turbulent flows., POORIYA BEYHAGHI, University of California, San Diego

— This work considers the problem of the efficient minimization of the infinite time average of a stationary ergodic process in the space of a handful of independent parameters which affect it. Problems of this class, derived from physical or numerical experiments which are sometimes expensive to perform, are ubiquitous in turbulence research. In such problems, any given function evaluation, determined with finite sampling, is associated with a quantifiable amount of uncertainty, which may be reduced via additional sampling. This work proposes the first algorithm of this type. Our algorithm remarkably reduces the overall cost of the optimization process for problems of this class. Further, under certain well-defined conditions, rigorous proof of convergence is established to the global minimum of the problem considered.

Monday, November 21, 2016 4:30PM - 6:27PM –

Session L8 Nonlinear Dynamics: Model Reduction B116 - Maciej Balajewicz, University of Illinois at Urbana-Champaign

4:30PM L8.00001 Lagrangian dimensionality reduction of convection dominated nonlinear flows, MACIEJ BALAJEWICZ, RAMBOD MOJGANI, University of Illinois at Urbana-Champaign — We introduce a new projection-based model reduction approach for convection dominated nonlinear fluid flows. In this method the evolution of the fluid is approximated in the Lagrangian frame of reference. More specifically, global basis functions are utilized for both the state of the system and the positions of the Lagrangian computational domain. In this approach, wave-like solutions exhibit low-rank structure and thus, can be approximated efficiently using a small number of reduced bases. The proposed approach is successfully demonstrated for the reduction of several simple but representative flow problems.

4:43PM L8.00002 Novel Stochastic Mode Reduction For General Irreversible Systems¹, MARKUS SCHMUCK, School of Mathematical and Computer Sciences and Maxwell Institute for Mathematical Sciences, Heriot-Watt University, Edinburgh EH144AS, MARC PRADAS, Department of Mathematics and Statistics, The Open University, Milton Keynes MK7 6AA, GRIGORIOS A. PAVLIOTIS, Department of Mathematics, Imperial College London, London SW7 2AZ, UK, SERAFIM KALLIADASIS, Department of Chemical Engineering, Imperial College London, London SW7 2AZ, UK — We outline a novel stochastic mode reduction strategy for nonlinear irreversible dynamical systems. Our methodology is based on the concept of maximum information entropy together with spectral characteristics of linear operators and a dynamic renormalization strategy [1,2]. It results in low-dimensional stochastic equations equipped with a systematically determined noise term. We demonstrate the performance and validity of our novel method with various physical model prototypes such as front propagation in reaction diffusion systems, phase separation in binary mixtures, and coarsening of interfaces. These are just a few examples demonstrating the wide applicability of our computational mode reduction.

1. M. Schmuck, M. Pradas, S. Kalliadasis & G.A. Pavliotis, Phys. Rev. Lett. 110:244101 2013.
2. M. Schmuck, M. Pradas, G.A. Pavliotis & S. Kalliadasis, IMA J.Appl. Math. 80:273-301 2015.

¹ERC Advanced Grant No. 247031 and EPSRC Grant No. EP/H034587

4:56PM L8.00003 Computing Finite-Time Lyapunov Exponents with Optimally Time Dependent Reduction¹, HESSAM BABAE, MOHAMMAD FARAZMAND, THEMIS SAPSIS, MIT, GEORGE HALLER, ETH — We present a method to compute Finite-Time Lyapunov Exponents (FTLE) of a dynamical system using Optimally Time-Dependent (OTD) reduction recently introduced by H. Babaee and T.P. Sapsis (*A minimization principle for the description of modes associated with finite-time instabilities*, Proceedings of the Royal Society of London A: Mathematical, Physical and Engineering Sciences, Vol. 472, 2016). The OTD modes are a set of finite-dimensional, time-dependent, orthonormal basis $\{\mathbf{u}_i(\mathbf{x}, t)\}_{i=1}^N$ that capture the directions associated with transient instabilities. The evolution equation of the OTD modes is derived from a minimization principle that optimally approximates the most unstable directions over finite times. To compute the FTLE, we evolve a single OTD mode along with the nonlinear dynamics. We approximate the FTLE from the reduced system obtained from projecting the instantaneous linearized dynamics onto the OTD mode. This results in a significant reduction in the computational cost compared to conventional methods for computing FTLE. We demonstrate the efficiency of our method for double Gyre and ABC flows.

¹ARO project 66710-EG-YIP

5:09PM L8.00004 Evaluating the accuracy of the dynamic mode decomposition¹, HAO ZHANG, SCOTT DAWSON, CLARENCE ROWLEY, Princeton University, ERIC DEEM, LOUIS CATTAFESTA, Florida State University — Dynamic mode decomposition (DMD) is a practical way to extract dynamic information about a fluid flow directly from data. As a data-driven method, DMD can suffer from error, which can be difficult to quantify without knowledge of an exact solution, free from noise or external disturbances. Here we propose an evaluation metric for the accuracy of DMD results (eigenvalues, modes, and eigenfunctions), by exploiting a connection between DMD and the Koopman operator, a linear operator acting on functions of the flow state. In particular, a DMD mode is considered "accurate" if the corresponding eigenfunction closely approximates a Koopman eigenfunction. With this definition, we can assess the accuracy of any individual DMD mode directly from data, without requiring the direct calculation of the Koopman operator. We demonstrate the use of this criterion with a range of examples including synthetic, numerical, and experimental data.

¹Supported by AFOSR grant FA9550-14-1-0289

5:22PM L8.00005 Sparse Identification of Nonlinear Dynamics (SINDy) , STEVEN BRUNTON

University of Washington, JOSHUA PROCTOR, Institute for Disease Modeling, NATHAN KUTZ, University of Washington — This work develops a general new framework to discover the governing equations underlying a dynamical system simply from data measurements, leveraging advances in sparsity techniques and machine learning. The so-called sparse identification of nonlinear dynamics (SINDy) method results in models that are parsimonious, balancing model complexity with descriptive ability while avoiding over fitting. The only assumption about the structure of the model is that there are only a few important terms that govern the dynamics, so that the equations are sparse in the space of possible functions; this assumption holds for many physical systems in an appropriate basis. We demonstrate the algorithm on a wide range of problems, from simple canonical systems, including the chaotic Lorenz system, to the canonical fluid vortex shedding behind an circular cylinder at $Re=100$. We also show that this method generalizes to parameterized systems and systems that are time-varying or have external forcing. With abundant data and elusive laws, data-driven discovery of dynamics will continue to play an increasingly important role in the characterization and control of fluid dynamics.

5:35PM L8.00006 A Low-Order Galerkin Model Based on DMD and Adjoint-DMD modes¹ , WEI ZHANG, MINGJUN WEI, Kansas State University

Dynamic Mode Decomposition (DMD) has emerged as a new tool for the understanding of flow dynamics associated with frequencies. The DMD modes computed by this process have been considered as an alternative of base functions for model order reduction. However, DMD modes are not orthogonal bases which are usually desired for the simplicity of Galerkin models. Therefore, we used the bi-orthogonal pair of DMD modes and adjoint DMD modes to solve this problem, and introduced an easy approach to derive a simple DMD-Galerkin projection model. The introduction of adjoint DMD modes also provides an easy way to rank DMD modes for order reduction. The approach is applied on a flow-passing-cylinder case in both transition and periodic stages. For the periodic case, DMD-Galerkin model is similar to POD-Galerkin model; and for the transition case, DMD-Galerkin model carries more clear frequency features.

¹Supported by ARL

5:48PM L8.00007 Model Order Reduction for Fluid Dynamics with Moving Solid Boundary¹ , HAOTIAN GAO, MINGJUN WEI, Kansas State University

We extended the application of POD-Galerkin projection for model order reduction from usual fixed-domain problems to more general fluid-solid systems when moving boundary/interface is involved. The idea is similar to numerical simulation approaches using embedded forcing terms to represent boundary motion and domain change. However, such a modified approach will not get away with the unsteadiness of boundary terms which appear as time-dependent coefficients in the new Galerkin model. These coefficients need to be pre-computed for prescribed motion, or worse, to be computed at each time step for non-prescribed motion. The extra computational cost gets expensive in some cases and eventually undermines the value of using reduced-order models. One solution is to decompose the moving boundary/domain to orthogonal modes and derive another low-order model with fixed coefficients for boundary motion. Further study shows that the most expensive integrations resulted from the unsteady motion (in both original and domain-decomposition approaches) have almost negligible impact on the overall dynamics. Dropping these expensive terms reduces the computation cost by at least one order while no obvious effect on model accuracy is noticed.

¹Supported by ARL

6:01PM L8.00008 Sensor Placement in Multiscale Phenomena using Multi-Resolution Dynamic Mode Decomposition , KRITHIKA MANOHAR, EURIKA KAISER, STEVEN L. BRUNTON, J. NATHAN KUTZ, University of Washington

Multiscale processes pose challenges in determining modal decompositions with physical meaning which can render sensor placement particularly difficult. Localized features in space or time may play a crucial role for the phenomenon of interest but may be insufficiently resolved due to their low energy contribution. We consider optimal sensor placement using spatial interpolation points within the framework of data-driven modal decompositions. In recent years, the Discrete Empirical Interpolation Method or DEIM and variants like QDEIM have gained popularity for interpolating nonlinear terms arising in model reduction using Proper Orthogonal Decomposition modes. We extend this sensor placement approach to multiscale physics problems using Multi-Resolution Dynamic Mode Decomposition or mrDMD (Kutz et al., 2015), an unsupervised multi-resolution analysis in the time-frequency domain that separates flow features occurring at different timescales. The discovered sensors achieve accurate flow state reconstruction in representative multiscale examples including global ocean temperature data with an energetic El Niño mode. Interestingly, this method places sensors near coastlines without imposing additional constraints, which is beneficial from an engineering perspective.

6:14PM L8.00009 Compressed sensing DMD with control , ZHE BAI, EURIKA KAISER, University of Washington, JOSHUA PROCTOR, Institute of Disease Modeling, J. NATHAN KUTZ, STEVEN BRUNTON, University of Washington — The dynamic mode decomposition (DMD) has been widely adopted in the fluid dynamics community, in part due to its ease of implementation, its connection to nonlinear dynamical systems, and its highly extensible formulation as a linear regression. This work combines the recent innovations of compressed sensing DMD and DMD with control, resulting in a new computational framework to extract spatiotemporal coherent structures using subsampled data from a complex system with inputs or control. The resulting compressed DMD with control (cDMDc), has two major uses in high-dimensional systems, such as a fluid flow: 1) if only subsampled or compressive measurements are available, it is possible to use compressed sensing to reconstruct full-dimensional DMD modes, and 2) if full data is available, it is possible to accelerate computations by first pre-compressing data and then reconstructing full modes from compressed DMD computations. In both cases, the addition of DMD with inputs and control makes it possible to disambiguate the natural unforced dynamics from the effect of actuation. We demonstrate this architecture on a number of relevant examples from fluid dynamics.

Monday, November 21, 2016 4:30PM - 6:40PM –
Session L9 General Fluid Dynamics: Drag Reduction B117 - Stefano Leonardi, University of Texas, Dallas

4:30PM L9.00001 Why fibers are better turbulent drag reducing agents than polymers.¹, ARNOUT BOELENS², University of Chicago, MURUGAPPAN MUTHUKUMAR, University of Massachusetts, Amherst

— It is typically found in literature that fibers are not as effective as drag reducing agents as polymers. However, for low concentrations, when adding charged polymers to either distilled or salt water, it is found that polymers showing rod-like behavior are better drag reducing agents than polymers showing coil-like behavior [1]. In this study [2], using hybrid Direct Numerical Simulation with Langevin dynamics, a comparison is performed between polymer and fiber stress tensors in turbulent flow. The stress tensors are found to be similar, suggesting a common drag reducing mechanism in the onset regime. Since fibers do not have an elastic backbone, this must be a viscous effect. Analysis of the viscosity tensor reveals that all terms are negligible, except the off-diagonal shear viscosity associated with rotation. Based on this analysis, we are able to explain why charged polymers showing rod-like behavior are better drag reducing agents than polymers showing coil-like behavior. Additionally, we identify the rotational orientation time as the unifying time scale setting a new time criterion for drag reduction by both flexible polymers and rigid fibers. References: [1] P.S. Virk (1975), Nature, 253, 109-110 [2] A.M.P. Boelens, M. Muthukumar (2016), PRE, 93, 052503

¹This research was supported by NSF Grant No. DMR-1404940 and AFOSR Grant No. FA9550-14-1-0164.

²Research performed while at University of Massachusetts, Amherst.

4:43PM L9.00002 Bubble drag reduction requires large bubbles¹, RUBEN VERSCHOOF, ROELAND VAN DER VEEN, CHAO SUN, DETLEF LOHSE, Physics of Fluids, MESA+ institute, University of Twente, the Netherlands — In the maritime industry, the injection of air bubbles into the turbulent boundary layer under the ship hull is seen as one of the most promising techniques to reduce the overall fuel consumption. A few volume percent ($\leq 4\%$) of bubbles can reduce the overall drag up to 40% and beyond. However, the exact mechanism is unknown, thus hindering further progress and optimization. Here we show that bubble drag reduction in turbulent flow *dramatically* depends on the bubble size. By adding minute concentrations (6 ppm) of the surfactant Triton X-100 into otherwise completely unchanged strongly turbulent Taylor-Couette flow containing bubbles, we dramatically reduce the drag reduction from more than 40% to about 4%, corresponding to the trivial effect of the bubbles on the density and viscosity of the liquid. The reason for this striking behavior is that the addition of surfactants prevents bubble coalescence, leading to much smaller bubbles. Our result demonstrates that bubble deformability is crucial for bubble drag reduction in turbulent flow.

¹We acknowledge support from STW and FOM

4:56PM L9.00003 Rigid spherical particles in highly turbulent Taylor-Couette flow¹, DENNIS BAKHUIS, RUBEN A. VERSCHOOF, VARGHESE MATHAI, SANDER G. HUISMAN, DETLEF LOHSE, Physics of Fluids Group, Mesa+ Institute and J.M. Burgers Centre for Fluid Dynamics, University of Twente, The Netherlands, CHAO SUN, Center for combustion energy, Tsinghua University, Beijing, China — Many industrial and maritime processes are subject to enormous frictional losses. Reducing these losses even slightly will already lead to large financial and environmental benefits. The understanding of the underlying physical mechanism of frictional drag reduction is still limited, for example, in bubbly drag reduction there is an ongoing debate whether deformability and bubble size are the key parameters. In this experimental study we report high precision torque measurements using rigid non-deformable spherical particles in highly turbulent Taylor-Couette flow with Reynolds numbers up to 2×10^6 . The particles are made of polystyrene with an average density of 1.036 g cm^{-3} and three different diameters: 8mm, 4mm, and 1.5mm. Particle volume fractions of up to 6% were used. By varying the particle diameter, density ratio of the particles and the working fluid, and volume fraction of the particles, the effect on the torque is compared to the single phase case. These systematic measurements show that adding rigid spherical particles only results in very minor drag reduction.

¹This work is financially supported by Netherlands Organisation for Scientific Research (NWO) by VIDI grant number 13477.

5:09PM L9.00004 Liquid Infused Surfaces in Turbulent Channel Flow¹, MATTHEW FU, YING LIU, HOWARD STONE, MARCUS HULTMARK, Princeton University — Liquid infused surfaces have been proposed as a robust method for turbulent drag reduction. These surfaces consist of functionalized roughness elements wetted with a liquid lubricant that is immiscible with external fluids. The presence of the lubricant creates mobile, fluid-fluid interfaces, each of which can support a localized slip. Collectively, these interfaces yield a finite slip velocity at the effective surface, which has been demonstrated to reduce skin friction drag in turbulent flows. Retention of the lubricant layer is critical to maintaining the drag reduction effect. A turbulent channel-flow facility is used to characterize the drag reduction and robustness of various liquid infused surfaces. Micro-manufactured surfaces are mounted flush in the channel and exposed to turbulent flows. The retention of fluorescent lubricants and pressure drop are monitored to characterize the effects of surface geometry and lubricant properties.

¹ Supported under ONR Grants N00014-12-1-0875 and N00014-12-1-0962 (program manager Ki-Han Kim) and by the Department of Defense (DoD) through the National Defense Science & Engineering Graduate Fellowship (NDSEG) Program

5:22PM L9.00005 Slip length of liquid-infused surfaces in high aspect-ratio microchannels¹, ARUNRAJ BALAJI, MATTHEW FU, MARCUS HULTMARK, Princeton University — Liquid-infused surfaces (LIS) derive their drag-reduction effects from the presence of flow inside lubricant-filled surface cavities or grooves. This behavior has been characterized by an effective slip length, which is known to be the primary parameter in determining drag-reduction. Though slip length has been theoretically parametrized as a function of LIS geometry, fluid properties, and channel dimensions, previous studies were performed without consideration of all three variables simultaneously. Specifically, existing models do not address the regime in which channel height is on the order of LIS-feature length scale. High aspect-ratio microchannels with rectangular-groove LIS along one wall are constructed and tested. Pressure measurements are used to determine effective slip length for various surface geometries, channel heights, and viscosity ratios. Results are compared with theoretical expectations.

¹Supported under ONR Grants N00014-12-1-0875 and N00014-12-1-0962 (program manager Ki-Han Kim)

5:35PM L9.00006 Optimizing Geometry Mediated Skin Friction Drag on Riblet-Textured Surfaces, SHABNAM RAAYAI, GARETH MCKINLEY, Massachusetts Inst of Tech-MIT — Micro-scale riblets have been shown to modify the skin friction drag on patterned surfaces. Shark skin is widely known as a natural example of this passive drag reduction mechanism and artificial riblet tapes have been previously used in the America's Cups tournament resulting in a 1987 victory. Previous experiments with riblet surfaces in turbulent boundary layer flow have shown 4-8% reduction in the skin friction drag. Our computations with sinusoidal riblet surfaces in high Reynolds number laminar boundary layer flow and experiments with V-grooves in laminar Taylor-Couette flow also show that the reduction in skin friction can be substantial and depends on the spacing and height of the riblets. In the boundary layer setting, this frictional reduction is also a function of the length of the plate in the flow direction, while in the Taylor Couette setting it depends on the gap size. In the current work, we use scaling arguments and conformal mapping to establish a simplified theory for laminar flow over V-groove riblets and explore the self-similarity of the velocity contours near the patterned surface. We combine these arguments with theoretical and numerical calculations using Matlab and OpenFOAM to show that the drag reduction achievable in laminar flow over riblet surfaces depends on a rescaled form of the Reynolds number combined with the aspect ratio of the texture (defined in terms of the ratio of the height to spacing of the riblets). We then use these results to explain the underlying physical mechanisms driving frictional drag reduction and offer recommendations for designing low drag surfaces.

5:48PM L9.00007 Development of reduced drag concepts for acoustic liners using experimental methods, CHRISTOPHER JASINSKI, Univ of Notre Dame, THOMAS CORKE, Notre Dame — Commercial aircraft have used acoustic liners to reduce engine noise for many years, although their drag production has been largely unstudied. The next generation of aircraft may benefit from additional surface area covered by acoustic liner, thus understanding their drag production mechanism is crucial for future designs. An accurate direct aerodynamic drag measurement technique has been developed using a force balance with linear air bearings. Using 3D-printed and conventional liners, low-drag designs are being developed. This paper will investigate the underlying fluid mechanics governing the drag production in acoustic liners and describe new attempts to reduce aerodynamic drag.

6:01PM L9.00008 ABSTRACT WITHDRAWN —

6:14PM L9.00009 Vapor layers reduce drag without the crisis, IVAN VAKARELSKI, King Abdullah University of Science and Technology, JOSEPH BERRY, DEREK CHAN, University of Melbourne, SIGURDUR THORODDSEN, King Abdullah University of Science and Technology — The drag of a solid sphere moving in fluid is known to be only a function of the Reynolds number, Re and diminishes rapidly at the drag crisis around $Re \sim 3 \cdot 10^5$. A Leidenfrost vapor layer on a hot sphere surface can trigger the onset of the drag crisis at lower Re . By using a range of high viscosity perfluorocarbon liquids, we show that the drag reduction effect, can occur over a wide range of Re , from as low as ~ 600 . The Navier slip model with a viscosity dependent slip length captures the observed drag reduction and wake shape.

6:27PM L9.00010 Octopus-inspired drag cancellation by added mass pumping, GABRIEL WEYMOUTH, FRANCESCO GIORGIO-SÉRCHE, University of Southampton — Recent work has shown that when an immersed body suddenly changes its size, such as a deflating octopus during rapid escape jetting, the body experiences large forces due to the variation of added-mass energy. We extend this line of research by investigating a spring-mass oscillator submerged in quiescent fluid subject to periodic changes in its volume. This system isolates the ability of the added-mass thrust to cancel the bluff body resistance (having no jet flow to confuse the analysis) and moves closer to studying how these effects would work in a sustained propulsion case by studying periodic shape-change instead of a one-shot escape maneuver. With a combination of analytical, numerical, and experimental results, we show that the recovery of added-mass kinetic energy can be used to completely cancel the drag of the fluid, driving the onset of sustained oscillations with amplitudes as large as four times the average body radius. Moreover, these results are fairly independent of the details of the shape-change kinematics as long as the Stokes number and shape-change number are large. In addition, the effective pumping frequency range based on parametric oscillator analysis is shown to predict large amplitude response region observed in the numerics and experiments.

Monday, November 21, 2016 4:30PM - 6:40PM —

Session L10 DFD GPC: Convection and Buoyancy Driven Flows: Theory II B118-119 - Charles Doering, University of Michigan

4:30PM L10.00001 New variational bounds on convective transport. I. Formulation and analysis¹, IAN TOBASCO, ANDRE N. SOUZA, CHARLES R. DOERING, University of Michigan — We study the maximal rate of scalar transport between parallel walls separated by distance h , by an incompressible fluid with scalar diffusion coefficient κ . Given velocity vector field \mathbf{u} with intensity measured by the Péclet number $Pe = h^2 \langle |\nabla \mathbf{u}|^2 \rangle^{1/2} / \kappa$ (where $\langle \cdot \rangle$ is space-time average) the challenge is to determine the largest enhancement of wall-to-wall scalar flux over purely diffusive transport, i.e., the Nusselt number Nu . Variational formulations of the problem are presented and it is determined that $Nu \leq c Pe^{2/3}$, where c is an absolute constant, as $Pe \rightarrow \infty$. Moreover, this scaling for optimal transport—possibly modulo logarithmic corrections—is asymptotically sharp: admissible steady flows with $Nu \geq c' Pe^{2/3} / [\log Pe]^2$ are constructed. The structure of (nearly) maximally transporting flow fields is discussed.

¹Supported in part by National Science Foundation Graduate Research Fellowship DGE-0813964, awards OISE-0967140, PHY-1205219, DMS-1311833, and DMS-1515161, and the John Simon Guggenheim Memorial Foundation.

4:43PM L10.00002 New variational bounds on convective transport. II. Computations and implications¹, ANDRE SOUZA, Georgia Tech, IAN TOBASCO, CHARLES R. DOERING, Univ of Michigan - Ann Arbor — We study the maximal rate of scalar transport between parallel walls separated by distance h , by an incompressible fluid with scalar diffusion coefficient κ . Given velocity vector field \mathbf{u} with intensity measured by the Péclet number $Pe = h^2 \langle |\nabla \mathbf{u}|^2 \rangle^{1/2} / \kappa$ (where $\langle \cdot \rangle$ is space-time average) the challenge is to determine the largest enhancement of wall-to-wall scalar flux over purely diffusive transport, i.e., the Nusselt number Nu . Variational formulations of the problem are studied numerically and optimizing flow fields are computed over a range of Pe . Implications of this optimal wall-to-wall transport problem for the classical problem of Rayleigh-Bénard convection are discussed: the maximal scaling $Nu \sim Pe^{2/3}$ corresponds, via the identity $Pe^2 = Ra(Nu - 1)$ where Ra is the usual Rayleigh number, to $Nu \sim Ra^{1/2}$ as $Ra \rightarrow \infty$.

¹Supported in part by National Science Foundation Graduate Research Fellowship DGE-0813964, awards OISE-0967140, PHY-1205219, DMS-1311833, and DMS-1515161, and the John Simon Guggenheim Memorial Foundation.

4:56PM L10.00003 Global heat transport scaling in plume-controlled regime in turbulent Rayleigh-Bénard convection¹, KAI LEONG CHONG, SHI-DI HUANG, KE-QING XIA, The Chinese University of Hong Kong — Previous study by Chong et al.² has introduced a normalized aspect-ratio Γ/Γ_{opt} ($\Gamma_{opt} = 29.37Ra^{-0.31}$) where the plume coverage at fixed Γ/Γ_{opt} is invariant with respect to Ra in the so-called plume-controlled regime in Rayleigh-Bénard convection. We have studied the global heat transport scaling (expressed as Nusselt number Nu) at fixed Γ/Γ_{opt} with the Rayleigh number Ra between 10^7 and 10^{10} at fixed Prandtl number $Pr = 4.38$ by direct numerical simulations. It is found that at $\Gamma/\Gamma_{opt} = 1$ where the thermal plume becomes highly coherent and system-sized, Nu exhibits the scaling $Nu - 1 \sim Ra^{0.327 \pm 0.001}$ over three decades of Ra . This scaling is different from that found at $\Gamma = 1$ for which $Nu - 1 \sim Ra^{0.308 \pm 0.001}$, and this difference in scaling can be shown evidently in the compensated plots. 1. This work was supported by RGC of HKSAR (No. CUHK404513), CUHK Direct Grant (No. 3132740) and through a HKPhD Fellowship. 2. Chong, K. L., Huang, S.-D., Kaczorowski, M. & Xia, K.-Q. 2015 Condensation of coherent structures in turbulent flows. Phys. Rev. Lett. 115, 264503.

5:09PM L10.00004 Universality of energy spectrum in turbulent Rayleigh-Bénard convection, KUNLUN BAI, JUDITH HOELLER, ERIC BROWN, Yale University — We present study of energy spectrum in turbulent Rayleigh-Bénard convection, in both cylindrical and cubic containers, tilting and non-tilting conditions, and with Rayleigh number ranging from 0.5×10^9 to 1×10^{10} . For these different conditions of geometry, tilt, and Rayleigh number, the temperature spectra measured on the system side walls are significantly different from each other. Even for the same condition, the spectrum varies depending on whether the sensors locate in the path of large-scale circulations. However, quite interestingly, once the signals of large-scale circulations are subtracted from the raw temperature, all spectra display a universal shape, regardless of system geometry, tilt, Rayleigh number, and location of sensors. It suggests that one could model the large-scale circulations and small-scale fluctuations separately in turbulent Rayleigh-Bénard convection.

5:22PM L10.00005 Study of global heat transport and plume morphology in severely-confined Rayleigh-Bénard convection¹, KE-QING XIA, KAI LEONG CHONG, The Chinese University of Hong Kong — We study systematically how severe geometrical confinement influences the global heat transport (expressed as Nusselt number Nu) and the plume morphology in Rayleigh-Bénard convection (RBC) by means of direct numerical simulations. Broad ranges of width-to-height aspect-ratio ($1/128 \leq \Gamma \leq 1$) and Rayleigh number ($3 \times 10^4 \leq Ra \leq 10^{11}$) at fixed Prandtl number $Pr = 4.38$ are considered in present study. It is found that Nu exhibits the scaling $Nu - 1 \sim Ra^{0.61}$ over three decades of Ra at $\Gamma = 1/128$ and the flow is dominated by finger-like, long-lived plume columns for such severely-confined situation. The Nu scaling and the flow structures contrast sharply to that found at $\Gamma = 1$ for which Nu exhibits the scaling $Nu - 1 \sim Ra^{0.31}$ and the flow is dominated by mushroom-like, fragmented thermal plumes. Analogy is made between the severely-confined RBC and strongly rotating RBC. 1. This work was supported by RGC of HKSAR (No. CUHK404513), CUHK Direct Grant (No. 3132740) and through a HKPhD Fellowship.

5:35PM L10.00006 Superstructures in Rayleigh-Bénard convection¹, RICHARD STEVENS, University of Twente, ROBERTO VERZICCO, University of Rome 'Tor Vergata', DETLEF LOHSE, University of Twente — We study the heat transfer and the flow structures in Rayleigh-Bénard convection as function of the Rayleigh number Ra and the aspect ratio. We consider three-dimensional direct numerical simulations (DNS) in a laterally periodic geometry with aspect ratios up to $\Gamma = L_x/L_z = L_y/L_z = 64$ at $Ra = 10^8$, where L_x and L_y indicate the horizontal domain sizes and L_z the height. We find that the heat transport converges relatively quickly with increasing aspect ratio. In contrast, we find that the large scale flow structures change significantly with increasing aspect ratio due to the formation of superstructures. For example, at $Ra = 10^8$ we find the formation of basically only one large scale circulation roll in boxes with an aspect ratio up to 8. For larger boxes we find the formation of multiple of these extremely large convection rolls. We illustrate this by movies of horizontal cross-section of the bulk and the boundary layer and analyze them by using spectra in the boundary layer and the bulk. In addition, we study the effect of the large scale flow structures on the mean and higher order temperature and velocity statistics in the boundary layer and the bulk by comparing the simulation results obtained in different aspect ratio boxes.

¹Foundation for fundamental Research on Matter (FOM), Netherlands Center for Multiscale Catalytic Energy Conversion (MCEC), SURFsara, Gauss Large Scale project

5:48PM L10.00007 ABSTRACT WITHDRAWN —

6:01PM L10.00008 Roughness as a Route to the Kraichnan Regime in Thermal Convection, SRIKANTH TOPPALADODDI, Yale University, SAURO SUCCI, Istituto per le Applicazioni del Calcolo "Mauro Picone" (C.N.R.), JOHN WETTLAUFER, Yale University, University of Oxford, NORDITA — We use highly resolved numerical simulations to study turbulent Rayleigh-Bénard convection in a cell with sinusoidally rough upper and lower walls in two dimensions. By varying the wavelength at a fixed amplitude, we find an optimal wavelength for which the Nusselt-Rayleigh scaling relation is ($Nu - 1 \propto Ra^{0.482}$). This is consistent with (i) the upper bound of Goluskin and Doering (2016) who prove that Nu can grow no faster than $\mathcal{O}(Ra^{1/2})$ as $Ra \rightarrow \infty$, and thus (ii) the concept that roughness facilitates the attainment of the so-called ultimate regime of Kraichnan (1962). In the limits of very small and very large wavelengths we recover the planar case results, demonstrating how controlling the wall geometry manipulates the interaction between the boundary layers and the core flow.

6:14PM L10.00009 The parameter space of windy convection¹, DAVID GOLUSKIN, University of Michigan — In horizontally periodic Rayleigh-Bénard convection at large Rayleigh numbers (Ra), wavenumber-zero horizontal winds can arise spontaneously and dramatically alter the flow. The resulting "windy convection" has been observed in 2D domains and horizontally anisotropic 3D domains. As Ra is raised, the fraction of total kinetic energy contained in the wind approaches 100%. Vertical heat transport is greatly depressed by the wind and grows very slowly (if at all) as Ra is raised. Two different types of windy convection have been observed at different Prandtl numbers (Pr). At smaller Pr , heat is vertically convected almost exclusively during discrete bursts that are separated by long quiescent phases. At larger Pr , convective transport remains significant at all times. Convection can thus be identified as either windy or non-windy, and windy states can be either bursting or non-bursting. The regions of the Ra - Pr parameter plane in which each type of convection can occur remain poorly understood, as do transitions between these regions. This talk will summarize the phenomenon of windy convection in 2D and 3D and present a preliminary exploration of the Ra - Pr plane in the 2D case.

¹Partially supported by NSF award DMS-1515161

6:27PM L10.00010 Modeling of the thermal boundary layer in turbulent Rayleigh–Bénard convection.¹, MOHAMMAD EMRAN, OLGA SHISHKINA, Max Planck Institute for Dynamics and Self-Organization, Goettingen. — We report modeling of the thermal boundary layer in turbulent Rayleigh–Bénard convection (RBC), which incorporates the effect of turbulent fluctuations. The study is based on the thermal boundary layer equation from Shishkina et al., Phys. Rev. Lett. 114, 114302 (2015) and new Direct Numerical Simulations (DNS) of RBC in a cylindrical cell of the aspect ratio 1, for the Prandtl number variation of several orders of magnitude. Our modeled temperature profiles are found to agree with the DNS much better than those obtained with the classical Prandtl–Blasius or Falkner–Skan approaches.

¹The work is supported by the Deutsche Forschungsgemeinschaft (DFG) under the grant Sh405/4 – Heisenberg fellowship and SFB963, Project A06.

Monday, November 21, 2016 4:30PM - 6:14PM – Session L11 Jets: Turbulent and Variable Density C120-121-122 - Stephen Solovitz, Washington State University - Vancouver

4:30PM L11.00001 Modeling Variable-Density Jets with Co-Flow Using BHR, DANIEL ISRAEL, Los Alamos National Laboratory — The two-fluid jet in a co-flow has two similarity breaking features which make it more interesting, and challenging, than the simple self-similar jet. First, it transitions from strong jet to weak jet, and second, from shear driven to buoyancy driven. These two simultaneous mechanisms make it a strong test for a turbulence model. The Extreme Fluids team at Los Alamos National Laboratory has an on-going experimental campaign examining an SF_6 jet injected downwards into a co-flowing air stream. Using simultaneous PIV/PLIF they have obtained measurements of important turbulence quantities, including the Reynolds stresses, and the velocity-density correlations. In the current work, these measurements are used to validate the BHR turbulence model. The BHR model (Besnard et al., 1992) is a variable-density turbulence model similar to the LRR model for shear flows, but with additional transport equations for $\bar{\rho}a_i = \bar{\rho}'u_i''$ and $b = \bar{\rho}'v'$. Here we examine both the conventional model form, as well as a new version (Schwarzkopf et al., 2016) which include two length-scale equations: one for the dissipation scale, and one for the turbulent transport scale.

4:43PM L11.00002 Investigation of mean flow and turbulence for a variable-density jet near transition¹, STEPHEN SOLOVITZ, Washington State University Vancouver, LARRY MASTIN, USGS Cascades Volcano Observatory, BIANCA VIGLIANO, TAMARA DIB, NASIM ALI, RAUL CAL, Portland State University, VOLCANIC PLUME RESEARCH TEAM COLLABORATION — Plumes can vary widely in size and speed in geophysical systems, with Reynolds numbers (Re) extending from thousands to billions. Concurrently, their densities also have significant deviations, resulting in Richardson numbers (Ri) from negligible levels to near one. To investigate a range of these flow conditions more closely, a laboratory-scale experiment considered helium jets exhausting into air. The tests considered Re from 1500 to 10000 and Ri magnitudes near 0.001, which encompasses a series of jet conditions near the exit, including laminar, transitioning, and turbulent flow. Using particle image velocimetry (PIV), instantaneous velocity fields were acquired, and these were used to determine the mean velocity, entrainment, and turbulent statistics. The laminar jet showed very little development or entrainment, with only minor fluctuations. Turbulent jets had rapid flow development, nearing fully-developed conditions earlier than similar non-buoyant jets. For the transitioning jet, the entrainment and turbulent stresses were significantly larger than even the fully turbulent jet, with axial normal stresses more than doubled. Examining the instantaneous flow fields, these increases coincided with large, non-axisymmetric eddies in the shear layer.

¹Supported by NSF grant: EAR-1346580

4:56PM L11.00003 Low-order Description and Conditional Averaging in a Transitional Variable-Density Jet¹, BIANCA VIGLIANO, TAMARA DIB, NASEEM ALI, Portland State University, LARRY MASTIN, US Geological Survey, STEPHEN SOLOVITZ, Washington State University - Vancouver, RAUL BAYOAN CAL, Portland State University — A vertically oriented jet is investigated using particle image velocimetry to identify characteristics of the flow that contribute to the distribution of turbulent kinetic energy and development of Reynolds shear stresses. Experiments with three different gases air, argon and helium— as the jet and a range of exit velocities were performed to allow for variety of the Reynolds and Richardson numbers. Five cases are examined in total. Proper orthogonal decomposition is applied to assess the energy distribution of all cases with respect to the number of modes. Reconstruction of the Reynolds shear stresses using 50% of the total turbulent kinetic energy is performed. Quadrant analysis with respect to the Reynolds shear stress is also performed for insight into the entrainment of the jet. Reynolds shear stresses are dominant in Q1 - advancing ejections and Q3 - impeding entrainment and exhibit negligible contributions from the remaining two quadrants. POD is able to identify most dominant structures and transition effects within the basis. Via quadrant analysis, advancing ejection and impeding entrainment further describe the streamwise development of each jet.

¹National Science Foundation

5:09PM L11.00004 Eulerian and Lagrangian accelerations in the intermediate field of turbulent circular jets., JIN-TAE KIM, University of Illinois at Urbana-Champaign, ALEX LIBERZON, Tel Aviv University, LEONARDO P. CHAMORRO, University of Illinois at Urbana-Champaign — Particle tracking velocimetry is used to study the structure of various acceleration components, vorticity, and strain within the intermediate field of a circular jet at $Re = 6000$. The total acceleration is decomposed into three sets: a) streamwise-radial; b) tangential-normal; and c) local-convective components. Probability density function (PDF) and joint PDFs of each set are characterized at various radial locations within a streamwise band contained within 16 and 17 pipe diameters. Results show that the acceleration components are described by two distributions; one of them exhibits symmetry and heavy tails, while the other is best fitted by a power-law type. The PDF tails are heavier with increasing the radial distance. The growing departure from the Gaussian distribution is a result of the comparatively increase in turbulence promoted by the mean shear of the jet. The variation of third and fourth moments between the streamwise-tangential and the radial-normal accelerations indicates the anisotropy of the jet. Although joint PDFs show distinctive distribution and depend on the distance from the jet core, the relative angles between the Lagrangian acceleration with velocity, vorticity and strain show similar PDF across radial distances.

5:22PM L11.00005 On the outer flow field and ‘episodic’ entrainment in a round turbulent jet, SACHIN SHINDE, Indian Institute of Technology Kanpur, India, PRASANTH PRABHAKARAN, Max Planck Institute for Dynamics and Self-Organization, Goettingen, RODDAM NARASIMHA, Jawaharlal Nehru Center for Advanced Scientific Research, Bangalore, India — We study a round turbulent jet at a Reynolds number of 2400 using Direct Numerical Simulation (DNS), with focus on the flow field outside the turbulent core and on its relevance to the entrainment process. Using DNS flow imagery, we present a detailed analysis of this outer flow field, which is found to exhibit considerable order in sub-regions whose location near the jet boundary varies with time. This order is shown to be largely governed, at any given time, by the vorticity field associated with elements of the coherent structures within the turbulent core at the time. This is particularly clear in the simpler cases, where the coherent vorticity on the core side of the boundary of the jet is (e.g.) of only one sign, and the instantaneous outer flow velocities are inversely proportional to the radial distance from an effective vortex center as required by the Biot-Savart relation. Interestingly, the outer flow velocities can be as high as more than a third of the mean centerline velocity. Such high velocities are shown to appear as part of strong inrush events, and their intermittent occurrence in space and time favors an episodic view of the entrainment process.

5:35PM L11.00006 High-speed imaging of submerged jet: visualization analysis using proper orthogonality decomposition.¹, YINGZHENG LIU, CHUANGXIN HE, Shanghai Jiao Tong University — In the present study, the submerged jet at low Reynolds numbers was visualized using laser induced fluoresce and high-speed imaging in a water tank. Well-controlled calibration was made to determine linear dependency region of the fluoresce intensity on its concentration. Subsequently, the jet fluid issuing from a circular pipe was visualized using a high-speed camera. The animation sequence of the visualized jet flow field was supplied for the snapshot proper orthogonality decomposition (POD) analysis. Spatio-temporally varying structures superimposed in the unsteady fluid flow were identified, e.g., the axisymmetric mode and the helical mode, which were reflected from the dominant POD modes. The coefficients of the POD modes give strong indication of temporal and spectral features of the corresponding unsteady events. The reconstruction using the time-mean visualization and the selected POD modes was conducted to reveal the convective motion of the buried vortical structures.

¹National Natural Science Foundation of China

5:48PM L11.00007 Vortex structures in the near field of a transversely forced jet, OYVIND HANSEN-BAUER, DHIREN MISTRY, NICHOLAS WORTH, JAMES DAWSON, NTNU — We investigate the effect of transverse acoustic forcing on the formation of vortex structures in the near field of an axisymmetric jet using stereoscopic particle image velocimetry. The jet is placed at different locations between the pressure anti-node and node within a standing wave, and velocity and vorticity fields were measured in the $x - r$ plane. At the pressure anti-node, the jet response exhibited an axisymmetric mode, $m = 0$, as harmonic fluctuations in pressure and the streamwise velocity components result in the periodic formation of vortex rings at the forcing frequency. As the jet was moved away from the anti-node, the shear layer roll-up and resulting vortex structures become increasingly asymmetric and three-dimensional due to time-varying spatial pressure gradients across the jet exit. The location where the transverse and streamwise velocity fluctuations were of equal magnitude coincided with sudden change in the jet response, characterised by shear layer roll-up and resulting vortex structures either side of the jet being in anti-phase. At the pressure node, harmonic transverse oscillations of the jet were observed forming vortices of equal circulation on either side of the jet in anti-phase. Meandering of the potential core was also observed.

6:01PM L11.00008 Effect of large density ratios on turbulence budgets in buoyant jets with coflow, JOHN CHARONKO, KATHY PRESTRIDGE, Los Alamos National Laboratory — Turbulence statistics and energy transport budgets have been measured in two fully turbulent jets with coflow at density ratios of $s = 1.2 \& 4.2$ to improve our understanding of variable-density mixing in turbulent flows. The exit Reynolds number was matched for both flows at $\sim 20,000$ 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 over 10,000 snapshots of the flow at three locations to assure statistical convergence, and the spatial resolution ($288 \mu\text{m}$) is well below the Taylor microscale. Variable-density effects caused changes in both the magnitude and distribution of the evolving turbulence, with differences most pronounced within the jet half-width. As the jet tends toward pseudo self-similarity, a new scaling based on effective diameter and density successfully scales the energy budgets of the two jets, but significant differences were still seen in the core. For the high density ratio jet, the turbulent kinetic energy production is negative on the centerline, as opposed to slightly positive, leading to large changes in advection and diffusion. A mechanism for these differences is proposed.

Monday, November 21, 2016 4:30PM - 6:27PM –
Session L12 Multiphase Flows: Particle-Particle and Particle-Shock Interactions C123
- Heather Zunino, Arizona State University

4:30PM L12.00001 Taylor-Couette flows with radial fluid injection during co- and counter- rotation, for controlled mixing applications., NIKOLAS WILKINSON, CARI DUTCHER, University of Minnesota - Twin Cities — Flow between rotating concentric cylinders, called Taylor-Couette (TC) flow, offers high control over hydrodynamics, making TC flow ideal to study mixing. However, traditional TC cell design limits the ability to study the initial solution mixing dynamics while the cell is operating, due to geometric confinement and complexity when both cylinders are rotating. Here, we present a new TC cell design that allows for radial injection of fluids into the annulus while both cylinders are rotating. This Taylor-Couette cell has radius ratio of $Ri/Ro = 0.89$, an aspect ratio of $L/d = 60$ with 16 injection ports, and allows for both cylinders to rotate simultaneously and independently. With this geometry, we discuss how the injection port modification effects flow instabilities, as well as how radial injection during cylinder rotation modifies the flow. In our current work, we are studying flocculation of micron clay particles with polyelectrolyte solutions and how the hydrodynamics effects assembly and structure of these materials during the flocculation process.

4:43PM L12.00002 Flow induced streamer formation in particle laden complex flows

, NANDINI DEBNATH¹, MAHTAB HASSANPOURFARD², Univ of Alberta, RANAJAY GHOSH³, University of Central Florida, JAPAN TRIVEDI⁴, THOMAS THUNDAT⁵, ALOKE KUMAR⁶, Univ of Alberta — We study the combined flow of a polyacrylamide (PAM)solution with polystyrene (PS) nanoparticles, through a microfluidic device containing an array of micropillars. The flow is characterized by a very low Reynolds number ($Re < < 1$). We find that for exceeding a critical Weissenberg number ($Wi \geq 20$), PS nanoparticles localize near pillar walls to form thin slender string-like structures, which we call 'streamers' due to their morphology. Post-formation, these streamers show significant viscous behavior for short observational time-scales, and at longer observational time scales elastic response dominates. Our abiotic streamers could provide a framework for understanding similar structures that often form in biological systems.

¹PhD student, Department of Mechanical Engineering

²PhD student, Department of Chemical and Materials Engineering

³Assistant Professor, Department of Mechanical and Aerospace Engineering

⁴Assistant professor, Department of Civil and Environmental Engineering

⁵Professor, Department of Chemical and Materials Engineering

⁶Assistant professor, Department of Mechanical Engineering

4:56PM L12.00003 Shear flows of dense suspensions: flow modification by particle clustering and mixing

, BERNHARD VOWINCKEL, MEITAL CARMI, EDWARD BIEGERT, ECKART MEIBURG, Univ of California - Santa Barbara — We investigate numerically the behavior of sheared, dense suspensions of neutrally buoyant particles, for finite Reynolds number values. This type of problem is of particular interest for multiple applications in environmental, mechanical as well as process engineering such as debris flows, slurries, and pneumatic conveying in pipelines. Controlling channel flows laden with dense suspensions is very important as it can result in jamming of the channel, hence, lowering the efficiency of a hydraulic facility. It was observed that there exists a regime for which a small increase in shear force can cause a drastic, discontinuous increase of the effective viscosity of the mixture. This abrupt transition is commonly referred to as discontinuous shear thickening. We carry out phase-resolved numerical simulations to understand the modification of the flow on the grain scale in full detail allowing for improved definitions of threshold conditions. As the properties of the carrier fluid remain unchanged during the simulation, the thickening must be caused by the disperse phase, for example, by effects of changes in spatial particle distribution, clustering, and mixing. We provide a detailed statistical analysis to answer this question.

5:09PM L12.00004 Preferential Concentration of Inertial Sub-Kolmogorov Particles. The roles of mass loading of particles, St and Re_λ numbers

, SHOLPAN SUMBEKOVA, Univ Grenoble Alpes, LEGI, F-38000 Grenoble, France, ALBERTO ALISEDA, Department of Mechanical Engineering, University of Washington, Seattle WA 98195- 2600, USA, ALAIN CARTELLIER, CNRS, LEGI, F-38000 Grenoble, France, MICKAEL BOURGOIN, Univ Lyon, Ens de Lyon, Univ Claude Bernard, CNRS, Laboratoire de Physique, F-69342 Lyon, France — Turbulent flows laden with inertial particles present multiple open questions and are a subject of great interest in current research. Due to their higher density compared to the carrier fluid, inertial particles tend to form high concentration regions, i.e. clusters, and low concentration regions, i.e. voids, due to the interaction with the turbulence. In this work, we present an experimental investigation of the clustering phenomenon of heavy sub-Kolmogorov particles in homogeneous isotropic turbulent flows. Three control parameters have been varied over significant ranges: $Re_\lambda \in [170 - 450]$, $St \in [0.1 - 5]$ and volume fraction $\phi_v \in [2 \times 10^{-6} - 2 \times 10^{-5}]$. The scaling of clustering characteristics, such as the distribution of Voronoi areas and the dimensions of cluster and void regions, with the three parameters are discussed. In particular, for the polydispersed size distributions considered here, clustering is found to be enhanced strongly (quasi-linearly) by Re_λ and noticeably (with a square-root dependency) with ϕ_v , while characteristic cluster and void lengths are driven primarily by Re_λ . Weak dependence on St supports "sweep-stick" mechanism of clustering.

5:22PM L12.00005 ABSTRACT WITHDRAWN —

5:35PM L12.00006 Erosion of a grooved surface caused by impact of particle-laden flow

, SOHYUN JUNG, EUNJIN YANG, HO-YOUNG KIM, Seoul Natl Univ — Solid erosion can be a life-limiting process for mechanical elements in erosive environments, thus it is of practical importance in many industries such as construction, mining, and coal conversion. Erosion caused by particle-laden flow occurs through diverse mechanisms, such as cutting, plastic deformation, brittle fracture, fatigue and melting, depending on particle velocity, total particle mass and impingement angle. Among a variety of attempts to lessen erosion, here we investigate the effectiveness of millimeter-sized grooves on the surface. By experimentally measuring the erosion rates of smooth and triangular-grooved surfaces under various impingement angles, we find that erosion can be significantly reduced within a finite range of impingement angles. We show that such erosion resistance is attributed to the swirls of air within grooves and the differences in erosive strength of normal and slanted impact. In particular, erosion is mitigated when we increase the effective area under normal impact causing plastic deformation and fracture while decreasing the area under slanted impact that cuts the surface to a large degree. Our quantitative model for the erosion rate of grooved surfaces considering the foregoing effects agrees with the measurement results.

5:48PM L12.00007 Experimental measurement of unsteady drag on shock accelerated micro-particles

, ANKUR BORDOLOI, ADAM MARTINEZ, KATHERINE PRESTRIDGE, Physics Division, Los Alamos National Laboratory — The unsteady drag history of shock accelerated micro-particles in air is investigated in the Horizontal Shock Tube (HST) facility at Los Alamos National laboratory. Drag forces are estimated based on particle size, particle density, and instantaneous velocity and acceleration measured on hundreds of post-shock particle tracks. We use previously implemented 8-frame Particle Tracking Velocimetry/Anemometry (PTVA) diagnostics to analyze particles in high spatiotemporal resolution from individual particle trajectories. We use a simultaneous LED based shadowgraph to register shock location with respect to a moving particle in each frame. To measure particle size accurately, we implement a Phase Doppler Particle Analyzer (PDPA) in synchronization with the PTVA. In this presentation, we will corroborate with more accuracy our earlier observation that post-shock unsteady drag coefficients ($C_D(t)$) are manifold times higher than those predicted by theoretical models. Our results will also show that all $C_D(t)$ measurements collapse on a master-curve for a range of particle size, density, Mach number and Reynolds number when time is normalized by a shear velocity based time scale, $t^* = d/(u_f - u_p)$, where d is particle diameter, and u_f and u_p are post-shock fluid and particle velocities.

6:01PM L12.00008 Early motion in a rapidly decompressed particle bed, HEATHER ZUNINO, RONALD ADRIAN, AMANDA CLARKE, Arizona State University — Rapid expansion of dense, pressurized beds of fine particles subjected to rapid reduction of the external pressure is studied in a vertical shock tube. Located at bottom of a high pressure chamber below the shock tube diaphragm, a particle bed expands when the diaphragm bursts, releasing a near-sonic expansion wave that impinges on the particle bed-gas interface. The expansion wave presents a very rapid unloading to the particle bed. A high-speed video camera and pressure sensors capture events occurring during bed expansion. Interesting structures during the first few milliseconds include two-dimensional instabilities of the particle bed's surface and roughly spatially periodic regions void of particles within the bed. One-dimensional and two-dimensional Fourier analyses are used to measure their frequencies in space-time. It is found that the frequencies and patterns exhibit a clear dependence on particle diameter in which cell frequency decreases and cell size increases with increasing particle size. This work is 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 and Academic Alliance Program, under Contract No. DE-NA0002378.

6:14PM L12.00009 Eulerian-Lagrangian Simulation of an Explosive Dispersal of Particles¹, BERTRAND ROLLIN, Embry-Riddle Aeronautical University, FREDERICK OUELLET, RAHUL KONERU, SUBRAMANIAN ANNAMALAI, CCMT - University of Florida — Explosive dispersal of solid particles can be observed in a wide variety of contexts, notably in natural phenomenon such as volcanic eruptions or in engineering applications such as detonation of multiphase explosives. As the initial blast wave crosses the surrounding layer of particles, compaction occurs shortly before particles disperse radially outward at high speed. During the dispersion phase, complex multiphase interactions occurs between particles and detonation products of the explosive. Using a Eulerian-Lagrangian approach, namely point particle simulations, we study the case of a bed of particles of cylindrical shape surrounding an explosive chord. Our interest lies in predicting the behavior of particles after detonation. In particular, capturing and describing the mechanisms responsible for late-time formation of stable particle jets is sought. Therefore, detonation of the explosive material is not simulated. Instead an equivalent energy source is used to initiate the simulation. We present a detailed description of our approach to solving this problem, and our most recent progress in the analysis of particles explosive dispersal.

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

Monday, November 21, 2016 4:30PM - 6:40PM –
Session L13 DFD GPC: Geophysical Fluid Dynamics: Turbulence under Stable Stratification C124 - Luca Brandt, KTH Royal Institute of Technology

4:30PM L13.00001 ABSTRACT WITHDRAWN –

4:43PM L13.00002 On the Orientation of Turbulent Structures in Stably Stratified Shear Flows, FRANK JACOBITZ, ADAM MOREAU, JOYLENE AGUIRRE, University of San Diego — The orientation of turbulent structures in stably stratified shear flows are investigated using the results of a series of direct numerical simulations. The Richardson number is varied from $Ri = 0$, corresponding to unstratified shear flow, to $Ri = 1$, corresponding to strongly stratified shear flow. The evolution of the turbulent kinetic energy changes from growth for small Richardson numbers to decay for strong stratification. The orientation of turbulent structures in the flows is determined by the three-dimensional two-point autocorrelation coefficient of velocity magnitude, vorticity magnitude, and fluctuating density. An ellipsoid is fitted to the surface given by a constant autocorrelation coefficient value and the major and minor axes are used to determine the inclination angle of turbulent structures in the plane of shear. The inclination angle is observed to be fairly unaffected by the choice of the autocorrelation coefficient value. It was found that the inclination angle decreases with increasing Richardson number. The structure of the turbulent motion, as characterized by the inclination angle, is therefore directly related to the eventual evolution of the turbulence, as described by the growth or decay rate of the turbulent kinetic energy.

4:56PM L13.00003 ABSRACT WITHDRAWN –

5:09PM L13.00004 ABSTRACT WITHDRAWN –

5:22PM L13.00005 Mixing efficiency of turbulent patches in stably stratified flows¹, AMRAPALI GARANAIK², SUBHAS KARAN VENAYAGAMOORTHY³, Colorado State University — A key quantity that is essential for estimating the turbulent diapycnal (irreversible) mixing in stably stratified flow is the mixing efficiency R_f^* , which is a measure of the amount of turbulent kinetic energy that is irreversibly converted into background potential energy. In particular, there is an ongoing debate in the oceanographic mixing community regarding the utility of the buoyancy Reynolds number (Re_b), particularly with regard to how mixing efficiency and diapycnal diffusivity vary with Re_b . Specifically, is there a universal relationship between the intensity of turbulence and the strength of the stratification that supports an unambiguous description of mixing efficiency based on Re_b ? The focus of the present 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. Field data analysis has done by identifying turbulent patches using Thorpe sorting method for potential density. The analysis clearly shows that high mixing efficiencies can persist at high buoyancy Reynolds numbers. This is contradiction to previous studies which predict that mixing efficiency should decrease universally for Re_b greater than $O(100)$.

¹Funded by NSF and ONR

²Ph.D. Candidate

³Associate Professor

5:35PM L13.00006 Multiscale equations for strongly stratified turbulent flows , GREG

CHINI, University of New Hampshire, CESAR ROCHA, Scripps Institution of Oceanography, University of California, San Diego, KEITH JULIEN, University of Colorado, Boulder, COLM-CILLE CAULFIELD, BP Institute, University of Cambridge — Strongly stratified turbulent shear flows are of fundamental importance owing to their widespread occurrence and their impact on diabatic mixing, yet direct numerical simulations of such flows remain challenging. Here, a reduced, multiscale description of turbulent shear flows in the presence of strong stable density stratification is derived via asymptotic analysis of the governing Boussinesq equations. The analysis explicitly recognizes the occurrence of dynamics on disparate spatiotemporal scales, and yields simplified partial differential equations governing the coupled evolution of slowly-evolving small aspect-ratio ('pancake') modes and isotropic, strongly non-hydrostatic stratified-shear (e.g. Kelvin–Helmholtz) instability modes. The reduced model is formally valid in the physically-relevant regime in which the aspect-ratio of the pancake structures tends to zero in direct proportion to the horizontal Froude number. Relative to the full Boussinesq equations, the model offers both computational and conceptual advantages.

5:48PM L13.00007 Interaction between a vertical turbulent buoyant jet and a thermocline¹ , EKATERINA EZHOVA, LUCA BRANDT, Linne FLOW Centre and SeRC, KTH Mechanics, Stockholm, Sweden, CLAUDIA CENEDESE, Physical Oceanography Department, Woods Hole Oceanographic Institution, Woods Hole, Massachusetts, USA — We study the behaviour of an axisymmetric vertical turbulent jet in an unconfined stratified environment by means of well-resolved large eddy simulations (LES). The stratification is two layers separated by a thermocline and the thermocline thickness considered is smaller and on the order of the jet diameter at the thermocline entrance. We quantify mean jet penetration, stratified turbulent entrainment and study the generation of internal waves. The mean jet penetration is predicted based on the conservation of the source energy in the thermocline. The entrainment coefficient for the thin thermocline agrees with the theoretical model for a two-layer stratification with a sharp interface. A secondary flow towards the jet top appears in the upper part of the thick thermocline. The jet generates internal waves at frequencies in agreement with similar experiments. We shall also report the results of LES of a turbulent plume in a stratified fluid modelling subglacial discharge from a submarine glacier in stratifications typical of Greenland fjords. We consider a free plume from a round source of various diameters with double the total discharge estimated from the field data. We quantify plume dynamics and compare the results for plumes and jets.

¹E. Ezhova acknowledges VR Swedish Research Council. The authors acknowledge computer time provided by SNIC (Swedish National Infrastructure for Computing).

6:01PM L13.00008 A numerical investigation of the interaction between a horizontal density gradient and an oscillating turbulent flow¹ , STEVEN KAPTEIN, MATIAS DURAN-MATUTE, Eindhoven Univ of Tech, VINCENZO ARMENIO, Universita degli studi di Trieste, FEDERICO ROMAN, IEFLUIDS S.r.l., HERMAN CLERCX, Eindhoven Univ of Tech — In coastal areas, river outflow provides a large buoyancy input that leads to strong horizontal density gradients. These density gradients are associated to complex hydrodynamics such as, penetration of fresh water currents in the ocean, coastal currents or strain-induced periodic stratification. One key governing mechanism is the interaction between stirring by the tides and horizontal density gradients which influences mixing . In order to investigate this mechanism and gain new insight into the mixing process, wall-resolving large eddy simulation (LES) are performed. The tide is simulated using a horizontal oscillating pressure gradient that acts perpendicular to a horizontal (unstable) linear density gradient. A decomposition of the density allows to apply periodic boundary conditions in the streamwise and spanwise directions, for both the velocity and the density. As the Reynolds number is limited by the computational time required for LES, simulations are performed for different values of the Richardson number.

¹Funded by STW

6:14PM L13.00009 ABSTRACT WITHDRAWN –

6:27PM L13.00010 Accurate Calculation of the Linear Response Function of General Circulation Models¹ , PEDRAM HASSANZADEH, Rice University, ZHIMING KUANG, Harvard University — A linear response function (LRF), M , relates the response, x , of a nonlinear system, such as the atmosphere, to weak external forcings, f , and tendencies, \dot{x} , via $\dot{x} = Mx + f$. Knowing the LRF of general circulation models (GCMs) helps with better understanding their internal and forced variability. But even for simple GCMs, M cannot be calculated from first principles due to the lack of a complete theory for eddy-mean flow feedbacks. We present a new framework to accurately calculate the LRFs of GCMs using Green's function: by applying a sufficiently large set of localized forcings, one at a time, to the GCM, then calculating the time-mean responses, and finally finding the LRF via matrix inversion. We discuss the accuracy and properties of the LRF of an idealized GCM that has been calculated using this approach. An eddy flux closure matrix that determines the turbulent eddy flux responses to mean-flow changes is also calculated. Some results on using this LRF to quantify the eddy feedbacks and probe causality in the midlatitude large-scale circulation will be discussed. It will also be shown that the poor performance of another common approach to calculating the LRFs, the Fluctuation-Dissipation Theorem, is linked to the non-normality of the LRFs of GCMs.

¹NSF AGS-1552385

Monday, November 21, 2016 4:30PM - 6:40PM –
Session L14 Free Surface Flows: Jets, Films, Propagation, and Spreading C125-126 -
Emmanuel Villermaux, Aix Marseille University

4:30PM L14.00001 The Whole Elephant: A Synoptic View of Liquid Rope Coiling , NEIL RIBE, Lab FAST, CNRS/Univ Paris-Sud — Liquid rope coiling is the instability that occurs when e.g. a thin stream of honey is poured onto toast. While we now have a fine-grained understanding of each of the four principal coiling modes (viscous, gravitational, inertio-gravitational and inertial), we still lack a global view of how the modes cohere to form a larger whole. Using a numerical continuation procedure, I determine how the dimensionless coiling frequency depends on the dimensionless fall height and flow rate, for several values of the dimensionless nozzle diameter. Starting with the onset of coiling, I propose a purely geometrical definition of the critical surface between coiling and no coiling as the locus of points where the radius a_1 of the rope at the contact point is just equal to the coil radius R . Coiling with $a_1 > R$ is impossible because the rope would intersect itself. I characterize the asymptotic limits of the critical surface as well as the structure of the supercritical volume inside that surface. The procedure reveals a new mode of coiling onset that has not yet been identified.

4:43PM L14.00002 Effect of Elasticity on Stability of Viscoelastic Liquid Curtain¹

, ALIREZA MOHAMMAD KARIM, WIESLAW SUSZYNISKI, LORRAINE FRANCIS, University of Minnesota, Twin Cities, MARCIO CARVALHO, Pontifcia Universidade Catlica do Rio de Janeiro, UNIVERSITY OF MINNESOTA, TWIN CITIES COLLABORATION, PONTIFICIA UNIVERSIDADE CATLICA DO RIO DE JANEIRO COLLABORATION, DOW CHEMICAL COMPANY COLLABORATION — Curtain coating is one the preferred methods for high-speed precision application of single-layer and multi-layer coatings in industry. Despite the extensive variety of applications of curtain coating, its operation is challenging and uniform coating is only obtained in a certain range of operating parameters, called the coating window. The two main physical mechanisms that limit curtain coating are the breakup of the liquid curtain, below a critical flow rate, and the catastrophic event of air entrainment, which occurs above a certain web speed. The rheological characteristics of the coating liquid play an important role on these mechanisms, but the fundamental understanding of the role of rheology is still not complete. In this work, we analyze the relative importance of shear and extensional viscosity on both curtain breakup and dynamic contact line instability (i.e. air entrainment). Aqueous solutions of polyethylene oxide (PEO) and polyethylene glycol (PEG) of different molecular weights were used as model liquids to obtain fluids with different levels of extensional thickening behavior.

¹We would like to acknowledge the financial support from the Dow Chemical Company.

4:56PM L14.00003 Effect of added polymer in free jets of a dilute polymer solution

, MARIE-CHARLOTTE RENOULT, JEAN-BAPTISTE CHARPENTIER, OLIVIER CRUMEYROLLE, INNOCENT MUTABAIZI, Normandie Univ, UNIHAVRE, CNRS, LOMC 76000 Le Havre, France — The instability of a free viscoelastic jet is experimentally investigated by extruding an aqueous solution containing five parts per million of Poly(ethylene oxide) into air from a sixty micrometers orifice at relative low speeds. A method of image analysis was developed to quantify the effect of the added polymer on the morphology and the stability of the jet breakup. Three main representations were considered: the area versus perimeter relation for all liquid objects detected on the images, i.e. jets and jet fragments, the equivalent diameter distribution of jet fragments and the standard deviation curve of jets profiles. The former two provide information on the morphology of jet fragments: distinction of two classes, products and residues, and existence of coalescence. The latter gives information on the jet breakup stability: measurement of the growth rate and initial amplitude of the jet instability and detection of beads-on-a-string structures in the jet interface deformation. Experimental results will be presented and compared to theory.

5:09PM L14.00004 Wave-front propagation of rinsing flows on rotating semiconductor wafers

, JOHN M. FROSTAD, University of British Columbia, ANDY YLITALO, DANIEL J. WALLS, Stanford University, DAVID S. L. MUI, LAM Research, GERALD G. FULLER, Stanford University — The semiconductor manufacturing industry is migrating to a cleaning technology that involves dispersing cleaning solutions onto a rotating wafer, similar to spin-coating. Advantages include a more continuous overall fabrication process, lower particle level, no cross contamination from the back side of a wafer, and less usage of harsh chemicals for a lower environmental impact. Rapid rotation of the wafer during rinsing can be more effective, but centrifugal forces can pull spiral-like ribbons of liquid radially outward from the advancing wave-front where particles can build up, causing higher instances of device failure at these locations. A better understanding of the rinsing flow is essential for reducing yield losses while taking advantage of the benefits of rotation. In the present work, high-speed video and image processing are used to study the dynamics of the advancing wave-front from an impinging jet on a rotating substrate. The flow-rate and rotation-speed are varied for substrates coated with a thin layer of a second liquid that has a different surface tension than the jet liquid. The difference in surface tension of the two fluids gives rise to Marangoni stresses at the interface that have a significant impact on the rinsing process, despite the extremely short time-scales involved.

5:22PM L14.00005 Simulating wave-turbulence on thin elastic plates with arbitrary boundary conditions

, WIM M. VAN REES, L. MAHADEVAN, Harvard University — The statistical characteristics of interacting waves are described by the theory of wave turbulence, with the study of deep water gravity wave turbulence serving as a paradigmatic physical example. Here we consider the elastic analog of this problem in the context of flexural waves arising from vibrations of a thin elastic plate. Such flexural waves generate the unique sounds of so-called thunder machines used in orchestras - thin metal plates that make a thunder-like sound when forcefully shaken. Wave turbulence in elastic plates is typically investigated numerically using spectral simulations with periodic boundary conditions, which are not very realistic. We will present the results of numerical simulations of the dynamics of thin elastic plates in physical space, with arbitrary shapes, boundary conditions, anisotropy and inhomogeneity, and show first results on wave turbulence beyond the conventionally studied rectangular plates. Finally, motivated by a possible method to measure ice-sheet thicknesses in the open ocean, we will further discuss the behavior of a vibrating plate when floating on an inviscid fluid.

5:35PM L14.00006 Propagation of a viscous thin film over an elastic membran

, ZHONG ZHENG, Department of Mechanical and Aerospace Engineering, Princeton University, IAN GRIFFITHS, Mathematical Institute, University of Oxford, HOWARD STONE, Department of Mechanical and Aerospace Engineering, Princeton University — We study the buoyancy-driven spreading of a thin viscous film over a thin elastic membrane. Neglecting the effects of membrane bending and the membrane weight, we study the case of constant fluid injection and obtain a system of coupled partial differential equations to describe the shape of the air-liquid interface, and the deformation and the radial tension of the stretched membrane. We obtain self-similar solutions to describe the dynamics. In particular, in the early time period, the dynamics is dominated by buoyancy-driven spreading of the liquid film, and membrane stretching is a response to the buoyancy-controlled distribution of liquid weight; the location of the liquid front obeys the power-law form $r_f(t) \propto t^{1/2}$. However, in the late time period, the system is quasi-steady, the air-liquid interface is flat, and membrane stretching, due to the liquid weight, causes the spreading of the liquid front; the location of the front obeys a different power-law form $r_f(t) \propto t^{1/4}$ before the edge effects of the membrane become significant. In addition, we report laboratory experiments for constant fluid injection using different viscous liquids and thin elastic membranes. Very good agreement is obtained between the theory and experiments.

5:48PM L14.00007 ABSTRACT WITHDRAWN —

6:01PM L14.00008 Dry patches in a flowing film : Predicting rewetting and the effects of inertia

, LUC LEBON, Laboratoire Matiere et Systemes Complexes (MSC), CNRS / Univ. Paris 7, JULIEN SEBILLEAU, IMFT / Univ. Toulouse, LAURENT LIMAT, Laboratoire Matiere et Systemes Complexes (MSC), CNRS / Univ. Paris 7 — We study the effects of inertia on the shape and stability of dry patches using liquids of decreasing viscosities. These dry patches are formed when a liquid film flows down along a substrate under partial wetting conditions. They become stationary and exhibit an arch shape well described by a simple viscous model developed long ago by Podgorski. Surprisingly, this arch shape appears to be robust when one decreases the fluid viscosity which increases inertial effects, but the evolution of the apex curvature upon flow rate is strongly affected. We here proposed an improved description of the dry patch evolution taking into account several physical effects as the hydrostatic pressure in the liquid film, the curvature of the contact line, and these inertial effects. These ones affect both the mechanical equilibrium of the rim surrounding the dry patch and the flow inside the rim. This model allows us to show that the dry patch shape remains extremely close to the viscous-Podgorski- prediction but with a rescaling of the apex curvature. It also allows us to get a better prediction of the apex curvature dependence upon flow rate and a prediction of the rewetting threshold above which dry patches are swept away by the film flow.

6:14PM L14.00009 Node dynamics and cusps size distribution at the border of liquid sheets, EMMANUEL VILLERMAUX, CHRISTOPHE ALMARCHA, Aix Marseille Université, CNRS, Centrale Marseille, IRPHE UMR 7342, 13384 Marseille, France — We study the intrinsic dynamics of cusps, or indentations, moving along a liquid sheet border, and characterize their ensemble statistics. Gordillo and collaborators (*J. Fluid Mech.*, 754 (2014)) have shown that the symmetrical stationary cusp is the only structure accommodating for both mass and momentum conservation at a steadily receding liquid sheet rim. Cusps are also known to typically move along a sheet border, to present an asymmetry, and to be distributed in size around a mean. We show here why an heterogeneous assembly of cusps travelling along the sheet rim occurs spontaneously, why big and small cusps coexist at the same time and, more precisely, we establish the specific link between the microscopic dynamics directing their motion, and the ensemble averaged distribution of their sizes.

6:27PM L14.00010 Non-classical dispersive shock waves in shallow water, PATRICK SPRENGER, MARK HOEFER, Univ of Colorado - Boulder — A classical model for shallow water waves with strong surface tension is the Kawahara equation, which is the Korteweg-de Vries (KdV) including a fifth order derivative term. A particular problem of interest to these types of equations is step initial data, known as the Riemann problem, which results in a shock in finite time. Unlike classical shock waves, where a discontinuity is resolved by dissipation, the dispersive regularization results in the discontinuity resolved as a dispersive shock wave (DSW). When parameter choices result in non-convex dispersion, three distinct dynamic regimes are observed that can be characterized solely by the amplitude of the initial step. For small jumps, a perturbed KdV DSW with positive polarity and orientation is generated, accompanied by small amplitude radiation from an embedded solitary wave leading edge, termed a radiating DSW. For moderate jumps, a crossover regime is observed with waves propagating forward and backward from the sharp transition region. For sufficiently large jumps, a new type of DSW is observed we term a translating DSW were a partial, non-monotonic, negative solitary wave at the trailing edge is connected to an interior nonlinear periodic wave and exhibits features common to both dissipative and dispersive shock waves.

Monday, November 21, 2016 4:30PM - 6:40PM –

Session L15 Bio: Cardiovascular Flow III E143/144 - Oscar Flores, Universidad Carlos III de Madrid

4:30PM L15.00001 Patient-specific analysis of blood stasis in the left atrium¹, OSCAR FLORES, ALEJANDRO GONZALO, MANUEL GARCIA-VILLALBA, Universidad Carlos III de Madrid, LORENZO ROSSINI, ALBERT HSIAO, ELLIOT MCVEIGH, ANDREW M. KAHN, JUAN C. DEL ALAMO, University of California, San Diego — Atrial fibrillation (AF) is a common arrhythmia in which the left atrium (LA) beats rapidly and irregularly. Patients with AF are at increased risk of thromboembolic events (TE), particularly stroke. Anticoagulant therapy can reduce the risk of TE in AF, but it can also increase the risks of adverse events such as internal bleeding. The current lack of tools to predict each patient's risk of LA thrombogenesis makes it difficult to decide whether to anticoagulate patients with AF. The aim of this work is to evaluate blood stasis in patient-specific models of the LA, because stasis is a known thrombogenesis risk factor. To achieve our aim, we performed direct numerical simulations of left atrial flow using an immersed boundary solver developed at the UC3M, coupled to a 0D model for the pulmonary circulation. The LA geometry is obtained from time-resolved CT scans and the parameters of the 0D model are found by fitting pulmonary vein flow data obtained by 4D phase contrast MRI. Blood stasis is evaluated from the flow data by computing blood residence time together with other kinematic indices of the velocity field (e.g. strain and kinetic energy). We focus on the flow in the left atrial appendage, including a sensitivity analysis of the effect of the parameters of the 0D model.

¹Funded by the Spanish MECD, the Clinical and Translational Research Institute at UCSD and the American Heart Association.

4:43PM L15.00002 Vortex and energy characteristics of flow in the left ventricle following progressive severities of aortic valve regurgitation, GIUSEPPE DI LABBIO, LYSE KADEM, Concordia University — During the heart's filling phase, a notorious vortex is known to develop in the left ventricle (LV). Improper development and poor energetic behavior of this vortex can be correlated with cardiac disease. In particular, during aortic valve regurgitation (leakage of blood through the aortic valve during LV filling), this vortex is forced to interact with a jet emanating from a regurgitant orifice in the valve. The ensuing flow in the left ventricle subject to this disease has yet to be fully characterized and may lead to new indices for evaluation of its severity. As such, this experimental work investigates flow in a model LV subject to aortic regurgitation on a novel double-activation left heart duplicator for six progressive grades of regurgitation (beginning from the healthy case). Double-activation (independent activation of the atrium and ventricle) is critical to the simulation of this pathology. Regurgitation is induced by restricting the closure of the aortic valve to a centralized orifice. The velocity fields for each case are acquired using 2D time-resolved particle image velocimetry. Viscous energy dissipation and vortex formation time are investigated and found to significantly increase as the pathology progresses, while a histogram of vorticity tends toward a shifted and depressed Gaussian distribution. Proper orthogonal decomposition reveals significant disruption of the dominant energetic coherent structures.

4:56PM L15.00003 Flow in patients with left ventricular thrombus using optimized echo PIV-PTV, KAUSHIK SAMPATH, THURA T HARFI, RICHARD T GEORGE, JOSEPH KATZ, Johns Hopkins Univ — Applications of echocardiographic particle image velocimetry and particle tracking velocimetry (echo PIV-PTV) for characterizing cardiovascular flows have been expanding. It involves acquisition and processing of time-resolved contrast echocardiograms of in vivo flows seeded with micro bubbles. Here, a set of image enhancement and particle tracking methods are implemented on in-vivo data from five patients with history of past or current confirmed left ventricular thrombus (LVT). Our aim is to correlate the LV mixing with thrombogenicity. For cases with persistent LVT and low cardiac efficiency, results show low velocities around the clot as well as low penetration depth of the jet at the exit from the mitral valve and strength of the associated LV vortex. The speeds increase with increasing kinesis (wall motion) and distance from the clot. Patients with recovering cardiac function and diminishing clot size exhibit improved flow, LV vortex strength and penetration even when their efficiency is lower than normal. Trends of the apical washing are consistent with those predicted by the E-wave propagation index.

5:09PM L15.00004 Experimental study of the intraventricular filling vortex in diastolic dysfunction, ARVIND SANTHANAKRISHNAN, MILAD SAMAEI, NICHOLAS NELSEN, Oklahoma State University — Heart failure with normal ejection fraction (HFNEF) is a clinical syndrome that is prevalent in over half of heart failure patients. HFNEF patients typically show diastolic dysfunction, caused by a decrease in relaxation capability of the left ventricular (LV) muscle tissue and/or an increase in LV chamber stiffness. Numerous studies using non-invasive medical imaging have shown that an intraventricular filling vortex is formed in the LV during diastole. We conducted 2D particle image velocimetry and hemodynamics measurements on a left heart simulator to investigate diastolic flow under increasing LV wall stiffness, LV wall thickness and heart rate (HR) conditions. Flexible-walled, optically clear LV physical models cast from silicone were fitted within a fluid-filled acrylic chamber. Pulsatile flow within the LV model was generated using a piston pump and 2-component Windkessel elements were used to tune the least stiff (baseline) LV model to physiological conditions. The results show that peak circulation of the intraventricular filling vortex is diminished in conditions of diastolic dysfunction as compared to the baseline case. Increasing HR exacerbated the circulation of the filling vortex across all cases.

5:22PM L15.00005 Patient-specific assessment of left ventricular thrombogenesis risk after acute myocardial infarction: a pilot clinical study, L ROSSINI, A KHAN, J C DEL ALAMO, UC San Diego, P MARTINEZ-LEGAZPI, C PREZ DEL VILLAR, Y BENITO, R YOTTI, A BARRIO, A DELGADO-MONTERO, A GONZALEZ-MANSILLA, F FERNANDEZ-AVILS, J BERMEJO, Hospital Gregorio Maranon, Madrid — Left ventricular thrombosis (LVT) is a major complication of acute myocardial infarction (AMI). In these patients, the benefits of chronic anticoagulation therapy need to be balanced with its pro-hemorrhagic effects. Blood stasis in the cardiac chambers, a risk factor for LVT, is not addressed in current clinical practice. We recently developed a method to quantitatively assess the blood residence time (RT) inside the left ventricle (LV) based on 2D color-Doppler velocimetry (echo-CDV). Using time-resolved blood velocity fields acquired non-invasively, we integrate a modified advection equation to map intraventricular stasis regions. Here, we present how this tool can be used to estimate the risk of LVT in patients with AMI. 73 patients with a first anterior-AMI were studied by echo-CDV and RT analysis within 72h from admission and at a 5-month follow-up. Patients who eventually develop LVT showed early abnormalities of intraventricular RT: the apical region with $RT > 2s$ was significantly larger, had a higher RT and a longer wall contact length. Thus, quantitative analysis of intraventricular flow based on echocardiography may provide subclinical markers of LV thrombosis risk to guide clinical decision making.

5:35PM L15.00006 Patient-Specific Modeling of Interventricular Hemodynamics in Single Ventricle Physiology¹, VIJAY VEDULA, JEFFREY FEINSTEIN, ALISON MARSDEN, Stanford University — Single ventricle (SV) congenital heart defects, in which babies are born with only functional ventricle, lead to significant morbidity and mortality with over 30% of patients developing heart failure prior to adulthood. Newborns with SV physiology typically undergo three palliative surgeries, in which the SV becomes the systemic pumping chamber. Depending on which ventricle performs the systemic function, patients are classified as having either a single left ventricle (SLV) or a single right ventricle (SRV), with SRV patients at higher risk of failure. As the native right ventricles are not designed to meet systemic demands, they undergo remodeling leading to abnormal hemodynamics. The hemodynamic characteristics of SLVs compared with SRVs is not well established. We present a validated computational framework for performing patient-specific modeling of ventricular flows, and apply it across 6 SV patients (3SLV + 3SRV), comparing hemodynamic conditions between the two subgroups. Simulations are performed with a stabilized finite element method coupled with an immersed boundary method for modeling heart valves. We discuss identification of hemodynamic biomarkers of ventricular remodeling for early risk assessment of failure.

¹This research is supported in part by the Stanford Child Health Research Institute and the Stanford NIH-NCATS-CTSA through grant UL1 TR001085 and due to U.S. National Institute of Health through NIH NHLBI R01 Grants 5R01HL12972702 and 5R01HL121754-03.

5:48PM L15.00007 Velocity and Vorticity in the Right Heart from 4DMRI Measurements, JEAN HERTZBERG, University of Colorado Boulder, JAMES BROWNING, Northeastern University, BRETT FENSTER, National Jewish Health Center — Measurements of blood flow in the human heart were made using time-resolved 3D cardiac magnetic resonance phase contrast flow imaging (4DMRI). This work focuses on blood flow in the right ventricle (RV) and right atrium (RA) in both normal subjects and patients with pulmonary hypertension (PH). Although cardiac output is unchanged early in the disease, details of the flow field differ between normals and PH patients. In particular, vorticity at peak diastole has been found to correlate with PH. The underlying physics of this difference are being explored by a qualitative visual comparison of 3D flow structures in the vena cava, RA, and RV between healthy subjects and pulmonary hypertensive patients.

6:01PM L15.00008 Hemodynamic Assessment of Compliance of Pre-Stressed Pulmonary Valve-Vasculature in Patient Specific Geometry Using an Inverse Algorithm, ULLHAS HEBBAR, ANUP PAUL, RUPAK BANERJEE, School of Dynamic Systems, University of Cincinnati — Image based modeling is finding increasing relevance in assisting diagnosis of Pulmonary Valve-Vasculature Dysfunction (PWD) in congenital heart disease patients. This research presents compliant artery – blood interaction in a patient specific Pulmonary Artery (PA) model. This is an improvement over our previous numerical studies which assumed rigid walled arteries. The impedance of the arteries and the energy transfer from the Right Ventricle (RV) to PA is governed by compliance, which in turn is influenced by the level of pre-stress in the arteries. In order to evaluate the pre-stress, an inverse algorithm was developed using an in-house script written in MATLAB and Python, and implemented using the Finite Element Method (FEM). This analysis used a patient specific material model developed by our group, in conjunction with measured pressure (invasive) and velocity (non-invasive) values. The analysis was performed on an FEM solver, and preliminary results indicated that the Main PA (MPA) exhibited higher compliance as well as increased hysteresis over the cardiac cycle when compared with the Left PA (LPA). The computed compliance values for the MPA and LPA were 14% and 34% lesser than the corresponding measured values. Further, the computed pressure drop and flow waveforms were in close agreement with the measured values. In conclusion, compliant artery – blood interaction models of patient specific geometries can play an important role in hemodynamics based diagnosis of PWD.

6:14PM L15.00009 Image-based modeling of blood flow and oxygen transfer in feto-placental capillaries, PHILIP PEARCE, Massachusetts Institute of Technology, OLIVER JENSEN, University of Manchester — During pregnancy, oxygen diffuses from maternal to fetal blood through the placenta. At the smallest scale of the feto-placental vasculature are the terminal villi, bulbous structures that are thought to be the main sites for oxygen transfer in the final trimester of pregnancy. The objective of this study is to investigate blood flow and oxygen transfer in the terminal villi of the placenta. Three-dimensional representations of villous and capillary surfaces, obtained from confocal laser scanning microscopy, are converted to finite-element meshes. Simulations of blood flow and oxygen transfer are performed to calculate the vascular flow resistance of the capillaries and the total oxygen transfer rate from the maternal blood. Scaling arguments, which predict the oxygen transfer across a range of Pecllet numbers, are shown to be an efficient tool for quantifying the effect of statistical variability and experimental uncertainty. The effect of commonly observed localised dilations in the fetal vasculature on oxygen transfer is quantified using an idealised model in a simplified geometry. The model predicts how, for a fixed pressure drop through a capillary, oxygen transfer is maximised by an optimal shape of the dilation, leading to an increase in oxygen transfer of up to 15

6:27PM L15.00010 Dynamic Mode Decomposition Bio-Markers for Left Ventricle

Flow, MARIA BORJA, MAE Department, UC San Diego, La Jolla 92093 CA, PABLO MARTINEZ-LEGAZPI, YOLANDA BENITO, RAQUEL YOTTI, FRANCISCO FERNANDEZ-AVILES, JAVIER BERMEJO, Hospital Gregorio Maranon, Madrid 28040, Spain, JUAN C. DEL ALAMO, MAE Department, UC San Diego, La Jolla 92093 CA — Dynamic mode decomposition (DMD) is a tool used in the fluid community to extract a set of modes that describe the underlying fluid dynamics in a set of flow fields generated experimentally or by numerical simulations. Despite advances in medical imaging, characterization of some cardiac dysfunctions has remained a challenge and diagnosis is often subjective. This study presents a novel DMD method to objectively characterize left ventricular (LV) flow in healthy volunteers and patients with dilated cardiomyopathy (DCM) and hypertrophic cardiomyopathy (HCM). Our approach is based on assessing temporal evolution dependent mode structures from two-dimensional velocity fields, obtained experimentally using echocardiographic color Doppler velocimetry, and defined with a common unit normal moving LV coordinate system. Using the mode structures as a basis, we reconstruct the flow field, determine the key contributing modes, and obtain a reduced order model. Using 20 healthy volunteers, 20 DCM patients and 20 HCM patients, our results show quantitative and qualitative differences between healthy and in the DCM and HCM patients. This study suggests that temporal evolution dependent modes can be used as bio-markers to assess in-vivo LV flow.

Monday, November 21, 2016 4:30PM - 6:27PM –

Session L16 Drops: Bouncers and Walkers D133/134 - Luiz Faria, MIT

4:30PM L16.00001 A model for Faraday pilot-waves over variable topography , LUIZ

FARIA, MIT — In 2005 Yves Couder and co-workers discovered that droplets walking on a vibrating bath possess certain features previously thought to be exclusive to quantum systems. These millimetric droplets synchronize with their Faraday wavefield, creating a macroscopic pilot-wave system. In this talk we exploit the fact that the waves generated are nearly monochromatic and propose a hydrodynamic model capable of capturing the interaction between bouncing drops and a variable topography. We show that our model is able to reproduce some important experiments involving the drop-topography interaction, such as non-specular reflection and single-slit diffraction.

4:43PM L16.00002 The chaotic interaction of two walkers¹ , LOIC TADRIST, NARESH SAMARA,

PETER SCHLAGHECK, TRISTAN GILET, Univ de Liege — A droplet bouncing on a vertically vibrated bath may be propelled horizontally by the Faraday waves that it generates at each rebound. This association of a wave and a particle is called a walker. Ten years ago, Yves Couder and co-workers noted that the dynamical encounter of two walkers may lead to either scattered trajectories or orbital motion. In this work, we investigate the interaction of two walkers more systematically. The walkers are launched towards each other with finely controlled initial conditions. Output trajectories are classified in four types: scattering, orbiting, wandering and complex. The interaction appears stochastic: the same set of initial parameters (to the measurement accuracy) can produce different outputs. Our analysis of the underlying chaos provides new insights on the stochastic nature of this experiment.

¹This work is supported by the ARC Quandrops of the Wallonia-Brussels Federation.

4:56PM L16.00003 Snell's law and walking droplets¹ , JOHN BUSH, Department of Mathematics, MIT,

GIUSEPPE PUCCI, Department of Mathematics, MIT; University of Haifa, BENJAMIN AUBIN, PIERRE-TOMAS BRUN, LUIZ FARIA, Department of Mathematics, MIT — Droplets walking on the surface of a vibrating bath have been shown to exhibit a number of quantum-like features. We here present the results of a combined experimental and theoretical investigation of such droplets crossing a linear step corresponding to a reduction in bath depth. When the step is sufficiently large, the walker reflects off the step; otherwise, it is refracted as it crosses the step. Particular attention is given to an examination of the regime in which the droplet obeys a form of Snell's Law, a behavior captured in accompanying simulations. Attempts to provide theoretical rationale for the dependence of the effective refractive index on the system parameters are described.

¹Supported by NSF through CMMI-1333242

5:09PM L16.00004 Diffraction and interference of walking drops¹ , GIUSEPPE PUCCI, De-

partment of Mathematics, MIT; The Hatter Department of Marine Technologies, University of Haifa, DANIEL M. HARRIS, Department of Mathematics, UNC Chapel Hill; Department of Mathematics, MIT, JOHN W. M. BUSH, Department of Mathematics, MIT — A decade ago, Yves Couder and Emmanuel Fort discovered a wave-particle association on the macroscopic scale: a drop can bounce indefinitely on a vibrating bath of the same liquid and can be piloted by the waves that it generates. These walking droplets have been shown to exhibit several quantum-like features, including single-particle diffraction and interference. Recently, the original diffraction and interference experiments of Couder and Fort (Couder, Y. & Fort, E. *Phys. Rev. Lett.* **97**, 154101 (2006)) have been revisited and contested (Andersen, A. et al. *Phys. Rev. E* **92**(1) 013006 (2015)). We have revisited this system using an improved experimental set-up, and observed a strong dependence of the behavior on system parameters, including drop size and vibrational forcing. In both the single- and the double-slit geometries, the diffraction pattern is dominated by the interaction of the walking droplet with a planar boundary. Critically, in the double-slit geometry, the walking droplet is influenced by both slits by virtue of its spatially extended wave field.

¹NSF support via CMMI-1333242

5:22PM L16.00005 Non-local features of a hydrodynamic pilot-wave system , ANDRE

NACHBIN, IMPA/Brazil, MILES COUCHMAN, JOHN BUSH, Math. Dept./MIT — A droplet walking on the surface of a vibrating fluid bath constitutes a pilot-wave system of the form envisaged for quantum dynamics by Louis de Broglie: a particle moves in resonance with its guiding wave field. We here present an examination of pilot-wave hydrodynamics in a confined domain. Specifically, we present a one-dimensional water wave model that describes droplets walking in single and multiple cavities. The cavities are separated by a submerged barrier, and so allow for the study of tunneling. They also highlight the non-local dynamical features arising due to the spatially-extended wave field. Results from computational simulations are complemented by laboratory experiments.

5:35PM L16.00006 The energetics of bouncing droplets , SAM TURTON, Department of Mathematics,

Massachusetts Institute of Technology, JAN MOLACEK, Max Planck Institute for Dynamics and Self-Organization, JOHN BUSH, Department of Mathematics, Massachusetts Institute of Technology — We present the results of a theoretical investigation of the energetics of droplets bouncing on the surface of a vertically vibrating bath. We first assess the relative magnitudes of the kinetic, surface and gravitational potential energies of both the droplet and its wave field. We then seek to rationalize the transitions between the various bouncing and walking states that arise as the vibrational forcing is increased. Our results are compared with prior theoretical and experimental work.

5:48PM L16.00007 The role of bouncing-phase variation for walking droplets¹, LUCAS

TAMBASCO, Massachusetts Institute of Technology, ANAND OZA, Courant Institute - NYU, LUIZ FARIA, JOHN BUSH, Massachusetts Institute of Technology — Experimental and theoretical studies of droplets walking on a vibrating bath show that the droplets impact phase depends on the driving acceleration. Experiments also show that this phase may change in the presence of boundaries or other walkers, indicating a dependence of phase on local wave amplitude. One expects that this phase variation may alter the stability of various dynamical states. We here introduce an integro-differential model for a walker's horizontal motion that accounts for the variability of impact phase, and use it to predict the stability of rectilinear walking and orbital solutions. Our model predictions are compared with those of previous constant-phase models and related to experiments whenever possible.

¹NSF support via CMMI-1333242

6:01PM L16.00008 Walking droplets in confined domains¹, PEDRO SÁENZ, JOHN BUSH, Massachusetts Institute of Technology — A millimetric liquid drop can walk spontaneously along the surface of a vibrating fluid bath, propelled by a resonant interaction with its own wave field. These walking droplets exhibit features previously thought to be exclusive to the microscopic quantum realm. We here explore experimentally the dynamics and statistics of this macroscopic wave-particle system in confined domains, or 'corrals'. Particular attention is given to characterizing the influence of the corral geometry on the emergent probability distributions. The relation to analogous quantum systems (specifically, quantum corrals, the quantum mirage and scarring in Bose-Einstein condensates) is discussed.

¹NSF support via CMMI-1333242

6:14PM L16.00009 Pairs of Bouncing Droplets, MILES COUCHMAN, PIERRE CALDAIROU, Department

of Mathematics, Massachusetts Institute of Technology, ANAND OZA, Courant Institute of Mathematical Sciences, New York University, JOHN BUSH, Department of Mathematics, Massachusetts Institute of Technology — Multiple droplets bouncing on the surface of a vibrating fluid bath interact through the waves generated at each bounce. We here present the results of an experimental study of the behavior of two interacting droplets. As the driving acceleration of the bath is increased progressively, static bound states are found to destabilize into a variety of dynamical states including oscillating, orbiting, and ratcheting states. The type of instability depends on the droplet sizes and their separation distance. Attempts to provide theoretical rationale for the observed behavior are described.

Monday, November 21, 2016 4:30PM - 6:40PM –

Session L17 Reacting Flows: Modeling D131 - Hong Im, KAUST

4:30PM L17.00001 Modelling of Turbulent Scalar Fluxes in the Broken Reaction

Zones Regime.¹, HONG G. IM, King Abdullah Univ of Sci & Tech (KAUST), NILANJAN CHAKRABORTY, University of Newcastle, MARKUS KLEIN, CHRISTIAN KASTEN, Universitt der Bundeswehr Mnchen, PAUL ARIAS, University of Michigan — The LES filtered species transport equation in turbulent reacting flow simulations contains the unclosed turbulent scalar flux that needs to be modelled. It is well known that the statistical behavior of this term and its alignment characteristics with resolved scalar gradient depend on the relative importance of heat release and turbulent velocity fluctuations. Counter-gradient transport has been reported in several earlier studies where the flames under investigation were located either in the corrugated flamelets or thin reaction zones regime of premixed turbulent combustion. Therefore it is useful to understand the statistical behavior of turbulent scalar fluxes if the flame represents the broken reaction zones regime (BRZR). The present analysis aims to provide improved understanding on this subject through an a-priori analysis of a detailed chemistry database consisting of three freely-propagating statistically planar turbulent H₂-air premixed flames representing three different regimes of combustion. Results indicate that heat release effects weaken with increasing Karlovitz number, but that counter-gradient transport can still occur for large LES filter size in the BRZR. Furthermore the behaviour of the flux and in particular its sign are different for reactant and product species.

¹KAUST, EPSRC, KAUST Supercomputing Lab, N8, Archer

4:43PM L17.00002 ABSTRACT WITHDRAWN –**4:56PM L17.00003 Flamelet Regime Diagram for Turbulent Combustion Simulations**

, WAI LEE CHAN, MATTHIAS IHME, Stanford University, HEMANTH KOLLA, JACQUELINE CHEN, Sandia National Laboratories — The flamelet model has been widely used in numerical combustion investigations, particularly for the closure of large-eddy simulations (LES) of turbulent reacting flows. In most cases, the simulation results demonstrated good agreements with their experimental counterparts. However, a systematic analysis of the flamelet model's applicability, as well as its potential limitations, is seldom conducted, and the model performance is usually based only on *a-posteriori* comparisons. The objective of this work is to derive a metric that can formally quantify the suitability of the flamelet model in different flame configurations. For this purpose, a flamelet regime diagram has been developed and studied in the context of direct numerical simulations (DNS) of a turbulent lifted jet flame. The implementation of the regime diagram in LES has been investigated through explicit filtering of the DNS results.

5:09PM L17.00004 Modified Flamelet-Based Model for Non-Premixed High Speed Combustion, ZHIPENG LOU, FOLUSO LADEINDE, State Univ of NY- Stony Brook, WENHAI LI, TTC Technologies, Inc. Centereach, NY — The influence of static pressure and the use of Troe's model on flamelet solutions in supersonic combustion are studied. With various values of the background static pressure, we have observed significant effects on the flamelet solutions in such quantities as the quenching stoichiometric scalar dissipation rate, reaction rate of species and progress variable, heat release rate, and the temperature profile. In addition, the Troe's model shows opposite effects for low and high pressure conditions. The baseline flamelet table has been constructed with respect to mixture fraction and its stoichiometric scalar dissipation rate, where the information on both the stable and unstable flamelet solutions have been included. We have also experimented with the addition of pressure as an independent variable in the table, toward modeling compressibility and/or pressure-sensitive properties and the variable quenching conditions in real dual-mode scramjet operations.

5:22PM L17.00005 Examination of flamelet differential molecular diffusion models in oxy-fuel turbulent jet flames, CHAO HAN, Purdue University, ROBERT BARLOW, Sandia National Laboratories, HAIFENG WANG, Purdue University — Flamelet modeling of a series of oxy-fuel turbulent jet flames is conducted to examine the models capability for predicting the turbulence-chemistry interactions and the effect of differential molecular diffusion. There are two sets of the oxy-fuel jet flames, A series and B series. The A series includes three flames with a fixed Re and different Da, and the B series includes three flames with a fixed Da and different Re. These flames enable us to analyze the scaling of turbulence-chemistry interactions and differential molecular diffusion with respect to Re and Da, respectively. The modeling results are carefully compared with the experimental data for a critical assessment of the model. The models capability to reproduce the scaling with respect to Re and Da is also examined in detail.

5:35PM L17.00006 A consistent transported PDF model for treating differential molecular diffusion, HAIFENG WANG, PEI ZHANG, Purdue University — Differential molecular diffusion is a fundamentally significant phenomenon in all multi-component turbulent reacting or non-reacting flows caused by the different rates of molecular diffusion of energy and species concentrations. In the transported probability density function (PDF) method, the differential molecular diffusion can be treated by using a mean drift model developed by McDermott and Pope (Journal of Computational Physics, 226, 947-993, 2007). This model correctly accounts for the differential molecular diffusion in the scalar mean transport and yields a correct DNS limit of the scalar variance production. The model, however, misses the molecular diffusion term in the scalar variance transport equation, which yields an inconsistent prediction of the scalar variance in the transported PDF method. In this work, a new model is introduced to remedy this problem that can yield a consistent scalar variance prediction. The model formulation along with its numerical implementation is discussed, and the model validation is conducted in a turbulent mixing layer problem.

5:48PM L17.00007 An Investigation of a Hybrid Mixing Timescale Model for PDF Simulations of Turbulent Premixed Flames¹, HUA ZHOU, Center for Combustion Energy, Tsinghua University, MIKE KURON, Department of Mechanical Engineering, University of Connecticut; Computer Aided Engineering Associates, Inc., ZHUYIN REN, Center for Combustion Energy and School of Aerospace Engineering, Tsinghua University, TIANFENG LU, Department of Mechanical Engineering, University of Connecticut, JACQUELINE H. CHEN, Combustion Research Facility, Sandia National Laboratories — Transported probability density function (TPDF) method features the generality for all combustion regimes, which is attractive for turbulent combustion simulations. However, the modeling of micromixing due to molecular diffusion is still considered to be a primary challenge for TPDF method, especially in turbulent premixed flames. Recently, a hybrid mixing rate model for TPDF simulations of turbulent premixed flames has been proposed, which recovers the correct mixing rates in the limits of flamelet regime and broken reaction zone regime while at the same time aims to properly account for the transition in between. In this work, this model is employed in TPDF simulations of turbulent premixed methane-air slot burner flames. The model performance is assessed by comparing the results from both direct numerical simulation (DNS) and conventional constant mechanical-to-scalar mixing rate model.

¹This work is granted by NSFC 51476087 and 91441202

6:01PM L17.00008 An Inadequacy Formulation for an Uncertain Flamelet Model, DAVID SONDAK, TODD OLIVER, CHRIS SIMMONS, ROBERT MOSER, University of Texas at Austin — We report progress on the development of an uncertain flamelet library for use in non-premixed turbulent combustion. A stochastic inadequacy operator is generalized from previous work and is now used to incorporate uncertainties in chemical reaction mechanisms in a flamelet model. The original form of the inadequacy operator was designed to enforce positivity of chemical species concentrations and conservation of species while representing inadequacies in reduced chemical mechanisms. As a first step towards generalization, we are exploring temperature dependent modifications to the inadequacy operator. The temperature dependence helps ensure that the operator is inactive in the absence of chemical reactions and becomes active only after ignition. A Bayesian inverse problem is used to calibrate the stochastic operator on a hydrogen-oxygen zero-dimensional reactor and to infer model parameters, and their uncertainties, from data obtained via a detailed chemical mechanism. The inferred model parameters are then propagated through a laminar, non-premixed, counterflow hydrogen-oxygen flame. Temperature and species profiles at various scalar dissipation rates are compared to those predicted from a five-reaction reduced model and the detailed model.

6:14PM L17.00009 Towards Model Inadequacy Representations for Flamelet-Based RANS Combustion Simulations, TODD OLIVER, M.K. LEE, DAVID SONDAK, CHRIS SIMMONS, ROBERT MOSER, Univ of Texas, Austin — Flamelet-based RANS simulations are commonly used in combustion engineering. In such simulations, chemical reactions are represented by a “flamelet-library” of laminar diffusion flame solutions generated with some chemical mechanism, and turbulence is represented using typical eddy-viscosity-based RANS closures. Modeling errors are introduced through both of these models as well as their interaction. In this work, we formulate and apply physics-based stochastic model inadequacy representations to capture the effects of possible modeling errors, allowing their impact on quantities of interest to be estimated. Specifically, the uncertainty introduced by inadequacy of the chemical mechanism is represented using a recently developed stochastic operator approach, which is extended to the diffusion flame here, leading to a stochastic diffusion flame library. A Karhunen-Loeve decomposition applied to these random fields enables low-dimensional representation of this uncertainty. A stochastic extension of typical eddy-viscosity-based RANS models is developed to represent inadequacy in the turbulence closures. The full stochastic model is demonstrated on simulations of a planar jet flame.

6:27PM L17.00010 Reduction of Chemical Models under Uncertainty¹, HABIB NAJM, Sandia National Laboratories, Livermore, CA 94551, USA, RICCARDO MALPICA GALASSI, MAURO VALORANI, Sapienza University of Rome, Rome, Italy — We discuss recent developments for dynamical analysis and reduction of hydrocarbon fuel chemical kinetic models under uncertainty. We rely on computational singular perturbation analysis, allowing for uncertainties in reaction rate parameters. We outline a construction for representation of uncertain reduced chemical models, and estimation of probabilities for inclusion of sets of reactions in the reduced model. We demonstrate the approach in the context of homogeneous ignition of a hydrocarbon fuel-air mixture, illustrating the robustness of the reduced model under parametric uncertainty.

¹Support provided by the US Dept. of Energy, Office of Basic Energy Sciences, Division of Chemical Sciences, Geosciences, & Biosciences.

Monday, November 21, 2016 4:30PM - 6:27PM –
Session L18 Flow Instability: Thin Film D135 - Satish Kumar, University of Minnesota

4:30PM L18.00001 Self-similarity and scaling transitions during rupture of thin free films of Newtonian fluids, SUMEET THETE, CHRISTOPHER ANTHONY, School of Chemical Engineering, Purdue University, West Lafayette, IN 47907, PANKAJ DOSHI, Pfizer Worldwide RD, Groton, Connecticut 06340, USA, MICHAEL HARRIS, OSMAN BASARAN, School of Chemical Engineering, Purdue University, West Lafayette, IN 47907 — Rupture of thin liquid sheets (free films) is central to diverse industrial and natural phenomena, e.g. foam stability. Rupture of Newtonian films is analyzed under the competing influences of inertial, viscous, van der Waals, and capillary forces by solving numerically a system of spatially one-dimensional evolution equations for film thickness and lateral velocity. As the dynamics close to the rupture singularity is self-similar, the dynamics is also analyzed by solving a set of ordinary differential equations in similarity space. For sheets with negligible inertia, the dominant balance of forces involves solely viscous and van der Waals forces. By contrast, for sheets of inviscid fluids, the dominant balance is between inertial, capillary, and van der Waals forces. For real fluids, the afore-mentioned viscous and inertial regimes are demonstrated to be transitory and hence can only describe the initial thinning of highly viscous and slightly viscous sheets. Moreover, regardless of the fluids viscosity, it is shown that for sheets that initially thin in either of these two regimes, their dynamics transition to a final inertial-viscous regime in which all forces except capillary force remains important, in accordance with Vaynblat, Lister, and Witelski (2001).

4:43PM L18.00002 Intrinsic instability of thin liquid films on nanostructured surfaces, ARIF ROKONI, HAN HU, Drexel University, LIYONG SUN, The Behrend College, YING SUN, Drexel University — The instability of a thin liquid film on nanostructures is not well understood but is important in liquid-vapor two-phase heat transfer (e.g., thin film evaporation and boiling), lubrication, and nanomanufacturing. In thin film evaporation, the comparison between the non-evaporating film thickness and the critical film breakup thickness determines the stability of the film: the film becomes unstable when the critical film breakup thickness is larger than the non-evaporating film thickness. In this study, a closed-form model is developed to predict the critical breakup thickness of a thin liquid film on 2D periodic nanostructures based on minimization of system free energy in the limit of a liquid monolayer. Molecular dynamics simulations are performed for water thin films on square nanostructures of varying depth and wettability and the simulations agree with the model predictions. The results show that the critical film breakup thickness increases with the nanostructure depth and the surface wettability. The model developed here enables the prediction of the minimum film thickness for stable thin film evaporation on a given nanostructure.

4:56PM L18.00003 Liquid-Film Coating on Topographically Patterned Rotating Cylinders, WEIHUA LI, University of Minnesota, MARCIO CARVALHO, PUC-Rio, SATISH KUMAR, University of Minnesota — The coating of discrete objects having surface topography is an important step in the manufacturing of a broad variety of products. To develop fundamental understanding of this problem, we study liquid-film flow on rotating cylinders having sinusoidal topographical features. The Stokes equations, augmented with a term accounting for centrifugal forces, are solved in a rotating reference frame using the Galerkin finite element method. When gravitational effects are negligible, there is a critical rotation rate below which liquid accumulates over the troughs before merging to form multiple larger drops whose number depends on the topography wavelength and rotation rate. When the rotation rate is above this critical value, liquid accumulates over the crests with similar merging events. When gravitational forces become significant, liquid accumulates over the troughs, leading to a more even distribution of liquid around the cylinder relative to the case where topography is absent. These observations are in agreement with predictions from a lubrication-theory-based model provided that the free-surface curvatures are sufficiently small. For sufficiently large pattern amplitude, recirculation and flow reversal are observed, phenomena that could strongly influence film drying.

5:09PM L18.00004 Drying of Multicomponent Thin Films on Substrates with Topography, TRUONG PHAM, XIANG CHENG, SATISH KUMAR, University of Minnesota — Drying of multicomponent thin liquid films is an important step in coating and printing processes. In many cases, the substrate on which the film rests may possess topography, either intended or unintended. We present a lubrication-theory-based model describing the fundamentals of drying on such substrates. The film consists of volatile solvent and additional non-volatile components such as colloidal particles, surfactants, and non-volatile solvents. A system of one-dimensional partial differential equations accounting for the film height, depth-averaged concentration of bulk non-volatile components, and interfacial concentration of insoluble surfactant is derived. Evaporation is included using the well-known one-sided description, and the governing equations are solved with finite-difference methods to study various limiting cases. The results highlight the influence of evaporation rate, and thermal, surfactant, and solutal Marangoni flows on the final film thickness and colloidal particle distribution. We find that in a realistic region of parameter space, the addition of a non-volatile solvent yields a dried film that conforms to the substrate topography.

5:22PM L18.00005 Doubly-excited pulse-waves on flowing liquid films: experiments and numerical simulations¹, IDRIS ADEBAYO, ZHIHUA XIE, Imperial College London, ZHIZHAO CHE, Tianjin University, China, ALEX WRAY, OMAR MATAR, Imperial College London — The interaction patterns between doubly-excited pulse waves on a flowing liquid film are studied both experimentally and numerically. The flowing film is constituted on an inclined glass substrate while pulse-waves are excited on the film surface by means of a solenoid valve connected to a relay which receives signals from customised Matlab routines. The effect of varying the system parameters i.e. film flow rate, inter-pulse interval and substrate inclination angle on the pulse interaction patterns are then studied. Results show that different interaction patterns exist for these binary pulses; which include a singular behaviour, complete merger, partial merger and total non-coalescence. A regime map of these patterns is then plotted for each inclination angles examined, based on the film Re and the inter-pulse interval. Finally, the individual effect of the system parameters on the merging distance of these binary pulses in the merger mode is then studied and the results validated using both numerical simulations and mathematical modelling.

¹Funding from the Nigerian Government (for Idris Adebayo), and the EPSRC through a programme grant MEMPHIS (EP/K003976/1) gratefully acknowledged.

5:35PM L18.00006 Mass transfer in thin films under counter-current gas: experiments and numerical study, MATHIEU LUCQUIAUD, GIANLUCA LAVALLE, PATRICK SCHMIDT, The University of Edinburgh, ILJA AUSNER, MARC WEHRLI, Sulzer Chemtech Ltd, LENNON O NARAIGH, University College Dublin, PRASHANT VALLURI, The University of Edinburgh — Mass transfer in liquid-gas stratified flows is strongly affected by the waviness of the interface. For reactive flows, the chemical reactions occurring at the liquid-gas interface also influence the mass transfer rate. This is encountered in several technological applications, such as absorption units for carbon capture. We investigate the absorption rate of carbon dioxide in a liquid solution. The experimental set-up consists of a vertical channel where a falling film is sheared by a counter-current gas flow. We measure the absorption occurring at different flow conditions, by changing the liquid solution, the liquid flow rate and the gas composition. With the aim to support the experimental results with numerical simulations, we implement in our level-set flow solver a novel module for mass transfer taking into account a variant of the ghost-fluid formalism. We firstly validate the pure mass transfer case with and without hydrodynamics by comparing the species concentration in the bulk flow to the analytical solution. In a final stage, we analyse the absorption rate in reactive flows, and try to reproduce the experimental results by means of numerical simulations to explore the active role of the waves at the interface.

5:48PM L18.00007 Three-dimensional modelling of thin liquid films over spinning disks¹, KUN ZHAO, ALEX WRAY, JUNFENG YANG, OMAR MATAR, Imperial College London — In this research the dynamics of a thin film flowing over a rapidly spinning, horizontal disk is considered. A set of non-axisymmetric evolution equations for the film thickness, radial and azimuthal flow rates are derived using a boundary-layer approximation in conjunction with the Karman-Pohlhausen approximation for the velocity distribution in the film. These highly nonlinear partial differential equations are then solved numerically in order to reveal the formation of two and three-dimensional large-amplitude waves that travel from the disk inlet to its periphery. The spatio-temporal profile of film thickness provides us with visualization of flow structures over the entire disk and by varying system parameters(volumetric flow rate of fluid and rotational speed of disk) different wave patterns can be observed, including spiral, concentric, smooth waves and wave break-up in exceptional conditions. Similar types of waves can be found by experimentalists in literature and CFD simulation and our results show good agreement with both experimental and CFD results. Furthermore, the semi-parabolic velocity profile assumed in our model under the waves is directly compared with CFD data in various flow regimes in order to validate our model.

¹EPSRC UK Programme Grant EP/K003976/1

6:01PM L18.00008 Three-dimensional wave evolution on electrified falling films.¹, RUBEN TOMLIN, DEMETRIOS PAPAGEORGIOU, GREG PAVLIOU, Imperial College London — We consider the full three-dimensional model for a thin viscous liquid film completely wetting a flat infinite solid substrate at some non-zero angle to the horizontal, with an electric field normal to the substrate far from the flow. Thin film flows have applications in cooling processes. Many studies have shown that the presence of interfacial waves increases heat transfer by orders of magnitude due to film thinning and convection effects. A long-wave asymptotics procedure yields a Kuramoto-Sivashinsky equation with a non-local term to model the weakly nonlinear evolution of the interface dynamics for overlying film arrangements, with a restriction on the electric field strength. The non-local term is always linearly destabilising and produces growth rates proportional to the cube of the magnitude of the wavenumber vector. A sufficiently strong electric field is able to promote non-trivial dynamics for subcritical Reynolds number flows where the flat interface is stable in the absence of an electric field. We present numerical simulations where we observe rich dynamical behavior with competing attractors, including “snaking” travelling waves and other fully three-dimensional wave formations.

¹EPSRC studentship (RJT)

6:14PM L18.00009 On self-similar rupture of thin-film equations¹, MICHAEL DALLASTON, Complex Multiscale Systems Group, Department of Chemical Engineering, Imperial College London, DMITRI TSELUIKO, Department of Mathematical Sciences, Loughborough University, ZHONG ZHENG, Department of Mechanical and Aerospace Engineering, Princeton University, MARCO FONTELOS, Instituto de Ciencias Matematicas, SERAFIM KALLIADASIS, Complex Multiscale Systems Group, Department of Chemical Engineering, Imperial College London — Many interfacial fluid dynamical settings consist of a thin film in the presence of two physical mechanisms, one stabilizing, typically surface tension, and the other one destabilizing. Examples include the Marangoni instability of a film heated from below, Rayleigh-Taylor instability of a film on a cylinder, and film dewetting due to intermolecular forces. Lubrication-type models of these phenomena lead to very similar equations for the evolution of the film thickness, differing only in the exponent of the coefficient function of the destabilizing term. However, the behavior of solutions can vary, depending on the value of this exponent. Here we report the results of analysis based on self-similarity, elements from dynamical systems theory and fully time-dependent computations. We find that branches of self-similar rupture solutions merge at critical values of the exponent, and, surprisingly, there are no stable solutions beyond the first value at which merging occurs. In this regime, time-dependent computations indicate the existence of a cascade of instabilities of increasingly short wavelengths.

¹This work was supported by the EPSRC under grant no. EP/K008595/1. The work of DT was partly supported by the EPSRC under grant no. EP/K041134/1.

Monday, November 21, 2016 4:30PM - 6:40PM – Session L19 Bio: Flapping and Swimming III D136 - Oscar Curet, Florida Atlantic University

4:30PM L19.00001 Scaling of hydrodynamics and swimming kinematics of shelled Antarctic sea butterfly, DEEPAK ADHIKARI, DONALD WEBSTER, JEANNETTE YEN, Georgia Tech — A portable tomographic PIV system was used to study fluid dynamics and kinematics of pteropods (aquatic snails nicknamed ‘sea butterflies’) in Antarctica. These pteropods (*Limacina helicina antarctica*) swim with a pair of parapodia (or “wings”) via a unique flapping propulsion mechanism that incorporates similar techniques as observed in small flying insects. The swimming velocity is typically 14 – 30 mm/s for pteropod size ranging 1.5 – 5 mm, and the pteropod shell pitches forward-and-backward at 1.9 – 3 Hz. It has been shown that pitching motion of the shell effectively positions the parapodia such that they flap downwards during both power and recovery strokes. The non-dimensional variables characterizing the motion of swimming pteropods are flapping, translating, and pitching Reynolds numbers (i.e. Re_f , Re_U , and Re_Ω). We found that the relationship between these Reynolds numbers show an existence of a critical Re_Ω , below which pteropods fail to swim successfully. We explore the importance of this critical Re_Ω by changing the viscosity of the seawater using methylcellulose. At higher viscosity, our results indicate that pteropods do not swim with optimal propulsion efficiency. Finally, we examine the wake signature of swimming pteropod, consisting of a pair of vortex rings, in the modified viscosity environment.

4:43PM L19.00002 The Complex Hydrodynamics of Swimming in the Spanish Dancer¹, ZHUOYU ZHOU, RAJAT MITTAL, Dept. of Mechanical Engineering, Johns Hopkins Univ — The lack of a vertebra seems to have freed marine gastropods to explore and exploit a stupendous variety of swimming kinematics. In fact, examination of just a few animals in this group reveal locomotory modes ranging from insect-like flapping, to fish-like undulatory swimming, jet propulsion, and rajiform (manta-like) swimming. There are also a number of marine gastropods that have bizarre swimming gaits with no equivalent among fish or marine mammals. In this latter category is the Spanish Dancer (*Hexabranchus sanguineus*) a sea slug that swims with a complex combination of body undulations and flapping parapodia. While the neurobiology of these animals has been relatively well-studied, less is known about their propulsive mechanism and swimming energetics. In this study, we focus on the hydrodynamics of two distinct swimmers: the Spanish Dancer, and the sea hare Aplysia; the latter adopts rajiform-like mode of swimming by passing travelling waves along its parapodia. In the present study an immersed boundary method is employed to examine the vortex structures, hydrodynamic forces and energy costs of the swimming in these animals.

¹NSF Grant No. 1246317

4:56PM L19.00003 Swimming of the Honey Bees¹, CHRIS ROH, MORTEZA GHARIB, California Institute of Technology — When the weather gets hot, nursing honey bees nudge foragers to collect water for thermoregulation of their hive. While on their mission to collect water, foragers sometimes get trapped on the water surface, forced to interact with a different fluid environment. In this study, we present the survival strategy of the honey bees at the air-water interface. A high-speed videography and shadowgraph were used to record the honey bees swimming. A unique thrust mechanism through rapid vibration of their wings at 60 to 150 Hz was observed.

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

5:09PM L19.00004 Development of a Diver-Operated Single Camera Volumetric Velocimetry System, VALERIE TROUTMAN, JOHN DABIRI, Stanford University — The capabilities of a single camera, volumetric velocimetry system for in situ measurement in marine environments are demonstrated by imaging a well-characterized flow in a laboratory environment. This work represents the first stages in the design of a SCUBA-diver operated system to study organisms and biological processes under the natural light in the water column. This system is primarily composed of a volumetric particle tracking diagnostic to investigate fluid-animal interactions. A target domain size of a 20 cm sided cube is sought as a key design feature for the capability of capturing the flow around a variety of benthic and freely swimming organisms. The integration of the particle tracking system with additional diagnostics will be discussed.

5:22PM L19.00005 A study of sea lion hydrodynamics using a robotic foreflipper platform, ADITYA A. KULKARNI, RAHI K. PATEL, MEGAN C. LEFTWICH, The George Washington University — Unlike most fish and mammals—that utilize BCF swimming—sea lions rely on their foreflippers to generate thrust without a characteristic flapping frequency. This unique swimming style allows the sea lion to be highly maneuverable, while also producing high amounts of thrust. To explore this motion, and the physics that underlies it, we use novel markerless tracking techniques on untrained sea lions at the Smithsonian National Zoo in Washington, D.C to get the complete motion during different maneuvers. High speed video and three-dimensional surface reconstruction techniques are used to extract the foreflippers kinematics during the thrust phase. Using this data, pitch angle is calculated with respect to the base of the flipper to build a scaled robotic flipper. Dye visualization is carried out in a water channel by injecting dye upstream of the leading edge of the flipper with flow speed set to explore different parameters, like Reynolds number or angular velocity. Results show low pressure on the upper surface of the flipper causes the fluid to be pulled around the flipper forming a vortex that moves fully out of the plane.

5:35PM L19.00006 Exploring the neural bases of goal-directed motor behavior using fully resolved simulations¹, NAMU PATEL, NEELESH A. PATANKAR, Northwestern Univ — Undulatory swimming is an ideal problem for understanding the neural architecture for motor control and movement; a vertebrates robust morphology and adaptive locomotive gait allows the swimmer to navigate complex environments. Simple mathematical models for neurally activated muscle contractions have been incorporated into a swimmer immersed in fluid. Muscle contractions produce bending moments which determine the swimming kinematics. The neurobiology of goal-directed locomotion is explored using fast, efficient, and fully resolved constraint-based immersed boundary simulations. Hierarchical control systems tune the strength, frequency, and duty cycle for neural activation waves to produce multifarious swimming gaits or synergies. Simulation results are used to investigate why the basal ganglia and other control systems may command a particular neural pattern to accomplish a task. Using simple neural models, the effect of proprioceptive feedback on refining the body motion is demonstrated. Lastly, the ability for a learned swimmer to successfully navigate a complex environment is tested.

¹This work is supported by NSF CBET 1066575 and NSF CMMI 0941674

5:48PM L19.00007 Coarse-grained models for interacting, flapping swimmers¹, ANAND OZA, LEIF RISTROPH, MICHAEL SHELLEY, Courant Institute of Mathematical Sciences, New York University, COURANT INSTITUTE APPLIED MATH LAB COLLABORATION — We present the results of a theoretical investigation into the dynamics of interacting flapping swimmers. Our study is motivated by ongoing experiments in the NYU Applied Math Lab, in which freely-translating, heaving airfoils interact hydrodynamically to choose their relative positions and velocities. We develop a discrete dynamical system in which flapping swimmers shed point vortices during each flapping cycle, which in turn exert forces on the swimmers. We present a framework for finding exact solutions to the evolution equations and for assessing their stability, giving physical insight into the preference for certain observed "schooling states". The model may be extended to arrays of flapping swimmers, and configurations in which the swimmers' flapping frequencies are incommensurate. Generally, our results indicate how hydrodynamics may mediate schooling and flocking behavior in biological contexts.

¹A. Oza acknowledges the support of the NSF Mathematical Sciences Postdoctoral Fellowship.

6:01PM L19.00008 More efficient swimming by spreading your fingers, WILLEM VAN DE WATER, JOSJE VAN HOUWELINGEN, DENNIS WILLEMSSEN, Applied Physics, Eindhoven University of Technology, WIM PAUL BREUGEM, JERRY WESTERWEEL, RENE DELFOS, ERNST JAN GRIFF, Laboratory for Aero and Hydrodynamics, Delft University of Technology — A tantalizing question in free-style swimming is whether the stroke efficiency during the pull phase depends on spreading the fingers. It is a subtle effect—not more than a few percent—but it could make a big difference in a race. We measure the drag of arm models with increasing finger spreading in a wind tunnel and compare forces and moments to the results of immersed boundary simulations. Virtual arms were used in the simulations and their 3D-printed real versions in the experiment. We find an optimal finger spreading, accompanied by a marked increase of coherent vortex shedding. A simple actuator disk model explains this optimum.

6:14PM L19.00009 Vortices revealed: Swimming faster, JOSJE VAN HOUWELINGEN, WILLEM VAN DE WATER, RUDIE KUNNEN, GERT JAN VAN HEIJST, HERMAN CLERCX, Fluid Dynamics Laboratory, Dept. Physics, Eindhoven University of Technology, P.O. Box 513, 5600 MB, Eindhoven — Understanding and optimizing the propulsion in human swimming requires insight into the hydrodynamics of the flow around the swimmer. Experiments and simulations addressing the hydrodynamics of swimming have been conducted in studies before, including the visualization of the flow using particle image velocimetry (PIV). The main objective in this study is to develop a system to visualize the flow around a swimmer in practice inspired by this technique. The setup is placed in a regular swimming pool. The use of tracer particles and lasers to illuminate the particles is not allowed. Therefore, we choose to work with air bubbles with a diameter of ~4 mm, illuminated by ambient light. Homogeneous bubble curtains are produced by tubes implemented in the bottom of the pool. The bubble motion is captured by six cameras placed in underwater casings. A first test with the setup has been conducted by pulling a cylinder through the bubbles and performing a PIV analysis. The vorticity plots of the resulting data show the expected vortex street behind the cylinder. The shedding frequency of the vortices resembles the expected frequency. Thus, it is possible to identify and follow the coherent structures. We will discuss these results and the first flow measurements around swimmers.

6:27PM L19.00010 Bird beaks bear the brunt of bashing impact, SABERUL ISLAM SHARKER, Utah State University, SEAN HOLEKAMP, Naval Undersea Warfare Center, FRANK FISH, Westchester University, JESSE BELDEN, Naval Undersea Warfare Center, TADD TRUSCOTT, Utah State University — Seabirds can dive from 30 meters reaching speeds of 24 meters per second as they impact the water reaching depths of 9 meters due to their momentum, and a further 25 meters by active flapping. It is thought that their geometry, particularly the beak, allows them to endure relatively high impact forces that could kill non-diving birds. Acceleration data of simplified models of diving birds agree with simulated data for one species, however, no reliable experimental data with real bird geometries exist for comparison. We experimentally measured the impact accelerations of twelve 3D printed models of diving birds (seven surface diving and five plunge diving) during water-entry at different impact velocities using accelerometers.

Monday, November 21, 2016 4:30PM - 6:40PM –

Session L20 Bio: General Topics I D137-138 - Daniel Anderson, George Mason University

4:30PM L20.00001 A Reduced-dimension Model of Liquid Plug Propagation in Tubes, DAVID HALPERN, Univ of Alabama - Tuscaloosa, HIDEKI FUJIOKA, JASON RYANS, DONALD GAVER, Tulane University — We developed a reduced dimensional model of the flow resistance by the motion of a viscous plug through a liquid lined tube. This is motivated by our interest in developing large-scale models of interfacial flows in pulmonary networks. Unfortunately, full CFD calculations are not viable, so we propose a semi-empirical formula for the resistance as a function of plug length, capillary number (Ca) and precursor film thickness. We developed CFD-based empirical relationships for the resistance contributors (front and rear meniscus and the plug core). The front meniscus resistance varies with Ca and the precursor film thickness. The rear meniscus resistance increases monotonically with decreasing Ca . We use a Poiseuille model in the core region, so the resistance linearly increases with plug length. With this we estimate the max wall shear and normal stress and gradients. The results show that for fingers of air propagating through airways, the epithelial cell damage correlates with the pressure gradient. However, for shorter plugs the front meniscus may provide substantial stresses that could modulate this behavior and may influence cell injury.

4:43PM L20.00002 Modeling glycocalyx of the tear film as poroelastic layer¹, JAVED SIDDIQUE, Penn State York, ANTONIO MASTROBERARDINO, Penn State Erie, RICHARD BRAUN, University of Delaware, DANIEL ANDERSON, George Mason University — In this study we investigate a one-dimensional model for the evolution of the tear film subject to locally elevated evaporation at its anterior surface. We formulate a thin film model based on a combination of lubrication theory and mixture theory in order to understand the dynamics between the aqueous layer and the glycocalyx, which we treat as a poroelastic region. The model includes the physical effects of evaporation, surface tension, and viscosity.

¹Simon Foundation, NSF, NIH

4:56PM L20.00003 Contact lens and tear film dynamics during blinking¹, TIMOTHY REID, DANIEL ANDERSON, George Mason University — We develop a mathematical model that couples the dynamics of the tear film and contact lens during blinking. We derive an ordinary differential equation for the motion of the contact lens (parallel to the cornea) driven and retarded by viscous forces in the thin fluid films separating the contact lens from the eyelids and the corneal surface. Using the contact lens motion and tear film dynamics models we calculate a numerical solution of tear film thickness, showing that the lens and lid motion influence the tear film dynamics. The numerical solution uses a mapped Chebyshev spectral method for the spatial derivatives to reduce the model to a system of differential algebraic equations.

¹National Science Foundation grants (DMS-1407087) and (DMS-1107848)

5:09PM L20.00004 Simulation Of The Synovial Fluid In A Deformable Cavity¹, NANCY MARTINEZ-GUTIERREZ, LAURA A. IBARRA-BRACAMONTES, Universidad Michoacana DE San Nicolas DE Hidalgo — The main components of a synovial joint are a cartilage and a biofluid known as the synovial fluid. The results were obtained using the FLUENT software to simulate the behavior of the synovial fluid within a deformable cavity with a simple geometry. The cartilage is represented as a porous region. By reducing the available region for the fluid, a fluid displacement into the cartilage is induced. The total pressure reached in the interface of the deformable cavity and the porous region is presented. The geometry and properties of the system are scaled to values found in a knee joint. The effect of deformation rate, fluid viscosity and properties of the porous medium on the total pressure reached are analyzed. The higher pressures are reached either for high deformation rate or when the fluid viscosity increases.

¹This study was supported by the Mexican Council of Science and Technology (CONACyT) and by the Scientific Research Coordination of the University of Michoacan in Mexico.

5:22PM L20.00005 The circulation of the cerebrospinal fluid (CSF) in the spinal canal, ANTONIO L SANCHEZ, University of California San Diego, CARLOS MARTINEZ-BAZAN, Universidad de Jaen, JUAN C LASHERAS, University of California San Diego — Cerebrospinal Fluid (CSF) is secreted in the choroid plexus in the lateral sinuses of the brain and fills the subarachnoid space bathing the external surfaces of the brain and the spinal canal. Absence of CSF circulation has been shown to impede its physiological function that includes, among others, supplying nutrients to neuronal and glial cells and removing the waste products of cellular metabolism. Radionuclide scanning images published by Di Chiro in 1964 showed upward migration of particle tracers from the lumbar region of the spinal canal, thereby suggesting the presence of an active bulk circulation responsible for bringing fresh CSF into the spinal canal and returning a portion of it to the cranial vault. However, the existence of this slow moving bulk circulation in the spinal canal has been a subject of dispute for the last 50 years. To date, there has been no physical explanation for the mechanism responsible for the establishment of such a bulk motion. We present a perturbation analysis of the flow in an idealized model of the spinal canal and show how steady streaming could be responsible for the establishment of such a circulation. The results of this analysis are compared to flow measurements conducted on in-vitro models of the spinal canal of adult humans.

5:35PM L20.00006 Filling a Conical Cavity , KYLE NYE, AZAR ESLAM-PANAH, Penn State University —

Root canal treatment involves the removal of infected tissue inside the tooth's canal system and filling the space with a dense sealing agent to prevent further infection. A good root canal treatment happens when the canals are filled homogeneously and tightly down to the root apex. Such a tooth is able to provide valuable service for an entire lifetime. However, there are some examples of poorly performed root canals where the anterior and posterior routes are not filled completely. Small packets of air can be trapped in narrow access cavities when restoring with resin composites. Such teeth can cause trouble even after many years and lead the conditions like acute bone infection or abscesses. In this study, the filling of dead-end conical cavities with various liquids is reported. The first case studies included conical cavity models with different angles and lengths to visualize the filling process. In this investigation, the rate and completeness at which a variety of liquids fill the cavity were observed to find ideal conditions for the process. Then, a 3D printed model of the scaled representation of a molar with prepared post spaces was used to simulate the root canal treatment. The results of this study can be used to gain a better understanding of the restoration for endodontically treated teeth.

5:48PM L20.00007 Calculation of pressure drop in the developmental stages of the medaka fish heart and microvasculature , SREYASHI CHAKRABORTY, Purdue University, PAVLOS VLACHOS,

Purdue University, Professor of Mechanical Engineering — Peristaltic contraction of the developing medaka fish heart produces temporally and spatially varying pressure drop across the atrioventricular (AV) canal. Blood flowing through the tail vessels experience a slug flow across the developmental stages. We have performed a series of live imaging experiments over 14 days post fertilization (dpf) of the medaka fish egg and cross-correlated the red blood cell (RBC) pattern intensities to obtain the two-dimensional velocity fields. Subsequently we have calculated the pressure field by integrating the pressure gradient in the momentum equation. Our calculations show that the pressure drop across the AV canal increases from 0.8mm Hg during 3dpf to 2.8 mm Hg during 14dpf. We have calculated the time-varying wall shear stress for the blood vessels by assuming a spatially constant velocity magnitude in each vessel. The calculated wall shear stress matches the wall shear stress sensed by human endothelial cells (10-12 dyne/sq. cm). The pressure drop per unit length of the vessel is obtained by doing a control volume analysis of flow in the caudal arteries and veins. The current results can be extended to investigate the effect of the fluid dynamic parameters on the vascular and cardiac morphogenesis.

6:01PM L20.00008 Vascular wall shear stress in zebrafish model of early atherosclerosis¹ , WOORAK CHOI, Pohang Univ of Sci & Tech, EUNSEOK SEO, Daegu Gyeongbuk Institute of Science and Technology , EUNSEOP YEOM, School of Mechanical Engineering, Pusan National University, SANG JOON LEE, Pohang Univ of Sci & Tech —

Although atherosclerosis is a multifactorial disease, the role of hemodynamic force has strong influence on the outbreak of the disease. Low and oscillating wall shear stress (WSS) is associated with the incidence of atherosclerosis. Many researchers have investigated relationships between WSS and the occurrence of atherosclerosis using *in vitro* and *in vivo* models. However, these models possess technological limitations in mimicking real biophysiological conditions and monitoring the temporal progression of atherosclerosis. In this study, a hypercholesterolaemic zebrafish model was established as a novel model to resolve these technical limitations. WSS in blood vessels of 15 days post-fertilisation zebrafish was measured using a micro PIV technique, and the spatial distribution of lipids inside blood vessels was quantitatively visualized using a confocal microscopy. As a result, lipids are mainly deposited in the regions of low WSS. The oscillating WSS is not induced by blood flows in the zebrafish disease model. The present hypercholesterolaemic zebrafish model would be useful for understanding the effect of WSS on the early stage of atherosclerosis.

¹This work was supported by the National Research Foundation of Korea (NRF) under a grant funded by the Korean government (MSIP) (No. 2008-0061991).

6:14PM L20.00009 What optimization principle explains the zebrafish vasculature?¹

, SHYR-SHEA CHANG, KYUNG IN BAEK, TZUNG HSIAI, MARCUS ROPER, Univ of California - Los Angeles — Many multicellular organisms depend on biological transport networks; from the veins of leaves to the animal circulatory system, to redistribute nutrients internally. Since natural selection rewards efficiency, those networks are thought to minimize the cost of maintaining the flow inside. But optimizing these costs creates tradeoffs with other functions, e.g. mixing or uniform distribution of nutrients. We develop an extended Lagrange multiplier approach that allows the optimization of general network functionals. We also follow the real zebrafish vasculature and blood flows during organism development. Taken together, our work shows that the challenge of uniform oxygen perfusion, and not transport efficiency, explain zebrafish vascular organization.

¹Ruth L. Kirschstein National Research Service Award (T32-GM008185)

6:27PM L20.00010 Time-resolved transglottal pressure measurements in a scaled up vocal fold model¹ , HUNTER RINGENBERG, University of Nebraska - Lincoln, MICHAEL KRANE, Penn State - ARL, DYLAN ROGERS, MITCHEL MISFELDT, TIMOTHY WEI, University of Nebraska - Lincoln —

Experimental measurements of flow through a scaled up dynamic human vocal fold model are presented. The simplified 10x scale vocal fold model from Krane, *et al.* (2007) was used to examine fundamental features of vocal fold oscillatory motion. Of particular interest was the temporal variation of transglottal pressure multiplied by the volume flow rate through the glottis throughout an oscillation cycle. Experiments were dynamically scaled to examine a range of frequencies, 100 – 200 Hz, corresponding to the male and female voice. By using water as the working fluid, very high resolution, both spatial and temporal resolution, was achieved. Time resolved movies of flow through symmetrically oscillating vocal folds will be presented. Both individual realizations as well as phase-averaged data will be shown. Key features, such as randomness and development time of the Coanda effect, vortex shedding, and volume flow rate data have been presented in previous APS-DFD meetings. This talk will focus more on the relation between the flow and aeroacoustics associated with vocal fold oscillations.

¹Supported by the NIH

Monday, November 21, 2016 4:30PM - 6:14PM –

Session L21 Magnetohydrodynamics: Flows D139-140 - Weili Luo, University of Central Florida

4:30PM L21.00001 Flow Structure Determined Enhancing and Inhibiting Convective Heat Transfers in Quasi 1D Magnetic Fluid

, WEILI LUO, JUN HUANG, University of Central Florida, TIANSHU LIU, University of Western Michigan — We have found previously [1] that the convective flow in magnetic fluid responds to applied magnetic fields differently, depending on the relative direction of the gradient of temperature to that of the field. In this work we report the velocity profiles from these flows obtained from optical flow method. The peculiar magnetic driving force as well as the special configurations give rise to unique flow patterns, distinctly depends on the specific relative orientation of the temperature to that of field. The streamline plots indicate formation of local or global flow structures that explain the different effects of field on the heat transfer in the sample. For one configuration, the magneto-thermo convection causing the "heat" to be localized, stopping the equilibration process in the system [2]. We will discuss the different responses to the applied magnetic fields for two different sample configurations in terms of relative orientation of the temperature and field gradients. [1] Jun Huang and Weili Luo "Heat Transfer Through Convection in a Quasi-One-Dimensional Magnetic Fluid." Journal of Thermal Analysis & Calorimetry, 113, p449 (2013). [2] Jun Huang, T. Liu, and Weili Luo, preprint 2016

4:43PM L21.00002 Thin film ferromagnets acting like a compressible fluid¹

EZIO IACOCCA, Univ of Colorado - Boulder, THOMAS SILVA, National Institute of Standards and Technology, MARK HOEFER, Univ of Colorado - Boulder — Spin dynamics in ferromagnetic materials are mathematically described by the Landau-Lifshitz equation of motion. Recently, it has been shown that this equation can be exactly rewritten as a system of hydrodynamic equations [1] that are analogues of the isentropic Euler equations of compressible gas dynamics. These equations exhibit intriguing features such as a velocity-dependent pressure law and broken Galilean invariance, implying that the ferromagnets fluid-like physics are reference-frame dependent. A magnetic Mach number is defined from which subsonic and supersonic conditions are identified. By introducing finite-sized obstacles, we numerically observe laminar flow or the nucleation of ordered vortex-antivortex pairs in the subsonic regime; and the formation of a Mach cone, wavefronts, and irregular vortex-antivortex pairs in the supersonic regime. Our approach identifies a deep connection between ferromagnetism and fluid dynamics, enabling new predictions for thin film ferromagnets and opening up a new paradigm for magnetic research. References: [1] Iacocca, Silva, and Hoefer, arXiv:1606.01565 (2016)

¹E.I. acknowledges support from the Swedish Research Council, Reg. No. 637-2014-6863. M.A.H partially supported by NSF CAREER DMS-1255422.

4:56PM L21.00003 On-Off intermittency detected at the onset of turbulence in magnetized ionized gases

, THIERY PIERRE, CNRS Paris — The transition to turbulence is investigated in a rotating linear magnetized plasma column (MISTRAL device) and the role of the noise is emphasized. The destabilization is induced by the injection of electrons on the axis of the device biasing the anode of the source plasma. Starting from a rotating plasma (laminar regime), the slight increase of the potential of the source plasma leads to the onset of intermittent bursts in the edge corresponding to a subcritical (hysteretic) bifurcation and to the transient destruction of the stable rotating plasma column. The statistical analysis of the time series of the density at the onset of the intermittency is performed and the recurrence time of the turbulent bursts and distribution of the duration of the laminar phases are analyzed. At the threshold, a power law is found with critical exponent -3/2. This dynamical behavior is similar to On-off intermittency (Platt, Spiegel, Tresser, PRL 70, 279,1993) induced by Gaussian noise superimposed on the control parameter. When the control parameter is increased, the distribution evolves towards an exponential decay law.

5:09PM L21.00004 Transition of energy transfer from MHD turbulence to kinetic plasma

, YAN YANG, WILLIAM MATTHAEUS, TULASI PARASHAR, Univ of Delaware, YIPENG SHI, Peking University, Beijing, China, MINPING WAN, SHIYI CHEN, South University of Science and Technology of China, Shenzhen, Guangdong, China — The classical energy cascade scenario is of great importance in explaining the heating of corona and solar wind. One can envision that energy residing in large-scale fluctuations is transported to smaller scales where dissipation occurs and finally drives kinetic processes that absorb the energy flux and energize charged particles. Here we inquire how the cascade operates in a compressible plasma, and how the characteristics of energy transfer vary going from MHD to kinetic scales. When filtering MHD equations, we can get an apparent inertial range over which the conservative energy cascade occurs and the scale locality of energy transfer is similar to the cases of incompressible MHD turbulence. Pervasive shocks not only make a significant difference on energy cascade and magnetic amplification, but can also introduce considerable pressure dilation, a complement of viscous and ohmic dissipation that can trigger an alternative channel of the conversion between kinetic and internal energy. The procedure can also be applied to the Vlasov equation and kinetic simulation, in comparison with MHD turbulence, and is a good candidate to investigate the energy cascade process and the analogous role of the (tensor) pressure dilation in collisionless plasma.

5:22PM L21.00005 Ferrofluid patterns in Hele-Shaw cells: Exact, stable, stationary shape solutions¹

, SERGIO LIRA, Universidade Federal de Alagoas, Maceio, Brazil, JOSE MIRANDA, Universidade Federal de Pernambuco, Recife, Brazil — We investigate a quasi-two-dimensional system composed by an initially circular ferrofluid droplet surrounded by a nonmagnetic fluid of higher density. These immiscible fluids flow in a rotating Hele-Shaw cell, under the influence of an in-plane radial magnetic field. We focus on the situation in which destabilizing bulk magnetic field effects are balanced by stabilizing centrifugal forces. In this framing, we consider the interplay of capillary and magnetic normal traction effects in determining the fluid-fluid interface morphology. By employing a vortex-sheet formalism we have been able to find a family of exact stationary N -fold polygonal shape solutions for the interface. A weakly nonlinear theory is then used to verify that such exact interfacial solutions are in fact stable.

¹We thank CNPq (Brazilian Research Council) for financial support.

5:35PM L21.00006 Flow channeling in magnetohydrodynamic jets and mixing layers

, DIVYA SRI PRATURI, SHARATH GIRIMAJI, Texas AM Univ — Incompressible hydrodynamic (HD) jets and mixing layers satisfy Rayleigh criterion, and hence are naturally susceptible to Kelvin-Helmholtz (KH) instability. However, for magnetohydrodynamic (MHD) jets and mixing layers at high magnetic field strengths we show by means of linear analysis and numerical simulations that the interaction of shear with Alfvén waves ceases KH instability to exist. Linear analysis shows that the behavior of velocity perturbation along the direction of shear is crucial in the evolution of KH instability. We prove that the evolution of this velocity perturbation becomes oscillatory at high magnetic field strengths. This oscillatory behavior results in the confinement of the fluid particle displacement to a narrow region in the flow. We label this phenomenon "flow channeling" to indicate the confinement of the fluid particles without needing any physical boundaries.

5:48PM L21.00007 Energy interactions in homogeneously sheared magnetohydrodynamic flows, DIANE COLLARD, Kansas State University, DIVYA SRI PRATURI, SHARATH GIRIMAJI, Texas AM Univ — We investigate the behavior of homogeneously sheared magnetohydrodynamic (MHD) flows subject to perturbations in various directions. We perform rapid distortion theory (RDT) analysis and direct numerical simulations (DNS) to examine the interplay between magnetic, kinetic, and internal energies. For perturbation wavevectors oriented along the spanwise direction, RDT analysis shows that the magnetic and velocity fields are decoupled. In the case of streamwise wavevectors, the magnetic and velocity fields are tightly coupled. The coupling is "harmonic" in nature. DNS is then used to confirm the RDT findings. Computations of spanwise perturbations indeed exhibit behavior that is impervious to the magnetic field. Computed streamwise perturbations exhibit oscillatory evolution of kinetic and magnetic energies for low magnetic field strength. As the strength of magnetic field increases, the oscillatory behavior intensifies even as the energy magnitude decays, indicating strong stabilization.

6:01PM L21.00008 A Gas-kinetic Scheme for the Two-Fluid MHD Equations with Resistivity¹, STEVEN ANDERSON, SHARATH GIRIMAJI, Texas A&M University, EDUARDO DA SILVA, DIOGO SIEBERT, JUAN SALAZAR, Universidade Federal de Santa Catarina - Joinville — The two-fluid MHD equations are a simplified model of plasma flow wherein a mixture of two species (electrons and ions) is considered. In this model, unlike single-fluid MHD, quasi-neutrality is not enforced, Ohm's Law is not used, and the fluids are not in thermal equilibrium - thus both fluids assume their own density, velocity, and temperature. Here we present a numerical scheme to solve the two-fluid MHD equations based on an extension of the gas-kinetic method. In contrast to previous implementations of the gas-kinetic scheme for MHD, the solution of the non-equilibrium distribution function for each gas at the cell interface is extended to include the effect of the electromagnetic forces as well as the inter-species collisions (resistivity). Closure of the fluid equations with the electromagnetic fields is obtained through Maxwell's equations, and physically correct divergences are enforced via correction potentials. Maxwell's equations are integrated via a simple Lax-Friedrichs type flux-splitting. To separate integration of the source and flux terms in the governing equations we use Strang splitting. Some numerical results are presented to demonstrate accuracy of the scheme and we discuss advantages and potential applications of the scheme.

¹This research was supported by National Science Foundation grant number DGE-1252521 and the Coordenação de Aperfeiçoamento de Pessoal de Nível Superior (CAPES) of Brazil

Monday, November 21, 2016 4:30PM - 6:40PM – Session L22 Microscale Flows: Microfluidic Devices E141/142 - Ivan C. Christov, Purdue University

4:30PM L22.00001 Modeling fluid–structure interactions in shallow microchannels, TANMAY C. SHIDHORE, IVAN C. CHRISTOV, School of Mechanical Engineering, Purdue University — Rectangular microfluidic conduits with deformable walls are some of the simplest and most extensively studied microfluidic devices, primarily due to their practical design applications in a variety of fields like biology, medical diagnostics (e.g., lab-on-a-chip), nanotechnology, etc. Experimentally, these devices are found to deform into a non-rectangular cross-section due to fluid–structure interactions occurring at the channel walls. These deformations significantly affect the flow profile, which results in a non-linear relationship between the flow rate and the pressure drop, which cannot be explained by a 'generalised Poiseuille flow solution'. To this end, we perform a numerical study of these fluid–structure interactions and their effect on the flow rate and the pressure drop occurring in microfluidic conduits with a single deformable wall. The behavior of several shallow conduit systems ($\ell \gg w \gg h$) with rigid base and side walls and a soft top wall (e.g., PDMS) is simulated under laminar flow conditions using the commercial software suite ANSYS. Simulation results are compared against experimental pressure drop–flow rate data from the literature and also newly developed analytical expressions for the wall deformation, the pressure and the normalized flow rate.

4:43PM L22.00002 Evaluation of strategies for size based separation of polydisperse vesicle suspensions, KARI J. STORSLETT, SUSAN J. MULLER, University of California, Berkeley — Microfluidic devices can be used to separate suspensions of deformable particles with different intrinsic characteristics (e.g. size) with reasonable throughputs and without external labeling. Using vesicle suspensions to test microfluidic separation schemes provides insight into cell separation. Two schemes for separating vesicle suspensions by size are discussed: filtration and inertial focusing. The filter physically prevents most large vesicles from passing through. The filtrate is collected at one outlet and the larger vesicles are collected at another. This device showed good size separation between the two collected suspensions and was able to reduce the polydispersity of the collected suspensions relative to the original suspension. The inertial separation device was based on a design studied by Di Carlo et al¹. This design was modified for our suspension and showed an ability to separate the suspension by size; however, the separated suspension's polydispersity was only slightly reduced. The advantage of the inertial separation device was its greatly increased throughput. A separation strategy may be selected based on the relative importance of high throughput vs. reduced polydispersity. (1) Di Carlo, D. et al. *Anal. Chem.* 2008, 80, 2204-2211.

4:56PM L22.00003 Continuous high throughput molecular adhesion based cell sorting using ridged microchannels., BUSHRA TASADDUQ, GONGHAO WANG, ALEXANDER ALEXEEV, ALI FATIH SARIOGLU, TODD SULCHEK, Georgia Institute of Technology — Cell molecular interactions govern important physiological processes such as stem cell homing, inflammation and cancer metastasis. But due to a lack of effective separation technologies selective to these interactions it is challenging to specifically sort cells. Other label free separation techniques based on size, stiffness and shape do not provide enough specificity to cell type, and correlation to clinical condition. We propose a novel microfluidic device capable of high throughput molecule dependent separation of cells by flowing them through a microchannel decorated with molecule specific coated ridges. The unique aspect of this sorting design is the use of optimized gap size which is small enough to lightly squeeze the cells while flowing under the ridged part of the channel to increase the surface area for interaction between the ligand on cell surface and coated receptor molecule but large enough so that biomechanical markers, stiffness and viscoelasticity, do not dominate the cell separation mechanism. We are able to separate Jurkat cells based on its expression of PSGL-1ligand using ridged channel coated with P selectin at a flow rate of 0.045ml/min and achieve 2-fold and 5-fold enrichment of PSGL-1 positive and negative Jurkat cells respectively.

5:09PM L22.00004 Programmed assembly of colloidal arrays using shaped microvortices

AVANISH MISHRA, Department of Mechanical Engineering, Purdue University, West Lafayette, ALOKE KUMAR, Department of Mechanical Engineering, University of Alberta, Edmonton, STEVEN WERELEY, Department of Mechanical Engineering, Purdue University, West Lafayette — Ability to program colloidal assemblies in desired spatial patterns and orientation remains a significant roadblock to the development of micro- and nanoparticle-based devices. In this work, by shaping electrothermal microvortices, we demonstrate a high-throughput assembly of particles in complex shapes. The microscale electrothermal vortices are generated by optical heating of an electrode layer in the presence of an AC electric field. Entrained in the electrothermal flow particles rapidly and reversibly assemble into the shapes of projected optical patterns. These microvortices can be dynamically reconfigured by changing the optical patterns, thus allowing us to alter the topology of particle clusters. Additionally, driven by an interplay of Stokes drag and dipole-dipole repulsion, the number of particles and inter-particle spacing in an array can also be dynamically tuned by changing the flow velocity. Using a net electrophoretic force in an asymmetrical AC electric field, we demonstrate permanent deposition of patterned particles. In this presentation, we plan to discuss the mathematical background on shaping the electrothermal flow, its implementations in forming colloidal arrays, and their applications.

5:22PM L22.00005 Using machine vision and data mining techniques to identify cell properties via microfluidic flow analysis¹

GEOFFREY HOROWITZ, SAMUEL BOWIE, ANNA LIU, NICHOLAS STONE, TODD SULCHEK, ALEXANDER ALEXEEV, Georgia Institute of Technology — In order to quickly identify the wide range of mechanistic properties that are seen in cell populations, a coupled machine vision and data mining analysis is developed to examine high speed videos of cells flowing through a microfluidic device. The microfluidic device contains a microchannel decorated with a periodical array of diagonal ridges. The ridges compress flowing cells that results in complex cell trajectory and induces cell cross-channel drift, both depend on the cell intrinsic mechanical properties that can be used to characterize specific cell lines. Thus, the cell trajectory analysis can yield a parameter set that can serve as a unique identifier of a cell's membership to a specific cell population. By using the correlations between the cell populations and measured cell trajectories in the ridged microchannel, mechanical properties of individual cells and their specific populations can be identified via only information captured using video analysis.

¹Financial support provided by National Science Foundation (NSF) Grant No. CMMI 1538161

5:35PM L22.00006 A Multi-Gradient Generator in a Single Microfluidic Device for Optical Microscopy and Interferometry¹

MANUEL BEDROSSIAN, JAY NADEAU, Caltech, CHRIS LINDENSMITH, Caltech/JPL — The goal of this work was to create a single microfluidic device capable of establishing multiple types of gradients in a quantifiable manner. Many microbial species are known to exhibit directed motility in the presence of stimuli. This phenomenon, known as taxis, can be used as a bio-signature and a means of identifying microorganisms. Directed microbial motility has been seen as a response to the presence of certain chemicals, light, heat, magnetic fields, and other stimuli. Microbial movement along the gradient vector, that cannot be explained by passive hydrodynamics or Brownian motion, can shed light on whether the sample contains living microbes or not. The ability to create multiple types of gradients in a single microfluidic device allows for high throughput testing of heterogeneous samples to detect taxis. There has been increased interest in the search for life within our solar system where liquid water is known to exist. Induced directional motility can serve as a viable method for detecting *living* organisms that actively respond to their environment. The device developed here includes a chemical, photonic, thermal, and magnetic gradient generator, while maintaining high optical quality in order to be used for microscopy as well as quantitative phase imaging

¹This work was funded by the Gordon and Betty Moore Foundation, who the authors wish to thank for their generosity

5:48PM L22.00007 Heat transfer enhancement in a cross-slot micro-geometry¹

ALLYSSON DOMINGUES, WALEED ABED, ROBERT POOLE, DAVID DENNIS, University of Liverpool — The cross-slot geometry is a common geometric shape in microfluidic applications. In this work we investigate, numerically and experimentally, the influence of a purely-inertial flow instability on the enhancement of heat transfer in a cross-slot micro-geometry where symmetry is broken but the flow remains steady. The cross-slot comprises two crossed square channels with opposed inlets and outlets, which generate a stagnation point at the geometric centre. The flow of a Newtonian fluid is steady, two-dimensional and produces a sharp symmetric boundary between fluid streams entering the cross-slot from opposite directions at low Reynolds numbers (Re). Therefore, only conduction heat transfer occurs between the fluid streams as there is virtually no mixing between them. Beyond a certain critical value of Re , approximately 40, a steady symmetry-breaking bifurcation occurs and convective heat transfer arises because an axially oriented spiral vortex is created in the outlet arms. The effects of this purely-inertial instability suggest it is an effective method of enhancing mixing and heat transfer in microfluidic devices that can be exploited in applications such as lab-on-chip and micro chemical-reaction devices at relatively low Reynolds numbers (i.e. $Re < 100$).

¹Work supported by CNPq grant 203195/2014-0

6:01PM L22.00008 A scalable micro-mixer for biomedical applications

LUCA CORTELEZZI, SIMONE FERRARI, ANGELO DUBINI, Politecnico di Milano — Our study presents a geometrically scalable active micro-mixer suitable for biomedical/bioengineering applications and potentially assimilable in a Lab-on-Chip. We designed our micro-mixer with the goal of satisfying the following constraints: small dimensions, because the device must be able to process volumes of fluid in the range of $10^{-6} \div 10^{-9}$ liters; high mixing speed, because mixing should be obtained in the shortest possible time; constructive simplicity, to facilitate realizability, assimilability and reusability of the micro-mixer; and geometrical scalability, because the micro-mixer should be assimilable to microfluidic systems of different dimensions. We studied numerically the mixing performance of our micro-mixer both in two- and three-dimensions. We characterize the mixing performance in terms of Reynolds, Strouhal and Péclét numbers in order to establish a practical range of operating conditions for our micro-mixer. Finally, we show that our micro-mixer is geometrically scalable, i.e., micro-mixers of different geometrical dimensions having the same nondimensional specifications produce nearly the same mixing performance.

6:14PM L22.00009 Electromagnetically driven liquid iris.¹

DEASUNG JANG, JIN WON JEONG, DAE YOUNG LEE, DAE GEUN KIM, SANG KUG CHUNG, Department of mechanical engineering, Myongji University — This paper describes a tunable liquid iris driven by electromagnetic actuation for miniature cameras. To examine the magnetic effect on a ferrofluid, the contact angle modification of a sessile ferrofluid droplet is tested using a neodymium magnet and an electric coil which 2.5 A current is applied to. The contact angle variations of the ferrofluid droplet for each test are 21.3 and 18.1 degrees, respectively. As a proof of concept, a pretest of a tunable iris actuated by electromagnetic effect is performed by using a hollow cylinder cell. As applying the current, the aperture diameter is adjusted from 4.06 mm at 0A to 3.21 mm at 2.0A. Finally, a tunable liquid iris (9 x 9 x 2 mm³), consisting of two connected circular microchannels, is realized using MEMS technology. the aperture diameter of the tunable liquid iris is able to be modified from 1.72 mm at 0 A to 1.15 mm at 2.6 A. This tunable optical iris has potential applications not only for portable electronic devices but also in biomedical fields such as optical coherence tomography and microsurgery.

¹This work was supported by 2016 Research Fund of MyongjiUniversity.

6:27PM L22.00010 Microfluidic particle manipulation using high frequency surface acoustic waves¹, YE AI, DAVID J. COLLINS, Singapore University of Technology and Design — Precise manipulation of particles and biological cells remains a very active research area in microfluidics. Among various force fields applied for microfluidic manipulations, acoustic waves have superior propagating properties in solids and fluids, which can readily enable non-contact cell manipulation in long operating distances. Exploiting acoustic waves for fluid and cell manipulation in microfluidics has led to a newly emerging research area, acoustofluidics. In this work, I will present particle and cell manipulation in microfluidics using high frequency surface acoustic waves (SAW). In particular, I will discuss a unique design of a focused IDT (FIDT) structure, which is able to generate a highly localized SAW field on the order of 20 m wide. This highly focused acoustic beam has an effective manipulation area size that is comparable to individual micron-sized particles. Here, I demonstrate the use of this highly localized SAW field for single particle level sorting with sub-millisecond pulses and selective capture of particles. Based on the presented studies on acoustic particle manipulation, I envision that the merging of acoustics and microfluidics could enable various particle and cell manipulations needed in microfluidic applications.

¹We acknowledge the support received from Singapore University of Technology and Design (SUTD)-Massachusetts Institute of Technology (MIT) International Design Center (IDG11300101) and SUTD Startup Research Grant (SREP13053) awarded to Y.A.

Monday, November 21, 2016 4:30PM - 6:27PM –
Session L25 Microscale Flows: Computations E145 - Francesco Costanzo, Pennsylvania State University

4:30PM L25.00001 Arbitrary Lagrangian-Eulerian (ALE) formulation for microacoustofluidics¹, NITESH NAMA, TONY JUN HUANG, FRANCESCO COSTANZO, Pennsylvania State University — We present an Arbitrary Lagrangian-Eulerian (ALE) formulation for the analysis of acoustic streaming flows. We employ a multiscale approach to separate the flow variables into first-and second-order components which results in two subproblems: a first-order problem, formulated in terms of the fluid displacement at a fast scale and a second-order problem formulated in terms of the Lagrangian flow velocity at a slow time scale. The Lagrangian velocity based formulation of the second-order problem removes the ambiguity concerning the second-order boundary condition at the oscillating walls and circumvents the need to employ the notion of Stokes drift, thereby allowing a direct comparison with the experiments. Moreover, the ALE formulation offers a natural extension to the more complex fluid-structure interaction problems in microacoustofluidic devices. Lastly, we present numerical test cases where the Eulerian flow velocities exhibit several non-physical features that are not observed in the corresponding Lagrangian flow velocities, indicating that a Lagrangian velocity based formulation is much more favorable and readily interpretable.

¹This work was supported by the National Science Foundation (CBET-1438126 and IIP-1346440).

4:43PM L25.00002 Autonomously Responsive Pumping by a Bacterial Flagellar Forest: A Mean-field Approach, JAMES MARTINDALE, HENRY C. FU, Univ of Utah — The design and fabrication of microscale pumps using magnetically actuated bacterial flagella opens the door for many applications such as the pumping and regulation of chemicals. Here, we discuss simulations for a pump consisting of a regular two-dimensional array of rigid helices. Recent work investigating the flows above a small, finite array by numerically calculating the full dynamics showed that having random phase differences between helices seems essential to produce the flow patterns observed in experiments. We developed a model which allows us to treat random phase differences in an infinite array. Using a mean-field approach we define pumping as the existence of a self-consistent tilt angle of the array. Pumping is then examined numerically as a function of several parameters in the magnetic actuation and helical geometry. We demonstrate how this pumping flow may be mechanically halted by way of magnetic actuation or autonomously halted by the polymorphic transformation of bacterial flagella in response to environmental stimuli.

4:56PM L25.00003 Directed swimming of nanoscale swimmers in an array of posts with non-circular section: modelling and shape optimization¹, JIAJUN TONG, MICHAEL SHELLEY, New York University — It has been shown experimentally that swimming of nanoscale rod-like bi-metallic swimmers can be biased and guided by an array of teardrop shaped posts in the solution, giving rise to a statistically directed motion in long time. This could be useful in many applications like concentrating nanoswimmers, or separating them from non-motile particles. We pose a model to study such directed swimming, taking into account the absorption and desorption of the swimmers to the vertical walls of posts. We emphasize the role of varying curvature along the circumference of a single post on the absorption and desorption. In seeking to enhance directed swimming, we apply shape optimization to find how we can design, based on experimental data, better posts which have higher efficiency of transporting swimmers.

¹This work was partially supported by the National Science Foundation under Award Number DMS-1463962.

5:09PM L25.00004 Tailoring tails in Taylor dispersion: how boundaries shape chemical delivery in microfluidics: computation and theory¹, RICHARD M. MC LAUGHLIN, MANUCHEHR AMINIAN, FRANCESCA BERNARDI, ROBERTO CAMASSA, DANIEL M. HARRIS, UNC Chapel Hill, UNC JOINT FLUIDS LAB TEAM — We present the results of a combined computational and theoretical study of the dispersion of a passive scalar in laminar shear flow through rectangular and elliptical channels. We show through Monte Carlo simulation and asymptotic analysis that the cross-sectional aspect ratio sets the sign of the average skewness at long times (relative to the Taylor diffusion timescale) which describes the longitudinal asymmetry of the tracer distribution. Universally, thin channels (aspect ratio $\ll 1$) result in negative average skewness, whereas as thick channels (aspect ratio ~ 1) result in positive average skewness. Our analysis also allows us to define a “golden” aspect ratio which separates thin from thick channels, the value of which is remarkably similar for both the rectangle and the ellipse. Further, by examining the median of the cross-sectionally averaged distribution, we establish that negative skewness correlates with solutes arriving with sharp fronts followed by a tapering tail. Future directions will be discussed.

¹Funding from NSF grant Nos.: RTG DMS-0943851, CMG ARC-1025523, DMS-1009750, and DMS-1517879.

5:22PM L25.00005 Dynamic density functional theory for nucleation: Non-classical predictions of mesoscopic nucleation theory¹, MIGUEL A. DURAN-OLIVENCIA, PETER YATSYSHIN, Imperial College London, JAMES F. LUTSKO, Université Libre de Bruxelles, SERAFIM KALLIADASIS, Imperial College London — Classical density functional theory (DFT) for fluids and its dynamic extension (DDFT) provide an appealing mean-field framework for describing equilibrium and dynamics of complex soft matter systems. For a long time, homogeneous nucleation was considered to be outside the limits of applicability of DDFT. However, our recently developed mesoscopic nucleation theory (MeNT) based on fluctuating hydrodynamics, reconciles the inherent randomness of the nucleation process with the deterministic nature of DDFT. It turns out that in the weak-noise limit, the most likely path (MLP) for nucleation to occur is determined by the DDFT equations. We present computations of MLPs for homogeneous and heterogeneous nucleation in colloidal suspensions. For homogeneous nucleation, the MLP obtained is in excellent agreement with the reduced order-parameter description of MeNT, which predicts a multistage nucleation pathway. For heterogeneous nucleation, the presence of impurities in the fluid affects the MLP, but remarkably, the overall qualitative picture of homogeneous nucleation persists. Finally, we highlight the use of DDFT as a simulation tool, which is especially appealing as there are no known applications of MeNT to heterogeneous nucleation.

¹We acknowledge financial support from the European Research Council via Advanced Grant No. 247031 and from EPSRC via grants No. EP/L020564 and EP/L025159.

5:35PM L25.00006 Steady film flow over 2D topography with air inclusion formed inside the trench.¹, JOHN TSAMOPOULOS, STYLIANOS VARCHANIS, YANNIS DIMAKOPOULOS, University of Patras — Liquid film flow along an inclined, solid substrate featuring periodic rectangular trenches may either completely wet the trench floor (Wenzel state) or pin on the entrance and exit corners of the trench (Cassie state) or assume any other configuration in between these two extremes. In the intermediate cases a second gas-liquid interface inside the trench is formed, which adheres to the walls of the trench forming two three-phase contact lines, and encloses a different amount of air under different physical conditions. The Galerkin finite element method is used to solve the Navier-Stokes equations in a physical domain, which is adaptively re-meshed. Multiple steady solutions, connected by turning points and transcritical bifurcations as well as isolated solution branches, are revealed by pseudo arc-length continuation. Two possible cases of a single air inclusion inside the trench are examined. The penetration of the liquid inside the trench is enhanced primarily by increasing either the wettability of the substrate or the capillarity or by decreasing the flow rate. Flow hysteresis may occur when the liquid does not penetrate deep enough inside the trench leading to different flow patterns. The interplay of inertia, viscous, gravity and capillary forces along with substrate wettability determines the volume of the air encapsulated in the trench and the extent of free surface deformation.

¹GSRT of Greece via the program "Excellence" and the LIMMAT foundation

5:48PM L25.00007 An ALE Finite Element Approach for Two-Phase Flow with Phase Change, ERIK GROS, Ecole Polytech Fed de Lausanne, GUSTAVO ANJOS, State University of Rio de Janeiro, JOHN THOME, Ecole Polytech Fed de Lausanne, LTCM TEAM, GESAR TEAM — In this work, two-phase flow with phase change is investigated through the Finite Element Method (FEM) in the Arbitrary Lagrangian-Eulerian (ALE) framework. The equations are discretized on an unstructured mesh where the interface between the phases is explicitly defined as a sub-set of the mesh. The two-phase interface position is described by a set of interconnected nodes which ensures a sharp representation of the boundary, including the role of the surface tension. The methodology proposed for computing the curvature leads to very accurate results with moderate programming effort and computational costs. Such a methodology can be employed to study accurately many two-phase flow and heat transfer problems in industry such as oil extraction and refinement, design of refrigeration systems, modelling of microfluidic and biological systems and efficient cooling of electronics for computational purposes. The latter is the principal aim of the present research. The numerical results are discussed and compared to analytical solutions and reference results, thereby revealing the capability of the proposed methodology as a platform for the study of two-phase flow with phase change.

6:01PM L25.00008 Three dimensional simulations of viscous folding in diverging microchannels, BINGRUI XU, Department of Aeronautics and Astronautics, Fudan Univ, JALEL CHERGUI, LIMSI, Univ. Paris-Sud, CNRS, Université Paris-Saclay, SEUNGWON SHIN, Department of Mechanical and System Design Engineering, Hongik University, DAMIR JURIC, LIMSI, Univ. Paris-Sud, CNRS, Université Paris-Saclay — Three dimensional simulations on the viscous folding in diverging microchannels reported by Cubaud and Mason are performed using the parallel code BLUE for multi-phase flows. The more viscous liquid L_1 is injected into the channel from the center inlet, and the less viscous liquid L_2 from two side inlets. Liquid L_1 takes the form of a thin filament due to hydrodynamic focusing in the long channel that leads to the diverging region. The thread then becomes unstable to a folding instability, due to the longitudinal compressive stress applied to it by the diverging flow of liquid L_2 . We performed a parameter study in which the flow rate ratio, the viscosity ratio, the Reynolds number, and the shape of the channel were varied relative to a reference model. In our simulations, the cross section of the thread produced by focusing is elliptical rather than circular. The initial folding axis can be either parallel or perpendicular to the narrow dimension of the chamber. In the former case, the folding slowly transforms via twisting to perpendicular folding, or it may remain parallel. The direction of folding onset is determined by the velocity profile and the elliptical shape of the thread cross section in the channel that feeds the diverging part of the cell.

6:14PM L25.00009 Mathematical modelling of Liquid –Liquid extraction in the slug flow regime in a microchannel, SUNDARI RAMJI, DINESH BHAGAVATULA, ARJUN RAKESH, PUSHPAVANAM S, None — Mixing in the slug flow regime in microchannels is enhanced by the presence of internal circulations induced by shear due to wall. This helps improve mass transfer in this flow regime. We exploit the low Re characteristic of the flow and seek a numerical solution to understand the structure of the vortex patterns formed in the two phases in the slug flow regime. We study liquid-liquid extraction in the system to determine the improvement in mass transfer. The system was analyzed for two cases when there is (i) no film surrounding the slug (ii) a thin film surrounding the slug. The 2D governing equations for fluid flow are solved using two approaches: a) a stream function formulation based on finite differences b) primitive variable formulation with the Chebyshev collocation method. The effect of viscosity ratio, slug length and film thickness on the vortex structure were studied. While secondary vortices were induced in the less viscous phase in the case where the thin film is absent, they are always generated in the slug irrespective of the viscosity ratio in the case where the film is present. The species balance equation was then solved numerically using two approaches: a) an Alternating Direction Explicit method and b) the Locally One Dimensional splitting technique. The effect of varying Peclet number from 0 to 10^4 on the solute transfer from the slug to the continuous phase was studied. The extraction performance is analyzed in terms of extraction efficiency and mass transfer coefficient.

Monday, November 21, 2016 4:30PM - 6:40PM –
Session L26 Suspensions: Rheology E146 - Roseanna Zia, Assistant Professor, Chemical and Biomolecular Eng. Cornell Univ.

4:30PM L26.00001 Gravitational collapse of colloidal gels: Origins of the tipping point¹, ROSEANNA ZIA, POORNIMA PADMANABHAN, Cornell University — Reversible colloidal gels are soft viscoelastic solids in which durable but reversible bonds permit on-demand transition from solidlike to liquidlike behavior; these $O(kT)$ bonds also lead to ongoing coarsening and age stiffening, making their rheology inherently time dependent. To wit, such gels may remain stable for an extended time, but then suddenly collapse, sedimenting to the bottom of the container (or creaming to the top) and eliminating any intended functionality of the material. Although this phenomenon has been studied extensively in the experimental literature, the microscopic mechanism underlying the collapse is not well understood. Effects of gel age, interparticle attraction strength, and wall effects all have been shown to affect collapse behavior, but the microstructural transformations underlying the tipping point remain murky. To study this behavior, we conduct large-scale dynamic simulation to model the structural and rheological evolution of colloidal gels subjected to various gravitational stresses, examining the detailed micromechanics in three temporal regimes: slow sedimentation prior to collapse; the tipping point leading to the onset of rapid collapse; and the subsequent compaction of the material as it approaches its final bed height.

¹Acknowledgment for funding and support from the Office of Naval Research; the National Science Foundation; and NSF XSEDE.

4:43PM L26.00002 Microfluidic rheology of active particle suspensions: Kinetic theory, ROBERTO ALONSO-MATILLA, BARATH EZHILAN, DAVID SAINTILLAN, UC San Diego — We analyze the effective rheology of a dilute suspension of self-propelled slender particles between two infinite parallel plates in a pressure-driven flow. We use a continuum kinetic model to study the dynamics and transport of particles, where hydrodynamic interactions induced by the swimmers are taken into account. Using finite volume simulations we study how the activity of the swimmer and the external flow modify the rheological properties of the system. Results indicate that at low flow rates, activity decreases the value of the viscosity for pushers and increases its value for pullers. Both effects become weaker with increasing the flow strength due to the alignment of the particles with the flow. In the case of puller particles, shear thinning is observed over the entire range of flow rates. Pusher particles exhibit shear thickening at intermediate flow rates, where passive stresses start dominating over active stresses, reaching a viscosity greater than that of the Newtonian fluid. Finally shear thinning is observed at high flow rates. Both pushers and pullers exhibit a Newtonian plateau at very high flow rates. We demonstrate a good agreement between numerical results and experiments.

4:56PM L26.00003 3D Numerical Study of the Shear Rheology of a Semi-dilute Viscoelastic Suspension, MENGFEI YANG, SREENATH KRISHNAN, ERIC SHAQFEH, Stanford University — The stress in suspensions of rigid particles in polymer solutions is of considerable interest in applications such as manufacturing processes and fracturing technologies. Deriving an analytic expression for the material functions of a viscoelastic suspension under shear is difficult due to the nonlinear particle-fluid and particle-particle interactions, and theoretical studies have been limited to dilute suspensions at low shear Weissenberg number (Wi) or low polymer concentrations. Previously, we performed 3D single-particle simulations and showed that the results agreed well with the existing theories in the appropriate parameter regimes. We found that suspensions in constant-viscosity elastic fluids shear-thicken over a range of Wi and their material properties plateau at higher Wi . However, discrepancies between simulation and existing experimental measurements for volume fractions as low as 2.5% suggested that interparticle hydrodynamic interactions could not be neglected. We now present 3D high fidelity numerical simulations of multiple spheres freely suspended in a sheared viscoelastic fluid using an immersed boundary framework to study the relationship between hydrodynamic interactions, particle structure formation, and the bulk rheology of viscoelastic suspensions. We observe that in a non-shear thinning elastic fluid, particles do not “chain”, but their interactions induce additional polymer stresses in the fluid which contribute to a stronger particle effect than predicted in the dilute limit.

5:09PM L26.00004 Tunable shear thickening: from understanding suspension thickening to controlling viscosity on the fly, ITAI COHEN, NEIL LIN, Cornell University, CHRIS NESS, JIN SUN, University of Edinburgh, MIKE CATES, University of Cambridge, BEN GUY, MICHAEL HERMES, WILSON POON, University of Edinburgh — Whether contact forces play a role in shear thickening of 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. More importantly, this finding directly suggests a strategy for active controlling the thickening viscosities of dense suspensions. We demonstrate that by strategic imposition of a high-frequency and low-amplitude shear perturbation orthogonal to the primary shearing flow, we can largely eradicate thickening. The orthogonal shear effectively becomes a regulator for controlling thickening in the suspension, allowing the viscosity to be reduced by up to two decades on demand.

5:22PM L26.00005 Yield and flow-induced phase transition in colloidal gels under startup shear, LILIAN JOHNSON, BENJAMIN LANDRUM, ROSEANNA ZIA, Robert Frederick Smith School of Chemical and Biomolecular Engineering, Cornell University — We study the micro-mechanical origins of the transition from solid-like to liquid-like behavior during flow startup of colloidal gels via large-scale dynamic simulation, with a view toward understanding connections to energy storage and phase transition. Such materials often exhibit an overshoot in stress, and prior studies of strong, dilute colloidal gels with a stringy microstructure connect this “yield” event to loss of network connectivity. Owing to the importance of Brownian transport in phase separation processes in colloids, here we study a reversible colloidal gel of hard spheres that interact via a short-range attraction of several kT , for which Brownian motion can lead to rapid quiescent coarsening. In the present study, we interrogate the shear stress for a range of imposed flow strengths, monitoring particle-level structure and dynamics, to determine the microscopic picture of gel yield. Our detailed studies of the microstructural evolution and macroscopic response during startup provide insight into the phase behavior during yield. We present a new model of stress development, phase transition, and structural evolution during transient yield in colloidal gels for which ongoing phase separation informs gel phenomenology.

5:35PM L26.00006 Rheological properties of Cubic colloidal suspensions., ARMAN BOROMAND, JOAO MAIA, Case Western Reserve University — Colloidal and non-colloidal suspensions are ubiquitous in many industrial application. There are numerous studies on these systems to understand and relate their complex rheological properties to their microstructural evolution under deformation. Although most of the experimental and simulation studies are centered on spherical particles, in most of the industrial applications the geometry of the colloidal particles deviate from the simple hard sphere and more complex geometries exist. Recent advances in microfabrication paved the way to fabricate colloidal particles with complex geometries for applications in different areas such as drug delivery where the fundamental understanding of their dynamics has remained unexplored. In this study, using dissipative particle dynamics, we investigate the rheological properties of cubic (superball) particles which are modeled as the cluster of core-modified DPD particles. Explicit representation of solvent particles in the DPD scheme will conserve the full hydrodynamic interactions between colloidal particles. Rheological properties of these cubic suspensions are investigated in the dilute and semi-dilute regimes. The Einstein and Huggins coefficients for these particles with different superball exponent will be calculate which represent the effect of single particle's geometry and multibody interactions on viscosity, respectively. The response of these suspensions is investigated under simple shear and oscillatory shear where it is shown that under oscillation these particles tend to form crystalline structure giving rise to stronger shear-thinning behavior recently measured experimentally.

5:48PM L26.00007 Enhancing Shear Thickening¹, FATEMEH MADRAKI, SARAH HORMOZI, Department of Mechanical Engineering, Ohio University, Athens, Ohio 45701-2979, USA, GUILLAUME OVARLEZ, University of Bordeaux, CNRS, Solvay, LOF, UMR 5258, 33608 Pessac, France, ELISABETH GUAZZELLI, OLIVIER POULIQUEN, Aix-Marseille Universite, CNRS, IUSTI UMR 7343, 13453 Marseille, France — A cornstarch suspension is the quintessential particulate system that exhibits shear thickening. By adding large non-Brownian spheres to a cornstarch suspension, we show that shear thickening can be significantly enhanced. More precisely, the shear thickening transition is found to be increasingly shifted to lower critical shear rates. This enhancement is found to be mainly controlled by the concentration of the large particles.

¹ANR(ANR-13-IS09-0005-01), ANR(ANR-11-LABX-0092), MIDEX (ANR-11-IDEX-0001-02), NSF (CBET-1554044-CAREER)

6:01PM L26.00008 Mathematical inference in one point microrheology¹, CHRISTEL HO-HENEGGER, University of Utah, SCOTT MCKINLEY, Tulane University — Pioneered by the work of Mason and Weitz, one point passive microrheology has been successfully applied to obtaining estimates of the loss and storage modulus of viscoelastic fluids when the mean-square displacement obeys a local power law. Using numerical simulations of a fluctuating viscoelastic fluid model, we study the problem of recovering the mechanical parameters of the fluid's memory kernel using statistical inference like mean-square displacements and increment auto-correlation functions. Seeking a better understanding of the influence of the assumptions made in the inversion process, we mathematically quantify the uncertainty in traditional one point microrheology for simulated data and demonstrate that a large family of memory kernels yields the same statistical signature. We consider both simulated data obtained from a full viscoelastic fluid simulation of the unsteady Stokes equations with fluctuations and from a Generalized Langevin Equation of the particle's motion described by the same memory kernel. From the theory of inverse problems, we propose an alternative method that can be used to recover information about the loss and storage modulus and discuss its limitations and uncertainties.

¹NSF-DMS 1412998

6:14PM L26.00009 Motion of a Spherical Particle Near Porous Boundaries, PHANI KANTH SANAGAVARAPU, PRABHU NOTT, Department of Chemical Engineering, Indian Institute of Science — The flow of suspensions near porous boundaries occur in many industrial processes and living systems. The questions that arise are, how do the permeability of the porous walls and the concomitant velocity slip affect the dynamics of individual particles, and the rheology of the suspension? Here we consider the motion of a single sphere near a plane permeable slab. We solve the Stokes equations using an eigenfunction expansion in the bi-spherical coordinate system. The boundary conditions at the porous wall are Darcys flux condition normal to the surface and Saffmans slip condition in the tangential direction. In addition, we impose the condition of zero net flux across the porous slab and obtain the non-zero pressure on the other side of the porous slab. The drag and torque on the particle moving near a porous wall are computed and compared with those obtained for an impermeable wall. We comment on the implications of our results on recent measurements of the suspension stress near porous boundaries.

6:27PM L26.00010 Normal stress differences in a sheared gas-solid suspension, SAIKAT SAHA, MEHEBOOB ALAM, Jawaharlal Nehru Centre, Jakkur PO, Bangalore 560064 — The stress tensor and normal stress differences are analyzed for a homogeneously sheared gas-solid suspension using Enskog-Boltzmann equation. Inelastic particles are suspended in a viscous fluid of viscosity μ_f and experience a Stokes drag force. Viscous heating due to shear is compensated by (i) the inelastic collisions between particles and (ii) the drag force experienced by the particles due to the interstitial fluid. Rheology of the particle phase is analyzed with anisotropic-Gaussian as the single particle distribution function. The first (\mathcal{N}_1) and second (\mathcal{N}_2) normal stress differences are computed as functions of the density (ν), Stokes number (St) and restitution coefficient (e). A comparison with the existing simulation data shows an excellent agreement for both \mathcal{N}_1 and \mathcal{N}_2 over the predictions from other Grad-level theories. Finally, in the limit of $St \rightarrow \infty$ ($\mu_f \rightarrow 0$), the related results from the conventional theory of dry granular flows are recovered.

Monday, November 21, 2016 4:30PM - 6:27PM –
Session L27 Non-Newtonian Flows: Applications E147-148 - Federico Toschi, Technische Universiteit Eindhoven

4:30PM L27.00001 Dynamics and morphology of flexible membranes in a nematic liquid crystal, ARTHUR EVANS, MICHAEL GRAHAM, SAVERIO SPAGNOLI, Univ of Wisconsin, Madison — Cellular membranes and nanoscale capsules inhabit highly viscous environments, and their overall dynamics and morphology depend on the fluid in which they are immersed. While some fluids of interest are purely Newtonian, biologically and industrially relevant materials often involve complex molecular constituents that imbue the surroundings with anisotropy or viscoelasticity. Moreover, fluid stresses are often comparable to the elastic resistance of immersed membranes or capsules, so that these structures deform in response to flow and order in the fluid. In this talk I will present a numerical implementation for examining immersed elastic bodies in a flowing liquid crystal, and discuss the implications for biological materials and self-assembly of soft structures. Even in the case of zero flow for rigid particles, the presence of topological defects in nematics leads to intriguing inter-particle interactions. For flexible membranes, the liquid crystal defect structure not only affects long-range interactions, but ultimately the morphology of the membranes themselves.

4:43PM L27.00002 Inferring earthquake statistics from soft-glass dynamics below yield stress¹, PINAKI KUMAR, FEDERICO TOSCHI, Eindhoven University of Technology, ROBERTO BENZI, University of Rome Tor Vergata, JEANNOT TRAMPERT, Utrecht University — The current practice to generate synthetic earthquake catalogs employs purely statistical models, mechanical methods based on ad-hoc constitutive friction laws or a combination of the above. We adopt a new numerical approach based on the multi-component Lattice Boltzmann method to simulate yield stress materials. Below yield stress, under shear forcing, we find that the highly intermittent in time, irreversible T1 topological changes in the soft-glass (termed plastic events) bear a statistical resemblance to seismic events, radiating elastic perturbations through the system. Statistical analysis reveals scaling laws for magnitude similar to the Gutenberg-Richter law for quakes, a recurrence time scale with similar slope, a well-defined clustering of events into causal-aftershock sequences and Poisson events leading to the Omori law. Additionally space intermittency reveals a complex multi-fractal structure, like real quakes, and a characterization of the stick-slip behavior in terms of avalanche size and time distribution agrees with the de-pinning transition. The model system once properly tuned using real earthquake data, may help highlighting the origin of scaling in phenomenological seismic power laws.

¹This research was partly funded by the Shell-NWO/FOM programme Computational sciences for energy research under project number 14CSER022.

4:56PM L27.00003 The Sedimentation of Particles under Orthogonal Shear in Viscoelastic Fluids¹, WILLIAM L. MURCH, SREENATH KRISHNAN, ERIC S. G. SHAQFEH, Stanford University — Many engineering applications, including oil and gas recovery, require the suspension of particles in viscoelastic fluids during fluid transport and processing. A topic of specific importance involves such particle suspensions experiencing an applied shear flow in a direction perpendicular to gravity (referred to as orthogonal shear). Previously, it has been shown that particle sedimentation coupled with an orthogonal shear flow can reduce the particle settling rate in elastic fluids. The underlying mechanism of this enhanced coupling drag is not fully understood, particularly at finite Weissenberg numbers. This talk examines the role of fluid elasticity on a single, non-Brownian, rigid sphere settling in orthogonal shear using experiments and numerical simulations. New experiments were performed in a Taylor-Couette flow cell using Boger fluids to study the coupling drag as a function of the shear and sedimentation Weissenberg numbers as well as particle confinement. The elastic effect was also studied with fully 3D simulations of flow past a rigid sphere, using the FENE-P constitutive model to describe the polymeric fluid rheology. These simulations show good agreement with the experiments and allow for further insight into the mechanism of elasticity-enhanced drag.

¹This material is based upon work supported by the National Science Foundation Graduate Research Fellowship.

5:09PM L27.00004 Modelling Polymer Deformation and Welding Behaviour during 3D Printing¹, CLAIRE MCILROY, PETER OLMSTED, Georgetown University — 3D printing has the potential to transform manufacturing processes, yet improving the strength of printed parts, to equal that of traditionally-manufactured parts, remains an underlying issue. The most common method, fused deposition modelling, involves melting a thermoplastic, followed by layer-by-layer extrusion of the material to fabricate a three-dimensional object. The key to the ensuring strength at the weld between these layers is successful inter-diffusion. However, as the printed layer cools towards the glass transition temperature, the time available for diffusion is limited. In addition, the extrusion process significantly deforms the polymer micro-structure prior to welding and consequently affects how the polymers “re-entangle” across the weld. We have developed a simple model of the non-isothermal printing process to explore the effects that typical printing conditions and amorphous polymer rheology have on the ultimate weld structure. In particular, we incorporate both the stretch and orientation of the polymer using the Rolie-Poly constitutive equation to examine how the melt flows through the nozzle and is deposited onto the build plate. We then address how this deformation relaxes and contributes to the thickness and structure of the weld.

¹National Institute for Standards and Technology (NIST) and Georgetown University

5:22PM L27.00005 Straining soft colloids in aqueous nematic liquid crystals¹, SAVERIO SPAGNOLI, PETER MUSHENHEIM, JOEL PENDERY, DOUGLAS WEIBEL, NICHOLAS ABBOTT, University of Wisconsin-Madison — Liquid crystals (LCs) are anisotropic, viscoelastic fluids that can be used to direct colloids into organized assemblies with unusual optical, mechanical, and electrical properties. In past studies, the colloids have been sufficiently rigid that their individual shapes and properties have not been strongly coupled to elastic stresses imposed by the LCs. We will discuss how soft colloids (micrometer-sized shells) behave in LCs. We reveal a sharing of strain between the LC and shells, resulting in formation of spindle-like shells and other complex shapes. These results hint at previously unidentified designs of reconfigurable soft materials with applications in sensing and biology. Related effects relevant to biocomotion will also be touched upon.

¹Wisconsin MRSEC Grant DMR- 1121288

5:35PM L27.00006 Viscoelastic fluid-structure interaction between a non-Newtonian fluid flow and flexible cylinder, ANITA DEY, YAHYA MODARRES-SADEGHI, JONATHAN ROTHSTEIN, Univ of Mass - Amherst — It is well known that when a flexible or flexibly-mounted structure is placed perpendicular to the flow of a Newtonian fluid, it can oscillate due to the shedding of separated vortices at high Reynolds numbers. If the same flexible object is placed in non-Newtonian flows, however, the structure's response is still unknown. Unlike Newtonian fluids, the flow of viscoelastic fluids can become unstable at infinitesimal Reynolds numbers due to a purely elastic flow instability. In this talk, we will present a series of experiments investigating the response of a flexible cylinder placed in the cross flow of a viscoelastic fluid. The elastic flow instabilities occurring at high Weissenberg numbers can exert fluctuating forces on the flexible cylinder thus leading to nonlinear periodic oscillations of the flexible structure. These oscillations are found to be coupled to the time-dependent state of viscoelastic stresses in the wake of the flexible cylinder. The static and dynamic responses of the flexible cylinder will be presented over a range of flow velocities, along with measurements of velocity profiles and flow-induced birefringence, in order to quantify the time variation of the flow field and the state of stress in the fluid.

5:48PM L27.00007 Orientation of contravariant and covariant polymers and associated energy transfer in elasto-inertial turbulence, KIYOSI HORIUTI, AOI SUZUKI, Tokyo Institute of Technology, Japan — It is generally assumed that the polymers in viscoelastic turbulence are advected affinely with the macroscopically-imposed deformation, while de Gennes (1986) hypothesized that stretched polymers may exhibit rigidity. We conduct assessment on this hypothesis in homogeneous isotropic turbulence by connecting mesoscopic Brownian description of elastic dumbbells to macroscopic DNS. The dumbbells are advected either affinely (contravariant) or non-affinely (covariant). We consider the elasto-inertial regime (Valente et al. 2014). Using the approximate solution of the constitutive equation for the polymer stress, we show that when the dumbbells are highly stretched, $-S_{ik}S_{kl}S_{li}$ term (S_{ij} is strain-rate tensor) governs the transfer of solvent energy either to dissipation or to the elastic energy stored in the polymers. In the contravariant polymer, the elastic energy production term $P_e < 0$ and the dissipation production term $P_\varepsilon > 0$. The elastic energy is transferred backwardly into the solvent and dissipation is enhanced. In the covariant polymer, $P_e > 0$ and $P_\varepsilon > 0$. When the dumbbells are aligned with one of eigenvectors of S_{ij} , P_e predominates P_ε , and marked reduction of drag is achieved.

6:01PM L27.00008 In a sea of sticky molasses: The physics of the Boston Molasses Flood, NICOLE SHARP, FYFD, JORDAN KENNEDY, SHMUEL RUBINSTEIN, Harvard University — On January 15th, 1919, shortly after 12:40 pm local time, a giant storage tank collapsed in Boston's crowded North End, releasing more than 8.7 million liters of molasses. Contemporary accounts estimated the initial wave was 7.6 meters tall and moved at more than 15 m/s. In moments, molasses engulfed the Commercial Street area, flattening buildings, damaging the elevated train, killing 21 people, and injuring 150 more. Molasses is a viscoelastic fluid 1.5 times as dense as water with a viscosity roughly 4000 times greater. This talk will explore the physics of the Boston Molasses Flood, including the effects of temperature fluctuations and molasses rheology on events leading up to the tank's collapse and their impact on subsequent rescue efforts.

6:14PM L27.00009 Nonlinear viscoelasticity and generalized failure criterion for biopolymer gels¹, THIBAUT DIVOUX, CNRS / Centre de Recherche Paul Pascal, BAVAND KESHAVARZ, Department of Mechanical Engineering, Massachusetts Institute of Technology, SBASTIEN MANNEVILLE, ENS Lyon, GARETH MCKINLEY, Department of Mechanical Engineering, Massachusetts Institute of Technology — Biopolymer gels display a multiscale microstructure that is responsible for their solid-like properties. Upon external deformation, these soft viscoelastic solids exhibit a generic nonlinear mechanical response characterized by pronounced stress- or strain-stiffening prior to irreversible damage and failure, most often through macroscopic fractures. Here we show on a model acid-induced protein gel that the nonlinear viscoelastic properties of the gel can be described in terms of a 'damping function' which predicts the gel mechanical response quantitatively up to the onset of macroscopic failure. Using a nonlinear integral constitutive equation built upon the experimentally-measured damping function in conjunction with power-law linear viscoelastic response, we derive the form of the stress growth in the gel following the start up of steady shear. We also couple the shear stress response with Bailey's durability criteria for brittle solids in order to predict the critical values of the stress σ_c and strain γ_c for failure of the gel, and how they scale with the applied shear rate. This provides a generalized failure criterion for biopolymer gels in a range of different deformation histories.

¹This work was funded by the MIT-France seed fund and by the CNRS PICS-USA scheme (36939). BK acknowledges financial support from Axalta Coating Systems.

Monday, November 21, 2016 4:30PM - 6:40PM –

Session L28 Particle-laden Flows: Clustering and Droplet Growth F149 - Perrine Pepiot, Cornell University

4:30PM L28.00001 Numerical investigation of the role of clustering during oxygen-carrier regeneration in Chemical Looping Combustion¹, HIMANSHU GOYAL, PERRINE PEPIOT, Cornell University — In the air-reactor of a dual-bed Chemical Looping Combustion (CLC) system, the spent oxygen-carrier, in the form of metal or reduced metal oxide, is oxidized with air, typically in a high velocity riser reactor. Such a configuration provides challenging modeling issues, as the granular flow is characterized by a highly fluctuating solid volume fraction due to the formation of dense clusters. This may strongly affect the solid residence time in the air-reactor, and therefore, the extent of the oxygen-carrier regeneration and ultimately, the overall reactivity of the carrier in the fuel reactor. Here, we investigate how clustering impacts gas-solid chemical reactions in the reactor using a detailed Lagrange-Euler computational framework. The simulations account for both mass and heat transfer between the gas phase and the metal oxide particles, and the evolution of oxygen content of the metal oxide particles, or equivalently, their degree of oxidation. Two particle models of different complexity are considered. Results are analyzed to quantify the relative importance on the regeneration process of the reactor hydrodynamics.

¹This material is based upon work supported by the National Science Foundation under Grant No. CBET-1638837

4:43PM L28.00002 Transport and Morphology of Nanoparticle Aggregates in Packed Beds: Findings from LBM simulations, NGOC PHAM, DIMITRIOS PAPAVASSILIOU, The University of Oklahoma — Aggregation of colloidal particles in porous media has attracted attention because of possible pore clogging and sedimentation, which reduces the particle breakthrough. In addition, in some systems, further attachment of colloidal particles on deposited aggregates is expected. In this study, the morphology of nanoparticle aggregates, propagating in beds packed with spheres under different electrolyte conditions, is numerically investigated. In our simulation, the nanoparticles are advanced by balancing forces such as drag, random force, buoyancy, gravitational force, electrostatic repulsion, and van der Waals attractive force. When the van der Waals forces take over, the aggregates are formed. The packed beds are made of spheres, either ideally packed or randomly packed in simulation boxes. Sequentially, the flow field of water inside the packed beds is generated, using the lattice Boltzmann method (LBM). In conjunction with that, a Lagrangian framework [1, 2] is applied to record the trajectories of the free nanoparticles and the aggregated nanoparticles. Within the scope of this study, we draw attention to the change of the morphology of the aggregates, reflected by their fractal dimension, under various electrolyte and packing conditions. REFERENCES 1. R. S. Voronov, S. VanGordon, V. I. Sikavitsas, D. V. Papavassiliou, Int. J. Num. Meth. Fluids, 67, 501-517, 2011 2. N.H. Pham, D.P. Swatske, J.H. Harwell, B-J Shiau, D.V. Papavassiliou, Int. J. Heat & Mass Transf., 72, 319-328, 2014.

4:56PM L28.00003 Columnar structure formation of a dilute suspension of settling spherical particles in a quiescent fluid¹, SANDER G. HUISMAN, THOMAS BAROIS, MICKAEL BOURGOIN, Univ Lyon, Ens de Lyon, Univ Claude Bernard, CNRS, Laboratoire de Physique, F-69342 Lyon, France, AGATHE CHOUIPPE, TODOR DOYCDEV, Institute for Hydromechanics, Karlsruhe Institute of Technology, 76131 Karlsruhe, Germany, PETER HUCK, CARLA BELLO MORALES, Univ Lyon, Ens de Lyon, Univ Claude Bernard, CNRS, Laboratoire de Physique, F-69342 Lyon, France, MARKUS UHLMANN, Institute for Hydromechanics, Karlsruhe Institute of Technology, 76131 Karlsruhe, Germany, ROMAIN VOLK, Univ Lyon, Ens de Lyon, Univ Claude Bernard, CNRS, Laboratoire de Physique, F-69342 Lyon, France — The settling of heavy spherical particles in a column of quiescent fluid is investigated. The performed experiments cover a range of Galileo numbers ($110 \leq Ga \leq 310$) for a fixed density ratio of $\Gamma = \rho_p/\rho_f = 2.5$. In this regime the wake undergoes several transitions for increasing Ga resulting in particle motions that are successively: vertical, oblique, oblique oscillating, and finally chaotic. In this work volume fractions up to $\Phi_V = \mathcal{O}(10^{-3})$ are investigated. Multi-camera recordings of settling particles are recorded and tracked over time in 3 dimensions. A variety of analysis including Voronoi analysis and pair angle analysis are performed and show a strong clustering behavior along with an enhancement settling velocity. The experimental findings are compared to simulations.

¹Procope 54366584, ANR-12-BS09-0011 "TEC2", UH242/1-2.

5:09PM L28.00004 Effects of clustering on heat transfer in particle-laden turbulence¹, HADI POURANSARI, ALI MANI, Stanford University — Particle-laden flows are ubiquitous in variety of natural and industrial phenomena. Rain droplets in clouds, protoplanetary disks, and combustion chambers are examples in which particles are interacting with a background turbulence. It is well known that interaction of particles and turbulent flow results in preferential concentration. The extent of preferential concentration depends on ratio of particle relaxation time and turbulent eddies time scale. In this work, we consider particle-laden turbulent flows, in which particles are heated. This is the case for example in the particle-based solar receivers where particles absorb external radiation and heat the background gas. We use three-dimensional variable density direct numerical simulations for the turbulent flow and Lagrangian point-particle tracking to study the implication of particle clustering in particle-to-gas heat transfer. We investigate variety of non-dimensional numbers including particle Stokes number, Reynolds number, and mass loading ratio. Using our statistical analyses we introduce a model to correct the particle-to-gas heat transfer to account for particle clustering. This can be employed in Reynolds average Navier Stokes (RANS) computations.

¹Supported by DOE under PSAAP2 program at Stanford University

5:22PM L28.00005 Coherent clusters of inertial particles in homogeneous turbulence

, LUCIA BAKER, Univ of Minnesota - Twin Cities , ARI FRANKEL, ALI MANI, Stanford University, FILIPPO COLETTI, Univ of Minnesota - Twin Cities — Clustering of heavy particles in turbulent flows manifests itself in a broad spectrum of physical phenomena, including sediment transport, cloud formation, and spray combustion. However, a clear topological definition of particle cluster has been lacking, limiting our ability to describe their features and dynamics. Here we introduce a definition of coherent cluster based on self-similarity, and apply it to the distribution of heavy particles in direct numerical simulations of homogeneous isotropic turbulence. We consider a range of particle Stokes numbers, with and without the effect of gravity. Clusters show self-similarity at length scales larger than twice the Kolmogorov length, with a specific fractal dimension. In the absence of gravity, clusters demonstrate a tendency to sample regions of the flow where strain is dominant over vorticity, and to align themselves with the local vorticity vector; when gravity is present, the clusters tend to align themselves with gravity, and their fall speed is different from the average settling velocity. This approach yields observations which are consistent with findings obtained from previous studies while opening new avenues for analysis of the topology and evolution of particle clusters in a wealth of applications.

5:35PM L28.00006 Particle clustering within a two-phase turbulent pipe jet¹ , TIMOTHY LAU, GRAHAM NATHAN, The University of Adelaide —

A comprehensive study of the influence of Stokes number on the instantaneous distributions of particles within a well-characterised, two-phase, turbulent pipe jet in a weak co-flow was performed. The experiments utilised particles with a narrow size distribution, resulting in a truly mono-disperse particle-laden jet. The jet Reynolds number, based on the pipe diameter, was in the range $10000 \leq Re_D \leq 40000$, while the exit Stokes number was in the range $0.3 \leq Sk_D \leq 22.4$. The particle mass loading was fixed at $\phi = 0.4$, resulting in a flow that was in the two-way coupling regime. Instantaneous particle distributions within a two-dimensional sheet was measured using planar nephelometry while particle clusters were identified and subsequently characterised using an in-house developed technique. The results show that particle clustering is significantly influenced by the exit Stokes number. Particle clustering was found to be significant for $0.3 \leq Sk_D \leq 5.6$, with the degree of clustering increasing as Sk_D is decreased. The clusters, which typically appeared as filament-like structures with high aspect ratio oriented at oblique angles to the flow, were measured right from the exit plane, suggesting that they were generated inside the pipe.

¹The authors acknowledge the financial contributions by the Australian Research Council (Grant No. DP120102961) and the Australian Renewable Energy Agency (Grant No. USO034).

5:48PM L28.00007 Global stability of inertial particle trajectories in a two dimensional flow

, SENBAGARAMAN SUDARSANAM, PHANINDRA TALLAPRAGADA, Mechanical Engineering, Clemson University — The trajectories of inertial particles moving even in a two dimensional fluid flow exhibit complex dynamics, in particular preferential clustering in some sub domains of the fluid. This preferential clustering is influenced by the vorticity field. Based on the Maxey-Riley equation, several Eulerian criteria have been proposed in the past that classify the fluid region into local stable and unstable regions which roughly act as attracting and repelling regions for inertial particles. We demonstrate through examples that the locally unstable regions of the fluid domain can nevertheless act as global attractors. This global stability of unstable regions can partly explain the experimental evidence that particle clustering in fluids is more robust than usually predicted. The example relies on fluid flow generated by point vortices. Such vortex fields are often encountered in several microfluidic flows where the manipulation of the motion of inertial particles has several important applications.

6:01PM L28.00008 Measurements of the cross-sectional distributions of spherical particles suspended in rectangular channel flows , TAKAHIRO IMANISHI, TAKUYA YABU, HIROSHI YAMASHITA, TOMOAKI ITANO, MASAKO SUGIHARA-SEKI, Kansai Univ — We investigated the inertial migration of neutrally buoyant spherical particles using millimeter-sized rectangular channels of various aspect ratios ($AR = 1 - 4.2$), in the range of Reynolds numbers (Re) from 100 to 2000. The Reynolds number was defined as UH/ν , where U is the maximum flow velocity, H is the length of the shorter face of the channel cross-section, and ν is the kinematic viscosity. Dilute suspensions of polystyrene particles of diameter $d = 300 - 650 \mu\text{m}$ were used. For the size ratio $d/H = 0.1 - 0.25$, the observation of particle positions at downstream cross-sections revealed that the particles were aligned in a straight or curved line nearly parallel to the longer face of the channel cross-section and their probability density function showed a sharp peak at a certain distance from the channel centerline. These focusing positions of particles were found to depend on Re , d/H and AR . They approached the channel centerline with increasing Re . As AR increased for constant Re and constant d/H , focusing positions moved closer to the channel centerline, and reached asymptotic positions for $AR > 2$.

6:14PM L28.00009 Correlation of Cloud Droplet Growth with the Scalar Fluctuations in a Turbulent Moist Convection¹ , KAMAL KANT CHANDRAKAR, WILL CANTRELL, KELKEN CHANG, DAVID CIOCHETTO, DENNIS NIEDERMEIER, Michigan Technological Univ, MIKHAIL OVCHINNIKOV, Pacific Northwest National Laboratory, RAYMOND SHAW, FAN YANG, Michigan Technological Univ — Cloud droplet growth in a turbulent environment is studied by creating turbulent moist Rayleigh-Bénard convection in the Michigan Tech Pi Chamber. Cloud formation is achieved by injecting aerosols into the water-supersaturated environment created by the isobaric mixing of saturated air at different temperatures. A range of steady-state cloud droplet number concentration is achieved by supplying aerosols at different rates. As steady-state droplet number concentration is decreased the mean droplet size increases as expected, but also the width of the size distribution increases. This increase in the width is associated with larger supersaturation fluctuations due to the slow droplet microphysical response (sink of the water vapor) compared to the fast turbulent mixing (source of the water vapor). The observed standard deviation of the squared droplet radius is a linear function of the combined time scale of the system $\tau_s^{-1} = \tau_c^{-1} + \tau_t^{-1}$; here, τ_c is the phase relaxation time and τ_t is the turbulence correlation time. A stochastic differential equation approach for supersaturation also predicts the same linear response. This finding has significance for cloud-radiation budgets and precipitation formation.

¹This work was supported by the National Science Foundation, grant AGS-1623429.

6:27PM L28.00010 Continuous evolution of cloud droplet spectrum in cumulus cloud¹

, TOSHIYUKI GOTOH, IZUMI SAITO, TAKESHI WATANABE, Nagoya Institute of Technology — We have developed a new method that can seamlessly simulate the continuous growth of cloud droplets to raindrops from the first principle. A cubic box ascending with a mean updraft was introduced and the updraft velocity was self-consistently determined in such a way that the mean turbulent velocity within the box vanished. All the degrees of freedom were numerically integrated by using the Lagrangian dynamics for the droplets and the Eulerian direct numerical simulation for the turbulence. The key processes included were turbulent transport, condensation/evaporation, Reynolds number dependent drag, collision-coalescence, and entrainment. We have examined the evolution of the droplet spectrum over 400 s for a few of the initial droplet spectra: (1) single peak, (2) double peaks, (3) observed distribution, each of which had the same initial mean radius $10 \mu\text{m}$ and the same mean droplet density $n_p = 125 \text{ cm}^{-3}$. The turbulence was in steady state at $R_\lambda = 86$ and $\epsilon = 33 \text{ cm}^2\text{s}^{-3}$. It is found that the mass spectrum peak moves slowly toward the larger radius in the early stage and then quickly evolves to have the second peak through the autoconversion to the accretion state. Effects of the condensation and coalescence would also be reported.

¹Grants-in-Aid for Scientific Research Nos.15H02218 and hp150088, hp160085 and jh160012

Monday, November 21, 2016 4:30PM - 6:40PM –

Session L29 CFD: Algorithms I F150 - Anne Staple, Virginia Tech

4:30PM L29.00001 Acceleration of incremental-pressure-correction incompressible flow computations using a coarse-grid projection method, ALI KASHEFI, 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 — Coarse grid projection (CGP) methodology is a novel multigrid method for systems involving decoupled nonlinear evolution equations and linear elliptic equations. The nonlinear equations are solved on a fine grid and the linear equations are solved on a corresponding coarsened grid. Mapping functions transfer data between the two grids. Here we propose a version of CGP for incompressible flow computations using incremental pressure correction methods, called IFEi-CGP (implicit-time-integration, finite-element, incremental coarse grid projection). Incremental pressure correction schemes solve Poisson's equation for an intermediate variable and not the pressure itself. This fact contributes to IFEi-CGP's efficiency in two ways. First, IFEi-CGP preserves the velocity field accuracy even for a high level of pressure field grid coarsening and thus significant speedup is achieved. Second, because incremental schemes reduce the errors that arise from boundaries with artificial homogenous Neumann conditions, CGP generates undamped flows for simulations with velocity Dirichlet boundary conditions. Comparisons of the data accuracy and CPU times for the incremental-CGP versus non-incremental-CGP computations are presented.

4:43PM L29.00002 ABSTRACT WITHDRAWN —

4:56PM L29.00003 Simulating incompressible flow on moving meshfree grids using General Finite Differences (GFD)¹, YAROSLAV VASYLIV, ALEXANDER ALEXEEV, Georgia Institute of Technology — We simulate incompressible flow around an oscillating cylinder at different Reynolds numbers using General Finite Differences (GFD) on a meshfree grid. We evolve the meshfree grid by treating each grid node as a particle. To compute velocities and accelerations, we consider the particles at a particular instance as Eulerian observation points. The incompressible Navier-Stokes equations are directly discretized using GFD with boundary conditions enforced using a sharp interface treatment. Cloud sizes are set such that the local approximations use only 16 neighbors. To enforce incompressibility, we apply a semi-implicit approximate projection method. To prevent overlapping particles and formation of voids in the grid, we propose a particle regularization scheme based on a local minimization principle. We validate the GFD results for an oscillating cylinder against the lattice Boltzmann method and find good agreement.

¹Financial support provided by National Science Foundation (NSF) Graduate Research Fellowship, Grant No. DGE-1148903

5:09PM L29.00004 A characteristic mapping method for two-dimensional incompressible Euler flows¹, BADAL YADAV, OLIVIER MERCIER, JEAN-CHRISTOPHE NAVÉ, Department of Mathematics and Statistics, McGill University, CRM, Montreal, Canada, KAI SCHNEIDER, I2M-CNRS, Centre de Mathématiques et d'Informatique, Aix-Marseille Université, Marseille, France — We propose an efficient semi-Lagrangian method for solving the two-dimensional incompressible Euler equations with high precision on a coarse grid. The new approach evolves the flow map using the gradient-augmented level set method (GALSM). Since the flow map can be decomposed into submaps (each over a finite time interval), the error can be controlled by choosing the remapping times appropriately. This leads to a numerical scheme that has exponential resolution in linear time. The computational efficiency and the high precision of the method are illustrated for a vortex merger and a four mode flow. Comparisons with a Cauchy-Lagrangian method are also presented.

¹KS thankfully acknowledges financial support from the French Research Federation for Fusion Studies within the framework of the European Fusion Development Agreement (EFDA).

5:22PM L29.00005 Reference Map Technique for Incompressible Fluid–Structure Interaction Problems, CHRIS RYCROFT, Harvard University, CHEN-HUNG WU, Massachusetts Institute of Technology, YUE YU, Lehigh University, KEN KAMRIN, Massachusetts Institute of Technology — We present a fully Eulerian approach to simulate soft structures immersed in an incompressible fluid. The flow is simulated on a fixed grid using a second order projection method to solve the incompressible Navier–Stokes equations, and the fluid–structure interfaces are modeled using the level set method. By introducing a reference map variable to model finite-deformation constitutive relations in the structure on the same grid as the fluid, the interfacial coupling is highly simplified. This fully Eulerian approach provides a computationally efficient alternative to moving mesh approaches. Example simulations featuring many-body contacts and flexible swimmers will be presented.

5:35PM L29.00006 Application of a derivative-free global optimization algorithm to the derivation of a new time integration scheme for the simulation of incompressible turbulence., SHAHROUZ ALIMOHAMMADI, DANIELE CAVAGLIERI, POORIYA BEYHAGHI, THOMAS R. BEWLEY, University of California San Diego — This work applies a recently developed Derivative-free optimization algorithm to derive a new mixed implicit-explicit (IMEX) time integration scheme for Computational Fluid Dynamics (CFD) simulations. This algorithm allows imposing a specified order of accuracy for the time integration and other important stability properties in the form of nonlinear constraints within the optimization problem. In this procedure, the coefficients of the IMEX scheme should satisfy a set of constraints simultaneously. Therefore, the optimization process, at each iteration, estimates the location of the optimal coefficients using a set of global surrogates, for both the objective and constraint functions, as well as a model of the uncertainty function of these surrogates based on the concept of Delaunay triangulation. This procedure has been proven to converge to the global minimum of the constrained optimization problem provided the constraints and objective functions are twice differentiable. As a result, a new third-order, low-storage IMEX Runge-Kutta time integration scheme is obtained with remarkably fast convergence. Numerical tests are then performed leveraging the turbulent channel flow simulations to validate the theoretical order of accuracy and stability properties of the new scheme.

5:48PM L29.00007 ABSTRACT WITHDRAWN —

6:01PM L29.00008 A Family of Convective-Like Energy-Stable Outflow Boundary Conditions for Incompressible Flow Simulations on Severely-Truncated Unbounded Domains¹, SUCHUAN DONG, Purdue University — A large class of flow problems are spatially developing and involves physically unbounded domains, e.g. wakes, jets, and shear layers. To numerically simulate such problems, it is necessary to truncate the domain to a finite size, and some open boundary condition (a.k.a. outflow boundary condition) will be required at the artificial boundary. Backflow instability is a commonly encountered issue with outflows or open boundaries at moderate and high Reynolds numbers. Simulations have been observed to instantly blow up when strong vortices or backflows occur at the outflow/open boundary. In this talk we present a family of convective-like open boundary conditions that effectively overcomes the backflow instability. A prominent feature of these boundary conditions is that they all ensure the energy stability of the system, even in situations where strong vortices or backflows occur at the outflow/open boundary. The proposed boundary conditions unify and provide an underlying connection between the usual convective boundary condition and the traction-free boundary condition. Several canonical wake and jet problems in open domains will be presented to demonstrate the accuracy and effectiveness of the method for overcoming backflow instability.

¹This work is partially supported by NSF (DMS-1318820 and DMS-1522537)

6:14PM L29.00009 A σ -coordinate model for 3D free-surface flows using an unstructured finite-volume technique, MIGUEL UH ZAPATA, Center for Mathematical Research, CIMAT-Merida — The aim of this work is to develop a numerical solution of three-dimensional free-surface flows using a σ -coordinate model, a projection method and an unstructured finite-volume technique. The coordinate transformation is used in order to overcome difficulties arising from free surface elevation and irregular geometry. The projection method consists to combine the momentum and continuity equations in order to establish a Poisson-type equation for the non-hydrostatic pressure. A cell-centered finite volume method with a triangular mesh in the horizontal direction is used to simulate the flows with free-surfaces, in which the average values of conserved variables are stored at the centre of each element. A parallel algorithm is also presented for the finite volume discretization of the 3D Navier-Stokes equations. The proposed parallel method is formulated by using a multi-color SOR method, a block domain decomposition and interprocessor data communication techniques with Message Passing Interface. The model has been validated by several benchmarks which numerical simulations are in good agreement with the corresponding analytical and existing experimental results.

6:27PM L29.00010 Coupling LAMMPS with Lattice Boltzmann fluid solver: theory, implementation, and applications.¹, JIFU TAN, TALID SINNO, SCOTT DIAMOND, University of Pennsylvania — Studying of fluid flow coupled with solid has many applications in biological and engineering problems, e.g., blood cell transport, particulate flow, drug delivery. We present a partitioned approach to solve the coupled Multiphysics problem. The fluid motion is solved by the Lattice Boltzmann method, while the solid displacement and deformation is simulated by Large-scale Atomic/Molecular Massively Parallel Simulator (LAMMPS). The coupling is achieved through the immersed boundary method so that the expensive remeshing step is eliminated. The code can model both rigid and deformable solids. The code also shows very good scaling results. It was validated with classic problems such as migration of rigid particles, ellipsoid particle's orbit in shear flow. Examples of the applications in blood flow, drug delivery, platelet adhesion and rupture are also given in the paper.

¹NIH

Monday, November 21, 2016 4:30PM - 6:40PM –
Session L30 Flow Instability: Falling Liquid Film F151 - Reed Ogrosky, University of Wisconsin - Madison

4:30PM L30.00001 Breakup of rivulet falling over an inclined plate., RAJESH SINGH, JANINE GALVIN, National Energy Technology Laboratory — The multiscale modeling of solvent absorption in a structured packing is a complex problem. The local hydrodynamics in the packing, specifically existing flow regimes, is a key factor for overall efficiency. A single packing unit is made of corrugated sheets arranged perpendicularly to each other. In this effort, breakup of rivulet over an inclined plate is examined, which might be helpful to explain some fundamental aspects of this system. Rivulet breakup is a complex phenomenon dictated by many factors such as solvent physical properties, contact angle (γ), inertia, plate inclination angle (θ), etc. The multiphase flow simulations using the volume of fluid method were conducted considering these factors. Decreasing solvent flow rate results in the transition of flow regimes from a film to a rivulet and then to a droplet. Demarcation between a stable and an unstable flow regime that leads to breakup is presented in terms of the critical Weber number (We_{cr}). Values of Weber number below We_{cr} correspond to breakup behavior and above to a stable rivulet. The impact of solvent properties is presented by the Kapitza number (Ka), which only depends on fluid properties. Variation of We_{cr} with Ka shows two trends depending on the Ka value of the solvent. Solvents with low Ka show a linear variation of We_{cr} with Ka whereas those with high Ka show a quadratic variation. The effect of plate inclination on the rivulet breakup reveals that We_{cr} decreases with increased θ value. In addition, higher values of γ promote breakup.

4:43PM L30.00002 Flow regimes and traveling waves for a model of gravity-driven film flows in cylindrical domains.¹, H. REED OGROSKY, Virginia Commonwealth University, ROBERTO CAMASSA, JEREMY MARZUOLA, NATHAN VAUGHN, University of North Carolina — Families of traveling wave solutions will be presented for a model of a falling viscous film on the interior of a vertical rigid tube. Each family contains a single solution at a ‘turnaround point’ with larger film thickness than all others in the family. It was previously conjectured that this turnaround point may represent a critical thickness separating two distinct flow regimes observed in physical experiments as well as two distinct types of behavior in transient solutions to the model. We will verify these hypotheses over a range of parameter values using a combination of numerical and analytical techniques. The linear stability of these solutions will also be discussed; both large- and small-amplitude solutions will be shown to be unstable, though the instability mechanisms are different for each wave type. Specifically, for small-amplitude waves, the region of relatively flat film away from the localized wave crest is subject to the same instability that makes the trivial flat-film solution unstable; for large-amplitude waves, this mechanism is present but dwarfed by a much stronger tendency to relax to a regime close to that followed by small-amplitude waves.

¹This research has been supported in part by funding from the NSF and NIEHS

4:56PM L30.00003 Hydrodynamic Characterization of Harmonically Excited Falling-Films: A Detailed Experimental and Computational Study, ALEXANDROS CHAROJIANNIS, Department of Chemical Engineering, Imperial College London, FABIAN DENNER, BEREND VAN WACHEM, Department of Mechanical Engineering, Imperial College London, MARC PRADAS, Department of Mathematics and Statistics, The Open University, SERAFIM KALLIADASIS, CHRISTOS MARKIDES, Department of Chemical Engineering, Imperial College London — We investigate the hydrodynamic characteristics of harmonically excited liquid-films flowing down a 20° incline by simultaneous application of Particle Tracking Velocimetry and Planar Laser-Induced Fluorescence (PLIF) imaging, complemented by Direct Numerical Simulations. By simultaneously implementing the above two optical techniques, instantaneous and highly localised flow-rate data were also retrieved, based on which the effect of local film topology on the flow-field underneath the wavy interface is studied in detail. Our main result is that the instantaneous flow rate varies linearly with the instantaneous film-height, as confirmed by both experiments and simulations. Furthermore, both experimental and numerical flow-rate data are closely approximated by a simple analytical relationship, which is reported here for the first time, with only minor deviations. This relationship includes the wave speed c and mean flow-rate \bar{Q} , both of which can be obtained by simple and inexpensive measurement techniques, thus allowing for spatiotemporally resolved flow-rate predictions to be made without requiring any knowledge of the full flow-field from below the wavy interface.

5:09PM L30.00004 Three-dimensional numerical simulations of falling liquid films¹, CHRISTOPHER PAIN, ZHIHUA XIE, DIMITRIOS PAVLIDIS, PABLO SALINAS, OMAR MATAR, Imperial College London — Falling liquid films down an inclined or vertical surface have rich wave dynamics, often occurring in many industrial applications, such as condensers, evaporators and chemical reactors. There are some numerical studies for falling liquid films, however most of them have focused on two-dimensional falling films or three-dimensional falling films in a periodic domain. The objective of this study is to investigate flow dynamics of fully developed three-dimensional falling films using the Navier-Stokes equations coupled with interface capturing approach. An adaptive unstructured mesh modelling framework is employed here to study this problem, which can modify and adapt unstructured meshes to better represent the underlying physics of multiphase problems and reduce computational effort without sacrificing accuracy. Numerical examples of two-dimensional and three-dimensional falling films in a long domain with different flow conditions are presented and discussed.

¹EPSRC UK Programme Grant MEMPHIS (EP/K003976/1)

5:22PM L30.00005 Hydrodynamic balance of solitary waves on falling liquid films¹, BEREND VAN WACHEM, FABIAN DENNER, ALEXANDROS CHAROJIANNIS, Imperial College London, MARC PRADAS, The Open University, CHRISTOS N. MARKIDES, SERAFIM KALLIADASIS, Imperial College London — Falling liquid films at sufficiently high Reynolds numbers are unstable to long-wave perturbations which at low frequencies evolve into fast moving solitary waves. These solitary waves are strongly nonlinear structures characterised by a dominant elevation with a long tail and steep front, typically with capillary ripples preceding the main wave hump. The objective of our work is to identify the key physical mechanisms governing these solitary waves through direct numerical simulations and experiments. Our results demonstrate that the height and shape of solitary waves is governed by a subtle balance between inertia and surface tension [Denner et al., Phys. Rev. E 93 (2016), 033121]. This leads, for instance, to a stabilisation of the wave height after the onset of flow recirculation in the solitary waves in the moving frame of reference, since the flow rate and, consequently, the effective inertia acting on the waves, are reduced as a result of the recirculation. In addition, the capillary ripples in front of the main solitary humps are strongly contributing to the hydrodynamic balance of solitary waves and we establish a connection between the creation of capillary ripples and the height, stability and speed of the solitary wave.

¹We acknowledge financial support from the Engineering and Physical Sciences Research Council through grants No. EP/K008595/1 and No. EP/M021556/1.

5:35PM L30.00006 Thermocapillary control of falling liquid films by substrate heating, ALICE THOMPSON, Univ of Manchester, SUSANA GOMES, MICHAEL DALLASTON, FABIAN DENNER, SERAFIM KALLIADASIS, Imperial College London — We analyse the problem of controlling a falling liquid film by selective heating of the substrate supporting the flow. Such heating affects the film dynamics through Marangoni stresses, and will be chosen in response to real-time observations of the film height profile. We begin by developing a new low-dimensional (LD) model for the dynamics of a thin film subject to heating which varies in space and time. The model includes the effects of convection and diffusion, so that local heating applied briefly at the substrate can have a long-lasting and wide-ranging effect on the surface temperature. We demonstrate that our LD model is in good agreement with full Navier-Stokes (NS) equations and we use it to develop heating strategies which drive the film towards either a uniform state or into a desired non-uniform profile. We further develop a hierarchy of control strategies subject to realistic limitations, such as having influence over only a few localised heating strips, the ability to sense the height profile at a few fixed locations, and dealing with time delays and uncertainty in observation or application of heating. We test the robustness of our control strategies in closed- and open-domain simulations of the LD model and also in fully coupled NS calculations.

5:48PM L30.00007 Three-dimensional direct numerical simulations of co/counter-current vertical gas-Liquid annular flows¹, ASMA FARHAOUI, LYES KAHOUADJI, Imperial College London, JALEL CHERGUI, DAMIR JURIC, LIMSI, CRNS, France, SEUNGWON SHIN, Hongik University, South Korea, RICHARD CRASTER, OMAR MATAR, Imperial College London — We carry out three-dimensional numerical simulations of co/counter current Gas-Liquid annular flows using the parallel code, BLUE, based on a projection method for the resolution of the Navier-Stokes equations and a hybrid Front-Tracking/Level-Set method for the interface advection. Gas-Liquid annular flows and falling films in a pipe are present in a broad range of industrial processes. This configuration consists of an important multiphase flow regime where the liquid occupies the area adjacent to the internal circumference of the pipe and the gas flows in the pipe core. Experimentally, four distinctive flow regimes were identified (dual-wave, thick ripple, disturbance wave and regular wave regimes), that we attempt to simulate. In order to visualize these different regimes, various liquid (water) and gas (air) flow-rates are investigated.

¹EPSRC UK Programme Grant EP/K003976/1

6:01PM L30.00008 Interfacial turbulence and regularization in electrified falling films¹, DMITRI TSELUIKO, Loughborough University, MARK BLYTH, University of East Anglia, TE-SHENG LIN, National Chiao Tung University, SERAFIM KALLIADASIS, Imperial College London — Consider a liquid film flowing down an inclined wall and subjected to a normal electric field. Previous studies on the problem [1] invoked the long-wave approximation. Here, for the first time, we analyze the Stokes-flow regime using both a non-local long-wave model and the full system of governing equations. For an obtuse inclination angle and strong surface tension, the evolution of the interface is chaotic in space and time. However, a sufficiently strong electric field has a regularizing effect, and the time-dependent solution evolves into an array of continuously interacting pulses, each of which resembles a single-hump solitary pulse. This is the so-called interfacial turbulence regime. For an acute inclination angle and a sufficiently small supercritical value of the electric field, solitary-pulse solutions do not exist, and the time-dependent solution is instead a modulated array of short-wavelength waves. When the electric field is increased, the evolution of the interface first becomes chaotic, but then is regularized so that an array of pulses is generated. A coherent-structure theory for such pulses is developed and corroborated by numerical simulations.

[1] T.S. Lin, M. Pradas, S. Kalliadasis, D.T. Papageorgiou, D. Tseluiko, SIAM J Appl Math 75, 538-563 (2015)

¹This work was supported by the EPSRC under grants EP/J001740/1 and EP/K041134/1.

6:14PM L30.00009 Attracting wave regimes for two layer falling films., GOKCEN CEKIC, Istanbul University, GRIGORY SISOEV, SGM Ltd, Moscow, Russia. — We analyze flows of two-layer falling films by using approximate long-wave model. There is a non-uniqueness of steady-traveling wave regimes as solutions of the problem which is utilised in mass transfer technologies. To select the wave regimes developing in two-layer films, systematic transient computations have been carried out to create a map of the attracting wave regimes, so-called dominating waves, which can be used to model real-life processes.

6:27PM L30.00010 Three-dimensional direct numerical simulation of falling liquid films¹, JALEL CHERGUI, DAMIR JURIC, LIMSI, CNRS, SUENGWON SHIN, Honkgik University, Republic of korea, LYES KA-HOUADJI, RICHARD CRASTER, OMAR MATAR, Imperial College London — The dynamics of a thin film falling down an inclined solid surface have attracted the attention of many researchers because of the richness and variety of waves which develop on its liquid-air interface. Besides experimental work, the problem has been widely studied in the literature but direct numerical simulations have been limited to two-dimensions or low-dimensional modeling for three-dimensional problems. We present a computational study of falling liquid films in a three-dimensional inclined rectangular domain (45°) using the massively parallel code BLUE for Lagrangian tracking of arbitrarily deformable phase interfaces. Calculations are carried out on $O(10^3)$ cores in a large domain (24cm \times 12cm \times 1.5 cm) for Reynolds and Kapitza numbers of 100 and 10, respectively, in order to obtain several three-dimensional wave patterns and solitary waves.

¹EPSRC UK Programme Grant MEMPHIS (EP/K003976/1)

Monday, November 21, 2016 4:30PM - 6:27PM – Session L31 Experimental Techniques - Density Gradient & Free Surface F152 - Brian Thurow, Auburn University

4:30PM L31.00001 3D Imaging of Density Gradients Using Plenoptic BOS, JENNA KLEMKOWSKY, CHRIS CLIFFORD, TIMOTHY FAHRINGER, BRIAN THUROW, Auburn University — The combination of background oriented schlieren (BOS) and a plenoptic camera, termed Plenoptic BOS, is explored through two proof-of-concept experiments. The motivation of this work is to provide a 3D technique capable of observing density disturbances. BOS uses the relationship between density and refractive index gradients to observe an apparent shift in a patterned background through image comparison. Conventional BOS systems acquire a single line-of-sight measurement, and require complex configurations to obtain 3D measurements, which are not always conducive to experimental facilities. Plenoptic BOS exploits the plenoptic camera's ability to generate multiple perspective views and refocused images from a single raw plenoptic image during post processing. Using such capabilities, with regards to BOS, provides multiple line-of-sight measurements of density disturbances, which can be collectively used to generate refocused BOS images. Such refocused images allow the position of density disturbances to be qualitatively and quantitatively determined. The image that provides the sharpest density gradient signature corresponds to a specific depth. These results offer motivation to advance Plenoptic BOS with an ultimate goal of reconstructing a 3D density field.

4:43PM L31.00002 Assessment of sources of error in Background Oriented Schlieren (BOS) measurements, LALIT RAJENDRAN, BHAVINI SINGH, MATTHEW GIARRA, SALLY BANE, PAVLOS VLACHOS, Purdue University — Background Oriented Schlieren (BOS) is used to measure density gradients in a flow by tracking the apparent distortion of a target dot pattern. The quality of a BOS measurement depends on several factors such as the dot pattern, illumination, density gradients, optical system, cross-correlation algorithms and density reconstruction. To understand their contributions to the final error in the measurement and to develop an optimal set of design rules, we generate high fidelity synthetic images using ray tracing simulations. Past studies use ad-hoc models (or none) for simulating these effects and do not represent the issues introduced in a typical BOS setup, thereby limiting their utility. We have developed and implemented an image generation methodology based on ray tracing, where light rays emitted from a dot pattern are traced through the experimental setup including the density gradients, to generate high fidelity images representative of a real experiment. We apply this methodology to perform a comprehensive analysis of the various sources of error in the BOS technique and to better understand the issues involved in designing a successful experiment. The results of this study can guide future experiments and provide directions to improve the image analysis tools.

4:56PM L31.00003 Three-dimensional (3D) shadowgraph technique visualizes thermal convection, JINZI HUANG, Courant Institute of Mathematical Sciences, JUN ZHANG, Courant Institute of Mathematical Sciences; NYU Department of Physics; NYU Shanghai, PHYSICS AND MATHS RESEARCH INSTITUTES, NYU SHANGHAI TEAM, APPLIED MATHS LAB, NYU TEAM — Shadowgraph technique has been widely used in thermal convection, and in other types of convection and advection processes in fluids. The technique reveals minute density differences in the fluid, which is otherwise transparent to the eyes and to light-sensitive devices. However, such technique normally integrates the fluid information along the depth of view and collapses the 3D density field onto a 2D plane. In this work, we introduce a stereoscopic shadowgraph technique that preserves the information of the fluid depth by using two cross-field shadowgraphs. The two shadowgraphs are coded with different and complementary colors, and each is seen by only one eye of the viewer. The two shadowgraphs can also be temporally modulated to achieve the same stereoscopic vision of the convective fluid. We further discuss ways to make use of this technique in order to extract useful information for research in fluids.

5:09PM L31.00004 High-speed schlieren imaging of rocket exhaust plumes, CARALYN COULTAS-MCKENNEY, KYLE WINTER, MICHAEL HARGATHER, New Mexico Tech — Experiments are conducted to examine the exhaust of a variety of rocket engines. The rocket engines are mounted in a schlieren system to allow high-speed imaging of the engine exhaust during startup, steady state, and shutdown. A variety of rocket engines are explored including a research-scale liquid rocket engine, consumer/amateur solid rocket motors, and water bottle rockets. Comparisons of the exhaust characteristics, thrust and cost for this range of rockets is presented. The variety of nozzle designs, target functions, and propellant type provides unique variations in the schlieren imaging.

5:22PM L31.00005 Studying the instantaneous velocity field in gas-sheared liquid films in a horizontal duct, JOAO VASQUES, University of Nottingham, MIKHAIL TOKAREV, ANDREY CHERDANTSEV, Kutateladze Institute of Thermophysics, Novosibirsk, Russia, DAVID HANN, BUDDHIKA HEWAKANDAMBY, BARRY AZZOPARDI, University of Nottingham — In annular flow, the experimental validation of the basic assumptions on the liquid velocity profile is vital for developing theoretical models of the flow. However, the study of local velocity of liquid in gas-sheared films has proven to be a challenging task due to the highly curved and disturbed moving interface of the phases, small scale of the area of interrogation, high velocity gradients and irregular character of the flow. This study reports on different optical configurations and interface-tracking methods employed in a horizontal duct in order to obtain high-resolution particle image velocimetry (PIV) data in such types of complex flows. The experimental envelope includes successful measurements in 2D and 3D waves regimes, up to the disturbance wave regime. Preliminary data show the presence of complex structures in the liquid phase, which includes re-circulation areas below the liquid interface due to the gas-shearing action, together with non-uniform transverse movements of the liquid phase close to the wall due to the presence of 3D waves at the interface. With the aid of the moving interface-tracking, PIV, time-resolved particle-tracking velocimetry and vorticity measurements were performed.

5:35PM L31.00006 Sensor-Free Surface Density Detector, HUIXUAN WU, University of Kansas — We have developed an optical-based method to measure the absolute air density on a wall surface in compressible turbulent boundary layers. The temporal resolution can be higher than 1MHz, and the spatial resolution can research 10 micron. For isothermal flows, our system can also be used to obtain the wall pressure distributions or volume-ratio of two-species gas. It is a powerful tool for observing turbulent fluctuations and flow separations in sub-, trans-, and supersonic airflows. The working principle of our method is to detect the air density by measuring the refractive index, which linearly depends on density and determines the transmission coefficient at the interface. For single- or multiple-point measurements, we do not need to install sensors on the wall surface, which is a big advantage compared to conventional methods. In 2D cases, a layer of anti-reflection coating is needed. The optical measurement range is not limited by the surface material or sensor. These advantages make our method a good complement or better alternative to the other approaches, such as focused laser differential interferometry technique, which provides density gradient, and pressure (temperature) sensitive paints, which depends significantly on the material properties.

5:48PM L31.00007 Free-surface tracking of submerged features to infer hydrodynamic flow characteristics, TRACY MANDEL, ITAY ROSENZWEIG, JEFFREY KOSEFF, Stanford University — As sea level rise and stronger storm events threaten our coastlines, increased attention has been focused on coastal vegetation as a potentially resilient, financially viable tool to mitigate flooding and erosion. However, the actual effect of this green infrastructure on near-shore wave fields and flow patterns is not fully understood. For example, how do wave setup, wave nonlinearity, and canopy-generated instabilities change due to complex bottom roughness? Answering this question requires detailed knowledge of the free surface. We develop easy-to-use laboratory techniques to remotely measure physical processes by imaging the apparent distortion of the fixed features of a submerged cylinder array. Measurements of surface turbulence from a canopy-generated Kelvin-Helmholtz instability are possible with a single camera. A stereoscopic approach similar to Morris (2004) and Gomit et al. (2013) allows for measurement of waveform evolution and the effect of vegetation on wave steepness and nonlinearity.

6:01PM L31.00008 Time and space analysis of turbulence of gravity surface waves¹, NICOLAS MORDANT, QUENTIN AUBOURG, SAMUEL VIBOUD, JOEL SOMMERIA, LEGI, Universite Grenoble Alpes & CNRS — Wave turbulence is a statistical state made of a very large number of nonlinearly interacting waves. The Weak Turbulence Theory was developed to describe such a situation in the weakly nonlinear regime. Although, oceanic data tend to be compatible with the theory, laboratory data fail to fulfill the theoretical predictions. A space-time resolved measurement of the waves have proven to be especially fruitful to identify the mechanism at play in turbulence of gravity-capillary waves [1]. We developed an image processing algorithm to measure the motion of the surface of water with both space and time resolution. We first seed the surface with slightly buoyant polystyrene particles and use 3 cameras to reconstruct the surface. Our stereoscopic algorithm is coupled to PIV so that to obtain both the surface deformation and the velocity of the water surface. Such a coupling is shown to improve the sensitivity of the measurement by one order of magnitude. We use this technique to probe the existence of weakly nonlinear turbulence excited by two small wedge wavemakers in a 13-m diameter wave flume. We observe a truly weakly nonlinear regime of isotropic wave turbulence. [1] Aubourg & Mordant, *Phys. Rev. Fluids* 1, 2016

¹This project has received funding from the European Research Council (ERC) under the European Union's Horizon 2020 research and innovation programme (grant agreement No 647018-WATU).

6:14PM L31.00009 Common-optical axis Fourier transform profilometry for water surface waves, MAHDI GHADIRI, ROUSLAN KRECHETNIKOV, University of Alberta — The Fourier transform profilometry – a single-shot optical profilometric measurement of surface deformation – has been widely used to visualize and measure water surface waves. This well-known method is based on an optical system composed of a video projector displaying a fringe pattern on the surface and a camera recording this pattern as the reference image. The deformed fringe pattern following deformation of the surface later is then recorded and compared to the reference image in order to produce a phase map, from which the height of the deformed surface is reconstructed through a phase-to-height relation. The biggest challenge encountered while applying this method for water surface is the light reflection which previously has been partially treated by enhancing the water light diffusivity with the addition of Titanium dioxide. As part of the effort to improve the accuracy and practical applicability of the method, in this talk we will present a new implementation of a common-optical axis geometry along with an appropriate phase-height relation. Furthermore, in the case of water surface waves, we introduce a proper light filtration, which removes all the reflections remaining after addition of Titanium dioxide. The proposed technique provides an order of magnitude improvement in the accuracy of detecting and reconstructing the surface deformation, which is crucial for studying small amplitude waves and bifurcation phenomena.

Monday, November 21, 2016 4:30PM - 6:40PM –
Session L32 On Turbulent Channels and Boundary Layers Oregon Ballroom 201 - Ronald Panton, University of Texas

4:30PM L32.00001 Pressure Fluctuations in Turbulent Wall Layers¹, RONALD PANTON, MYOUNGKYU LEE, ROBERT MOSER, University of Texas — Pressure fluctuation profile data from the channel flow DNS of Lee and Moser [*J. Fluid Mech.*, vol 774, 2015] extend to $Re_\tau \approx 5200$. In the outer region, with $Y = y/h$, the overlap layer pressure correlates very well by a log law; $\lim_{Y \rightarrow 0} \langle p^2 \rangle^+ \sim (1/\eta) \ln Y + D_o$. The constant $\eta = -0.380$ is remarkable like the von Kármán value. In the inner region, the defect variable $\mathcal{P}(y^+) \equiv \langle p^2 \rangle^+ - \langle p^2 \rangle^+|_{y=0}$ absorbs the Re_τ dependence. The inner overlap equation is; $\lim_{y+ \rightarrow \infty} \mathcal{P} \sim (1/\eta) \ln y^+ + D_i$. Together, the overlap laws imply that the wall pressure relation is $\langle p^2 \rangle^+|_{y=0} \sim (-1/\eta) \ln Re_\tau + D_i - D_o$. A completely equivalent expression, which is finite as $Re_\tau \rightarrow \infty$, is obtained by rescaling the pressure variable; $\langle p^2 \rangle^\#|_{y=0} \equiv (u_\tau/U_o) \langle p^2 \rangle^+|_{y=0} = C_1 + C_2(u_\tau/U_o)$. Here, the constants are related to η , D_o , and D_i . Additionally, it was found that the wavenumber spectrum $E_{pp}\{k_x/h\}$ does not have a k^{-1} region. However, the trends do not rule out this at higher Re .

¹This 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.

4:43PM L32.00002 Vortex statistics in turbulent channel flows¹, JOS HUGO ELSAS², LUCA ROBERTO AUGUSTO MORICONI, Univ Fed Rio de Janeiro — In order to address the role of coherent structures in wall bounded turbulence, we study the statistics of morphological and kinematic properties of vortices, such as circulation, radius and height distributions. To accomplish that, we introduce a novel vortex identification method named as "vorticity curvature criterion" which is based on the local properties of the vorticity field. We furthermore employ a background subtraction procedure to remove shearing background effects expected to be present in the topology of the streamwise/wall-normal plane flow configurations. We discuss, through a comparative study of performance with the usual swirling strength criterion, and extending the previous analyses to the detection of coherent structures in the spanwise/wall normal planes, isotropization issues for the paradigmatic case of numerical turbulent channel flows.

¹We acknowledge the funding from CNPq, CAPES and Faperj

²Presenting author

4:56PM L32.00003 Direct numerical simulation of the fully developed turbulent boundary layer¹, MELISSA KOZUL, DANIEL CHUNG, Univ of Melbourne — The term 'fully developed' is commonly applied to channel and pipe flows that are statistically stationary in time and no longer exhibit streamwise development. Following the temporal turbulent boundary layer simulation of Kozul, Chung & Monty (*J. Fluid Mech.*, vol. 796, 2016, pp. 437-472) where streamwise development was removed with periodic boundary conditions, we now remove the remaining development in time, giving a turbulent boundary layer that is 'fully developed' at finite Reynolds numbers. This is achieved by rescaling in the wall-normal direction and assuming arrested boundary-layer growth, motivated by a large-eddy turnover time estimated to be much shorter than the growth time scale of the boundary-layer. Analysis of outer-layer similarity shows that this setup, with only one additional computational term, gives a dominant balance equivalent to the high-Reynolds number asymptotics for both the spatially and temporally developing turbulent boundary layers. Our idealised, but non-physical simulation thus allows us to enforce the infinite Reynolds number dominant balance assumptions commonly made at finite Reynolds numbers. This simple setup could be used to generate inflow conditions for spatial simulations, or as a test case for model development and analysis.

¹Australian Research Council

5:09PM L32.00004 Turbulent Boundary Layers and Sediment Suspension Absent Mean Flow-Induced Shear: An Experimental Study, BLAIR JOHNSON, Arizona State University, EDWIN COWEN, Cornell University — We investigate turbulent boundary layers in the absence of mean shear at solid and sediment boundaries in an experimental facility designed to generate high Reynolds number ($Re_\lambda \sim 300$) horizontally homogeneous isotropic turbulence via randomly actuated synthetic jet arrays (RASJA - Variano & Cowen 2008). One array is an 8×8 grid of jets, while the other is a 16×16 array. We control the turbulence levels, including the integral length scale and dissipation rate, by changing the mean on-times in the jet algorithm. Particle image velocimetry (PIV) measurements are used to study the isotropic turbulence and the boundary layer. The flow is characterized by statistical metrics such as the mean flow and turbulent velocities, turbulent kinetic energy, spectra, and integral length scale. We consider the turbulent kinetic energy transport equation and examine the relationships between dissipation, production, and turbulent transport, in absence of mean flows. We compare an impermeable flat boundary, a flat permeable sediment boundary, and a rippled sediment boundary. We find that while an immobile sediment bed acts as a sink of turbulence, an active sediment boundary with frequent suspension increases turbulent velocity fluctuations. Because traditional viscous stresses due to mean velocity gradients suggest no bed friction, we develop a method for considering Reynolds stresses over short time periods as a surrogate for understanding bed stress in a zero mean shear environment.

5:22PM L32.00005 Estimation of turbulent channel flow based on the wall measurement with a statistical approach, YOSUKE HASEGAWA, The University of Tokyo, TAKAO SUZUKI, The Boeing Company — A turbulent channel flow at $Re_{tau} = 100$ with periodic boundary conditions is estimated with linear stochastic estimation only based on the wall measurement, i.e. the shear-stress in the streamwise and spanwise directions as well as the pressure over the entire wavenumbers. The results reveal that instantaneous measurement on the wall governs the success of the estimation in $y+ \gtrsim 20$. Degrees of agreement are equivalent to those reported by Chevalier et al. (2006) using a data-assimilation approach. This suggests that the instantaneous wall information dictates the estimation rather than the estimator solving the dynamical system. We feed the velocity components from the linear stochastic estimation via the body-force term into the NavierStokes system; however, the estimation slightly improves in the log layer, indicating some benefit of involving a dynamical system but over-suppression of turbulent kinetic energy beyond the viscous sublayer by the linear stochastic estimation. Motions inaccurately estimated in the buffer layer prevent from further reconstruction toward the centerline even if we relax the feedback forcing and let the flow evolve nonlinearly through the estimator. We also argue the inherent limitation of turbulent flow estimation based on the wall measurement.

5:35PM L32.00006 Characterization of linear-like Orr bursts in fully turbulent channel flows¹, MIGUEL P. ENCINAR, JAVIER JIMENEZ, Technical University of Madrid — The linearised Orr-Sommerfeld equation predicts that initially small perturbations of the cross-shear velocity become transiently amplified when tilted by the effect of a mean shear. Such transient behaviour can also be found in the large-scale structures of fully developed nonlinear shear turbulence, although affected by the non linearity of the flow. We investigate the dynamics of the bursting structures in properly filtered large-box turbulent channels at $Re_\tau = 950$, and find that all velocity components play an important role in their formation. This implies that their underlying geometry is three dimensional. We explore the latter using spatio-temporal conditionally averaged structures that show the formation of tilted rollers at the moment of the burst, and reveal a relation between the Orr-like bursts and the vertical momentum transfer.

¹Funded by the ERC COTURB project.

5:48PM L32.00007 Experimental measurements of a non-equilibrium thermal boundary layer flow, DRUMMOND BILES, ALIREZA EBADI, Student, CHRIS WHIE, Professor — Data from a newly constructed non-equilibrium and thermal boundary layer wind tunnel is presented. The bottom wall of the tunnel is a sectioned-wall design composed of twelve aluminum 6061 plates with resistive heaters adhered to their underside. Each section is heated and controlled using independent feedback loop controllers. The freestream temperature is controlled by an upstream array of resistive heaters and a feedback controller. Experimental data with strong perturbations that produce non-equilibrium boundary layer flow behaviors is presented. Data for ZPG conditions are provided for validation purposes, and the effects of non-equilibrium behaviors on the transport of momentum and heat are discussed.

6:01PM L32.00008 Transitional boundary layer in low-Prandtl-number convection at high Rayleigh number¹, JOERG SCHUMACHER, VINODH BANDARU, TU Ilmenau (Germany), AMBRISH PANDEY, IIT Kanpur (India), JANET SCHEEL, Occidental College Los Angeles (USA) — The boundary layer structure of the velocity and temperature fields in turbulent Rayleigh-Bénard flows in closed cylindrical cells of unit aspect ratio is revisited from a transitional and turbulent viscous boundary layer perspective. When the Rayleigh number is large enough the boundary layer dynamics at the bottom and top plates can be separated into an impact region of downwelling plumes, an ejection region of upwelling plumes and an interior region (away from side walls) that is dominated by a shear flow of varying orientation. This interior plate region is compared here to classical wall-bounded shear flows. The working fluid is liquid mercury or liquid gallium at a Prandtl number of $Pr = 0.021$ for a range of Rayleigh numbers of $3 \times 10^5 \leq Ra \leq 4 \times 10^8$. The momentum transfer response to these system parameters generates a fluid flow in the closed cell with a macroscopic flow Reynolds number that takes values in the range of $1.8 \times 10^3 \leq Re \leq 4.6 \times 10^4$. It is shown that particularly the viscous boundary layers for the largest Ra are highly transitional and obey some properties that are directly comparable to transitional channel flows at friction Reynolds numbers below 100.

¹This work is supported by the Deutsche Forschungsgemeinschaft.

6:14PM L32.00009 Wall-resolved LES of high Reynolds number airfoil flow near stall condition for wall modeling in LES: LESFOIL revisited¹, KENGO ASADA, Tokyo University of Science, SOSHI KAWAI, Tohoku University — Wall-resolved large-eddy simulation (LES) of an airfoil flow involving a turbulent transition and separations near stall condition at a high Reynolds number 2.1×10^6 (based on the freestream velocity and the airfoil chord length) is conducted by using K computer. This study aims to provide the wall-resolved LES database including detailed turbulence statistics for near-wall modeling in LES and also to investigate the flow physics of the high Reynolds number airfoil flow near stall condition. The LES well predicts the laminar separation bubble, turbulent reattachment and turbulent separation. The LES also clarified unsteady flow features associated with shear-layer instabilities: high frequency unsteadiness at $St \approx 130$ at the laminar separation bubble near the leading edge and low frequency unsteadiness at $St \approx 1.5$ at the separated turbulent shear-layer near the trailing edge. Regarding the near-wall modeling in LES, the database indicates that the pressure term in the mean streamwise-momentum equation is not negligible at the laminar and turbulent separated regions. This fact suggests that widely used equilibrium wall model is not sufficient and the inclusion of the pressure term is necessary for wall modeling in LES of such flow.

¹This research used computational resources of the K computer provided by the RIKEN Advanced Institute for Computational Science through the HPCI System Research project (Project ID: hp140028). This work was supported by KAKENHI (Grant Number: 16K18309).

6:27PM L32.00010 Transitions to different kinds of turbulence in a channel with soft walls.¹, VISWANATHAN KUMARAN, SAGAR SRINIVAS, Indian Inst of Science — The flow in a soft-walled channel undergoes a transition to turbulence at a Reynolds number which is a fraction of the transition Reynolds number of 1200 for a rigid channel, due to a dynamical instability caused by a fluid-wall coupling. The turbulent flow after transition in a channel with walls made of polyacrylamide gel is experimentally characterised. There are two other types of turbulence observed in sequence as the Reynolds number is increased. The first is the soft-wall turbulence, which involves wall oscillations primarily tangential to the surface, coupled with large fluid velocity fluctuations. The fluid velocity fluctuations share many of the characteristics of those in the flow past a rigid surface, but there are significant differences; the velocity fluctuations do not seem to decay to zero at the wall, and the mechanism of turbulence production seems to be different. As the Reynolds number is increased, there is a second wall-flutter transition which involves solid displacement perpendicular to the wall, and takes place only if the wall is unrestrained. The two transitions take place in sequence from a laminar flow when the soft-wall transition Reynolds number is less than 1200, and from a turbulent flow if the soft-wall transition Reynolds number exceeds 1200.

¹Science and Engineering Research Board, Government of India.

Monday, November 21, 2016 4:30PM - 6:40PM –
Session L33 Turbulent Boundary Layers: Superhydrophobic Surfaces Oregon Ballroom
202 - Rayhaneh Akhavan, University of Michigan Ann Arbor

4:30PM L33.00001 Effect of Interface Curvature on Turbulent Skin-Friction Drag Reduction with Super-Hydrophobic Micro-Grooves, RAYHANEH AKHAVAN, AMIRREZA RASTEGARI, University of Michigan, Ann Arbor — Effect of interface curvature on Drag Reduction (DR) with Super-Hydrophobic (SH) Micro-Grooves (MGs) was investigated by DNS with lattice Boltzmann methods. The liquid/gas interfaces in the SH MGs were modeled as curved, stationary, shear-free boundaries, with the interface shape determined from the Young-Laplace equation. The full range of interface protrusion angles, ranging from 0° to -90° , were investigated. DRs of 35% to 63% were realized in DNS, in turbulent channel flows at a $Re_{bulk} = 7200$ ($Re_{\tau_0} \approx 222$) with longitudinal MGs of size $14 \leq g^{+0} \leq 56$ & $g^{+0}/w^{+0} = 7$ on both walls, where g^{+0} and w^{+0} denote the widths and spacings of the MGs, in wall units of the base flow, respectively. The presence of interface curvature led to increases of 2.3% to 4.5% in the magnitude of DR, and drops of -3.5% to -13.5% in the slip velocity, at low protrusion angles, and drops of -2.2% to -12.5% in the magnitude of DR, and either drops of up to -16.5% or increases of up to 6% in the slip velocity, at high protrusion angles, compared to flat interfaces. In addition, the instantaneous pressure fluctuations on curved SH interfaces at low protrusion angles were significantly lower (by a factor of ~ 2) than those on flat interfaces.

4:43PM L33.00002 The Common Mechanism of Turbulent Skin-Friction Drag Reduction with Super-Hydrophobic Micro-Grooves and Riblets, AMIRREZA RASTEGARI, RAYHANEH AKHAVAN, University of Michigan, Ann Arbor — Drag Reduction (DR) with Super-Hydrophobic (SH) longitudinal Micro-Grooves (MGs) and riblets was investigated by DNS using lattice Boltzmann methods. The liquid/gas interfaces on the SH MGs were modeled as curved, stationary, shear-free boundaries, with the meniscus shape determined from the Young-Laplace equation. For comparison, the same geometries were also studied as riblets. DRs of 35% to 63% with SH MGs, and 10% to -17% with riblets, were realized in DNS in turbulent channel flow at $Re_b = 7200$, with MGs of size $14 \leq g^{+0} \leq 56$; $g^{+0}/w^{+0} = 7$, and protrusion angles of 0° to 90° , where g^{+0} and w^{+0} denote the widths and spacings of the MGs in base flow wall units. It was found that 100% of the DR with riblets, and 95% to 100% of the DR with SH MGs, arises from the effective slip on the walls and the resultant drop in the friction Reynolds number of the flow due to this effective slip. Modifications to the turbulence dynamics were always drag enhancing (DE) with riblets and generally DE with SH MGs. Increasing the riblet wall curvature significantly increased the wall slip velocity at the riblet tips. But this translated to an increase in DR only for $g^{+0} \approx 14$, due to significant enhancement of turbulence production at larger MG widths.

4:56PM L33.00003 The effects of interface deformation of superhydrophobic surface on turbulent flows, SHAO-CHING HUANG, JOHN KIM, University of California, Los Angeles — Direct numerical simulations of a turbulent channel flow over superhydrophobic surface are performed to study the effects of gas-liquid interface deformation. An immersed boundary method is developed to resolve the deformed gas-liquid interface. Turbulence statistics obtained from idealized interface configurations is compared to those obtained from previous studies using the flat interface assumption. Implications on the drag reduction mechanism will be discussed.

5:09PM L33.00004 Superhydrophobic surfaces in turbulent channel flow¹, YIXUAN LI, KARIM ALAME, KRISHNAN MAHESH, University of Minnesota, Twin Cities — The drag reduction effect of superhydrophobic surfaces in turbulent channel flow is studied using direct numerical simulation. The volume of fluid (VOF) methodology is used to resolve the dynamics of the interface. Laminar flow simulations show good agreement with experiment, and illustrate the relative importance of geometry and interface boundary condition. An analytical solution for the multi-phase problem is obtained that shows good agreement with simulation. Turbulent simulations over a longitudinally grooved surface show drag reduction even in the fully wetted regime. The statistics show that geometry alone can cause an apparent slip to the external flow. Instantaneous plots indicate that the grooves prevent the penetration of near wall vorticity, yielding overall drag reduction. Results for spectra, wall pressure fluctuations and correlations will be presented. Unsteady effects on the air-vapor interface will be discussed. Results for random roughness surfaces will be presented.

¹Supported by Office of Naval Research

5:22PM L33.00005 A Passive Drag Reduction Surface Design¹, CONG WANG, DAVID JEON, MORTEZA GHARIB, Caltech — Super hydrophobic surface could induce an air layer over the surface when submerged in water. This air layer is responsible for many fascinating properties of super hydrophobic surface, such as drag reduction. Unfortunately, the air layer is fragile and can be depleted by fast shear/turbulent flow. In this work, a dimpled surface with non-uniform surface wettability is proposed to increase the air layer stability by trapping air in individual dimples. A central pumping system is connected to each dimple to supply air and regulate pressure inside air bubble. Particle Image Velocimetry (PIV) is used to investigate the drag reduction effect of different geometry designs.

¹This work is supported by Office of Naval Research under Grant No. N00014-15-1-2479.

5:35PM L33.00006 DNS of turbulent flows over superhydrophobic surfaces: effect of texture randomness, JONGMIN SEO, ALI MANI, Stanford Univ — Superhydrophobic surfaces (SHS) are non-wetting surfaces consisting of hydrophobic material and nano/micro-scale structures. When in contact with overlaying liquid flows, such structures can entrap gas and therefore suppress the direct contact between water and solid, reducing skin friction. SHS patterns can utilize a wide range of geometries including posts, ridges, and etched holes, either in a pre-specified arrangement or randomly distributed. In this work we investigate how the randomness of such patterns affect the drag reduction and interfacial robustness when these surfaces are under turbulent flows. We perform direct numerical simulations of turbulent flows over randomly patterned slip surface on a wide range of texture parameters. We present slip lengths of randomly distributed SHS for texture widths $w^+ = 4 - 26$, and solid fractions from 11% to 25%. For fixed gas fraction and texture size, the slip lengths of randomly distributed textures are less than those of aligned textures. We show that the geometric randomness of texture distribution weakens the interfacial robustness of the gas pocket. Support from Office of Naval Research (ONR) under grant #3002451214 is gratefully acknowledged.

5:48PM L33.00007 Modification of Turbulent Boundary Layer Flows by Superhydrophobic Surfaces, JAMES W. GOSE, KEVIN GOLOVIN, Univ. of Michigan, JULIO BARROS, MICHAEL P. SCHULTZ, US Naval Academy, ANISH TUTEJA, MARC PERLIN, STEVEN L. CECCIO, Univ. of Michigan — Measurements of near zero pressure gradient turbulent boundary layer (TBL) flow over several superhydrophobic surfaces (SHSs) are presented and compared to those for a hydraulically smooth baseline. The surfaces were developed at the University of Michigan as part of an ongoing research thrust to investigate the feasibility of SHSs for skin-friction drag reduction in turbulent flow. The SHSs were previously evaluated in fully-developed turbulent channel flow and have been shown to provide meaningful drag reduction. The TBL experiments were conducted at the USNA in a water tunnel with a test section 2.0 m (L) \times 0.2 m (W) \times 0.2 m (H). The free-stream speed was set to 1.26 m/s which corresponded to a friction Reynolds number of 1,500. The TBL was tripped at the test section inlet with a 0.8 mm diameter wire. The upper and side walls provided optical access, while the lower wall was either the smooth baseline or a spray coated SHS. The velocity measurements were obtained with a TSI FSA3500 two-component Laser-Doppler Velocimeter (LDV) and custom-designed beam displacer operated in coincidence mode. The LDV probe volume diameter was 45 μm (approx. one wall-unit). The measurements were recorded 1.5 m downstream of the trip. When the measured quantities were normalized using the inner variables, the results indicated a significant reduction in the near wall viscous and total stresses with little effect on the flow outside the inner layer.

6:01PM L33.00008 Using time-dependent experiments and simulations to establish the role of surfactant in increasing drag over superhydrophobic surfaces, PAOLO LUZZATTO-FEGIZ, UC Santa Barbara, FRANCOIS PEAUDECERF, JULIEN R. LANDEL, RAYMOND E. GOLDSTEIN, University of Cambridge — Superhydrophobic surfaces (SHS) can potentially achieve drag reduction for both internal and external flow applications. However, experiments have provided inconsistent results, with many studies reporting significantly decreased performance. While a complete explanation is yet to be found, it has been proposed that surfactants, ubiquitous in flow applications, could be responsible, as Marangoni stresses could develop when the edges of the SHS are not aligned with the flow. However, testing this hypothesis has been challenging. Even careful experiments with purified water have shown large interfacial stresses; adding surfactant yields only small drag increases, potentially revealing a pre-existing contamination of the interface. Other common physical processes, such as thermal Marangoni stresses and interface deflection, could also explain the lower performance. We address this question with numerical simulations, including surfactant kinetics, and SHS experiments in a micro-channel, where we control temperature gradients and interface deflections. By imposing a time-dependent pressure gradient, we are able to drive complex interface dynamics that can only be explained by surfactant gradients. Our results demonstrate the role of surfactants in increasing drag over superhydrophobic surfaces.

6:14PM L33.00009 Flow through an Array of Superhydrophobic Pillars: The Role of the Air-Water Interface Shape on Drag Reduction¹, JEONG-HYUN KIM, JONATHAN ROTHSTEIN, University of Massachusetts Amherst — In this study, measurements of the pressure drop and the velocity fields associated with the flow of water through a regular array of superhydrophobic pillars were systematically performed to investigate the role of the air-water interface shape on drag reduction. A microfluidic channel was created with circular and superhydrophobic apple-core-shaped pillars bridging across the entire channel. The apple-core-shaped pillars were designed to trap an air pocket along the side of the pillars. The shape of the interface was systematically modified from concave to convex by changing the static pressure within the microchannel. For superhydrophobic pillars having a circular cross section, $D/D_0 = 1.0$, a drag reduction of 7% and a slip velocity of 20% the average channel velocity along the air-water interface were measured. At large static pressures, the interface was driven into the pillars resulting in a decrease in the effective size of the pillars, an increase in the effective spacing between pillars and a pressure drop reduction of as much as 18% when the interface was compressed to $D/D_0 = 0.8$. At low static pressures, the pressure drop increased significantly even as the slip velocity increased as the expanding air-water interface constricted flow through the array of pillars.

¹This research was supported by the National Science Foundation under grant CBET-1334962.

6:27PM L33.00010 Effects of roughness height, pressure and streamwise distance on stress profiles in the inner part of turbulent boundary layer over super-hydrophobic surfaces¹, HANGJIAN LING, JOSEPH KATZ, Johns Hopkins University, SIDDARTH SRINIVASAN, GARETH MCKINLEY, Massachusetts Institute of Technology, KEVIN GOLOVIN, ANISH TUTEJA, University of Michigan, VENKATA PILLUTLA, ABHIJEET., WONJAE CHOI, University of Texas at Dallas — Digital holographic microscopy is used for measuring the mean velocity and stress in the inner part of turbulent boundary layers over sprayed or etched super-hydrophobic surfaces (SHSs). The slip velocity and wall friction are calculated directly from the mean velocity and its gradient along with the Reynolds shear stress at the top of SHSs "roughness". Effects of the normalized rms roughness height k_{rms}^+ , facility pressure p and streamwise distance x from the beginning of SHSs on mean flow are examined. For $k_{rms}^+ < 1$ and $pk_{rms}/\sigma < 1$ (σ is surface tension), the SHSs show 10-28% wall friction reduction, 15-30% slip velocity and $\lambda^+ = 3-10$ slip length. Increasing Reynolds number and/or k_{rms} to establish $k_{rms}^+ > 1$, and increasing p to achieve $pk_{rms}/\sigma > 1$ suppress the drag reduction, as roughness effects and associated near wall Reynolds stress increase. When the roughness effect is not dominant, the measurements agree with previous theoretical predictions of the relationships between drag reduction and slip velocity. The significance of spanwise slip relative to streamwise slip varies with the SHSs texture. Transitions from a smooth wall to a SHS involve overshoot of Reynolds stress and undershoot of viscous stress, trends that diminish with x .

¹Sponsored by ONR

Monday, November 21, 2016 4:30PM - 6:40PM –
Session L34 Turbulence: Compressible Flow Oregon Ballroom 203 - Diego Donzis, Texas AM University

4:30PM L34.00001 Compressibility effects on turbulent mixing¹, JOHN PANICKACHERIL JOHN, DIEGO DONZIS, Texas A&M University — We investigate the effect of compressibility on passive scalar mixing in isotropic turbulence with a focus on the fundamental mechanisms that are responsible for such effects using a large Direct Numerical Simulation (DNS) database. The database includes simulations with Taylor Reynolds number (R_λ) up to 100, turbulent Mach number (M_t) between 0.1 and 0.6 and Schmidt number (Sc) from 0.5 to 1.0. We present several measures of mixing efficiency on different canonical flows to robustly identify compressibility effects. We found that, like shear layers, mixing is reduced as Mach number increases. However, data also reveal a non-monotonic trend with M_t . To assess directly the effect of dilatational motions we also present results with both dilatational and solenoidal forcing. Analysis suggests that a small fraction of dilatational forcing decreases mixing time at higher M_t . Scalar spectra collapse when normalized by Batchelor variables which suggests that a compressive mechanism similar to Batchelor mixing in incompressible flows might be responsible for better mixing at high M_t and with dilatational forcing compared to pure solenoidal mixing. We also present results on scalar budgets, in particular on production and dissipation.

¹Support from NSF is gratefully acknowledged.

4:43PM L34.00002 DNS study on shock/turbulence interaction in homogeneous isotropic turbulence at low turbulent Mach number, KENTO TANAKA, TOMOAKI WATANABE, KOJI NAGATA, AKIHIRO SASOH, YASUHIKO SAKAI, Nagoya Univ, TOSHIYUKI HAYASE, Tohoku Univ, NAGOYA UNIV COLLABORATION — The interaction between homogeneous isotropic turbulence and normal shock wave is investigated by direct numerical simulations (DNSs). In the DNSs, a normal shock wave with a shock Mach number 1.1 passes through homogeneous isotropic turbulence with a low turbulent Mach number and a moderate turbulent Reynolds number. The statistics are calculated conditioned on the distance from the shock wave. The results showed that the shock wave makes length scales related to turbulence small. This effect is significant for the Taylor microscale defined with the velocity derivative orthogonal to the shock wave. The decrease in the Kolmogorov scale is also found. Statistics of velocity derivative are found to be changed by the shock wave propagation. The shock wave causes enstrophy amplification due to the dilatation/vorticity interaction. By this interaction, the vorticity components parallel to the shock wave is more amplified than the normal component. The strain rate is also amplified by the shock wave.

4:56PM L34.00003 Effects of Compressibility on Turbulent Relative Particle Dispersion, BHIMSEN SHIVAMOGGI, University of Central Florida — In this paper, phenomenological developments are used to explore the effects of compressibility on the relative particle dispersion (RPD) in 3D fully-developed turbulence (FDT). The role played by the compressible FDT cascade physics underlying this process is investigated. Compressibility effects are found to lead to reduction of RPD, development of the ballistic regime and particle clustering, corroborating the laboratory experiment and numerical simulation results (Cressman et al., 2004) on the motion of Lagrangian tracers on a surface flow that constitutes a 2D compressible subsystem. These formulations are developed from the scaling relations for compressible FDT and are validated further via an alternative dimensional/scaling development for compressible FDT similar to the one given for incompressible FDT by Batchelor and Townsend (1956). The rationale for spatial intermittency effects is legitimized via the nonlinear scaling dependence of RPD on the kinetic energy dissipation rate.

5:09PM L34.00004 Universality and scaling in compressible turbulence¹, DIEGO DONZIS, SHRIRAM JAGANNATHAN, Texas AM University — A large database of Direct Numerical Simulations (DNS) of stationary compressible isotropic turbulence at a range of Taylor Reynolds numbers ($R_\lambda \approx 38 - 450$) and turbulent Mach numbers ($M_t \approx 0.1 - 0.6$) is used to explore universality. While in incompressible turbulence self-similarity analysis leads to a single scaling parameter (R_λ), compressible turbulence expands the parameter space due to the coupling between hydrodynamics and thermodynamics, and the dependence on the mode of external forcing. While for the former it is common to use M_t as a scaling parameter, the effects of the latter are harder to quantify, and their consequences may have been attributed to a certain lack of universality. For instance, when the dilatational mode is forced, the variance and skewness of pressure shows significant scatter when plotted against M_t . Using a Helmholtz decomposition, we split the velocity field into solenoidal and dilatational modes, and propose scaling parameters that include the contribution from both modes. When expressed against these parameters, we observe a universal scaling regime regardless of the mode of excitation of forcing. Other quantities that follow this behavior are also discussed.

¹Support from NSF and AFOSR is gratefully acknowledged.

5:22PM L34.00005 The role of bulk viscosity on the decay of compressible, homogeneous, isotropic turbulence, ERIC JOHNSEN, SHAOWU PAN, University of Michigan — The practice of neglecting bulk viscosity in studies of compressible turbulence is widespread. While exact for monatomic gases and unlikely to strongly affect the dynamics of fluids whose bulk-to-shear viscosity ratio is small and/or of weakly compressible turbulence, this assumption is not justifiable for compressible, turbulent flows of gases whose bulk viscosity is orders of magnitude larger than their shear viscosities (e.g., CO₂). To understand the mechanisms by which bulk viscosity and the associated phenomena affect compressible turbulence, we conduct DNS of freely decaying compressible, homogeneous, isotropic turbulence for ratios of bulk-to-shear viscosity ranging from 0-1000. Our simulations demonstrate that bulk viscosity increases the decay rate of turbulent kinetic energy; while enstrophy exhibits little sensitivity to bulk viscosity, dilatation is reduced by an order of magnitude within the two eddy turnover time. Via a Helmholtz decomposition of the flow, we determined that bulk viscosity damps the dilatational velocity and reduces dilatational-solenoidal exchanges, as well as pressure-dilatation coupling. In short, bulk viscosity renders compressible turbulence incompressible by reducing energy transfer between translational and internal modes.

5:35PM L34.00006 ABSTRACT WITHDRAWN —

5:48PM L34.00007 Detailed thermodynamic analyses of high-speed compressible turbulence¹, COLIN TOWERY, RYAN DARRAGH, University of Colorado - Boulder, ALEXEI POLUDNENKO, Texas A&M University, PETER HAMILTON, University of Colorado - Boulder — Interactions between high-speed turbulence and flames (or chemical reactions) are important in the dynamics and description of many different combustion phenomena, including autoignition and deflagration-to-detonation transition. The probability of these phenomena to occur depends on the magnitude and spectral content of turbulence fluctuations, which can impact a wide range of science and engineering problems, from the hypersonic scramjet engine to the onset of Type Ia supernovae. In this talk, we present results from new direct numerical simulations (DNS) of homogeneous isotropic turbulence with turbulence Mach numbers ranging from 0.05 to 1.0 and Taylor-scale Reynolds numbers as high as 700. A set of detailed analyses are described in both Eulerian and Lagrangian reference frames in order to assess coherent (structural) and incoherent (stochastic) thermodynamic flow features. These analyses provide direct insights into the thermodynamics of strongly compressible turbulence. Furthermore, presented results provide a non-reacting baseline for future studies of turbulence-chemistry interactions in DNS with complex chemistry mechanisms.

¹This work was supported by the Air Force Office of Scientific Research (AFOSR) under Award No. FA9550-14-1-0273, and the Department of Defense (DoD) High Performance Computing Modernization Program (HPCMP) under a Frontier project award.

6:01PM L34.00008 Turbulent jumps and shock-structure in shock-turbulence interactions using shock-resolving direct numerical simulations¹, CHANG-HSIN CHEN, DIEGO DONZIS, Texas AM Univ — Substantial efforts have been made to understand the canonical interaction between isotropic turbulence and a normal shock. Evidence from theories, experiments and simulations, however, has shown that the interaction is complex and that the outcome is determined not only by mean flow behavior, as suggested by early theories, but also by characteristics of turbulence fluctuations typically quantified by parameters such as the Reynolds (R_λ) and the turbulent Mach number (M_t). An important, yet unresolved, issue is the accurate determination of departures from Rankine-Hugoniot relations due to turbulent fluctuations upstream of the shock. We present an analytic study, based on the quasi-equilibrium assumption, that yield turbulent jumps that depend not only on the mean flow but also on turbulence characteristics. In particular, the focus will be on thermodynamic jumps. Our analytical results agree well with new shock-resolving simulations at a range of Reynolds and Mach numbers. In the context of these results we also present a comparison of previous theory on the dilatation at the shock with the new DNS data. This is further discussed in the context of the transition from wrinkled to broken regimes and the difficulties associated with identifying a shock for very vigorous turbulence.

¹Support from AFOSR is gratefully acknowledged.

6:14PM L34.00009 Spectral Behavior of Weakly Compressible Aero-Optical Distortions¹, EDWIN MATHEWS, KAN WANG, MENG WANG, ERIC JUMPER, University of Notre Dame — In classical theories of optical distortions by atmospheric turbulence, an appropriate and key assumption is that index-of-refraction variations are dominated by fluctuations in temperature and the effects of turbulent pressure fluctuations are negligible. This assumption is, however, not generally valid for aero-optical distortions caused by turbulent flow over an optical aperture, where both temperature and pressures fluctuations may contribute significantly to the index-of-refraction fluctuations. A general expression for weak fluctuations in refractive index is derived using the ideal gas law and Gladstone-Dale relation and applied to describe the spectral behavior of aero-optical distortions. Large-eddy simulations of weakly compressible, temporally evolving shear layers are then used to verify the theoretical results. Computational results support theoretical findings and confirm that if the log slope of the 1-D density spectrum in the inertial range is $-m_\rho$, the optical phase distortion spectral slope is given by $-(m_\rho + 1)$. The value of m_ρ is then shown to be dependent on the ratio of shear-layer free-stream densities and bounded by the spectral slopes of temperature and pressure fluctuations.

¹Supported by HEL-JTO through AFOSR Grant FA9550-13-1-0001 and Blue Waters Graduate Fellowship Program.

6:27PM L34.00010 Predicting the mean fields of compressible turbulent boundary layer via a symmetry approach., WEI-TAO BI, BIN WU, ZHEN-SU SHE, SKLTCS, COE, Peking Univ. — A symmetry approach for canonical wall turbulence is extended to develop mean-field predictions for compressible turbulent boundary layer (CTBL). A stress length and a weighted heat flux length are identified to obey the multilayer dilation symmetry of canonical flows, giving rise to predictions of the mean velocity and temperature profiles for a range of Reynolds number (Re), Mach number (Ma) and wall temperature (Tw). Also predicted are the streamwise developments of the shape factor, the boundary layer edge velocity and the boundary layer thicknesses, etc. Only three parameters are involved in the predictions, which have sound physics and organized behaviors with respect to the Re, Ma and Tw effects. The predictions are extensively validated by direct numerical simulation and experimental data, showing better accuracies than the previous theories. The results provide new quantifications that can be used to assess computations, measurements and turbulence models of CTBL, as well as to provide new insights for the CTBL physics.

Monday, November 21, 2016 4:30PM - 6:27PM –
Session L35 Turbulence: LES Simulations Oregon Ballroom 204 - Onkar Sahni, MANE, RPI

4:30PM L35.00001 Effect of stationary and dynamic transverse squared bars over the turbulent behavior in a channel flow, JESUS RAMIREZ PASTRAN, CARLOS DUQUE-DAZA, Department of Mechanical and Mechatronic Engineering, Universidad Nacional de Colombia, Colombia, OMAR D LOPEZ, Department of Mechanical Engineering, Universidad de los Andes, Colombia — Turbulent flows over rough surfaces are present in different industrial scenarios. Generally, roughness is used to modify the boundary layer behavior, in order to improve heat transfer rates and mixing processes, which is usually accompanied by an increase of skin-friction drag. In the present work two different techniques for modification of the turbulent boundary layer were explored: first, the use of an arrangement of transverse squared bars (synthetic roughness); second, the use of an oscillating movement of the squared bars. In both cases the goal was to assess the increase or decrease of the skin-friction drag and the changes in the turbulent behavior of the flow. Large Eddy Simulations were carried out in order to study a fully developed turbulent channel flow with a smooth upper wall and a synthetically roughed lower wall with a friction Reynolds number around 180. Channel flow over walls with stationary bars and with one of the bars oscillating in the spanwise direction were also considered. Consistency between skin-friction coefficient modification and evolution of Q-structures was observed. Finally, a comparison of changes on some of the TKE terms between smooth surfaces and synthetically rough surfaces allowed to identify the effect of the squared bars for each case.

4:43PM L35.00002 Wall-pressure fluctuations beneath a spatially evolving turbulent boundary layer¹, KRISHNAN MAHESH, PRAVEEN KUMAR, University of Minnesota — Wall-pressure fluctuations beneath a turbulent boundary layer are important in applications dealing with structural deformation and acoustics. Simulations are performed for flat plate and axisymmetric, spatially evolving zero-pressure-gradient turbulent boundary layers at inflow Reynolds number of 1400 and 2200 based on momentum thickness. The simulations generate their own inflow using the recycle-rescale method (*Lund et al., J. of Comput. Phys., 1998, 140 (2): 233-258*). The results for mean velocity and second-order statistics show excellent agreement with the data available in literature. The spectral characteristics of wall-pressure fluctuations and their relation to flow structure will be discussed.

¹This work is supported by ONR.

4:56PM L35.00003 Application of BCM-LES model to flow and pressure fields over urban roughness, TETSURO TAMURA, HIDENORI KAWAI, Tokyo Institute of Technology, RAHUL BALE, KEIJI ONISHI, MAKOTO TSUBOKURA, Riken — BCM (Building Cube Method) enables high-resolution CFD (Computational Fluid Dynamics) simulation by high parallelization performance. This study discusses the applicability of LES (Large Eddy Simulation) based on BCM to prediction of wind velocity and pressure around various building blocks in urban area. First, we validate the computed results of flows past 3D square cylinder in turbulent boundary layer. Fundamental accuracy of the surface pressure distribution on square cylinder is investigated by high-resolution BCM simulation with IBM (Immersed Boundary Method). Next, the BCM is applied to flow simulation of real urban area (Domain size: 25x12km). As a result of this simulation, the development process of urban boundary layer from coastal area to Tokyo central area is examined. Accordingly we show the present numerical model based on BCM-LES can represent sufficiently spatially fine structures and temporally unsteady fluctuations of turbulent flows with good accuracy. It is clarified that the complex pressure distributions acting on the buildings have been also reproduced from the sense of wind-resistance design of buildings in cities

5:09PM L35.00004 The flow past a circular patch of vegetation with a low submergence depth and low solid volume fractions, GOKHAN KIRKIL, Kadir Has University — The effect of the Solid Volume Fraction (SVF) on the flow structure within and past a circular array of surface-mounted cylinders that extends over 75% of the water depth, h is investigated using Detached Eddy Simulation (DES). This set up mimics the case of a submerged patch of rigid vegetation in a channel. The diameter of the cylinders in the array is $d = 0.02D$, where D is the diameter of the circular array. The channel Reynolds number is close to 20,000 and the Reynolds number defined with D is around 24,000. DES is conducted for SVF = 10% and 25%. It is found that as the SVF increases, fairly strong horseshoe vortex system forms around the upstream face of the vegetation patch, the strength of the separated shear layers on the sides of the vegetation patch increases and the length of the recirculation region behind the patch decreases. While an increase of the SVF results in a large increase of the turbulent kinetic energy in the wake, the opposite is observed within the porous vegetation patch.

5:22PM L35.00005 Fluid-structure interaction of turbulent boundary layer over a compliant surface¹, SREEVATSA ANANTHARAMU, KRISHNAN MAHESH, University of Minnesota — Turbulent flows induce unsteady loads on surfaces in contact with them, which affect material stresses, surface vibrations and far-field acoustics. We are developing a numerical methodology to study the coupled interaction of a turbulent boundary layer with the underlying surface. The surface is modeled as a linear elastic solid, while the fluid follows the spatially filtered incompressible Navier-Stokes equations. An incompressible Large Eddy Simulation finite volume flow approach based on the algorithm of Mahesh et al. (*JCP* 197.1 (2004): 215-240) is used in the fluid domain. The discrete kinetic energy conserving property of the method ensures robustness at high Reynolds number. The linear elastic model in the solid domain is integrated in space using finite element method and in time using the Newmark time integration method. The fluid and solid domain solvers are coupled using both weak and strong coupling methods. Details of the algorithm, validation, and relevant results will be presented.

¹This work is supported by NSWCCD, ONR

5:35PM L35.00006 Large Eddy Simulation for Oscillating Airfoils with Large Pitching and Surging Motions, ONKAR SAHNI, REED CUMMING, STEVEN TRAN, ALEXANDER KOCHER, MANE, RPI — Many applications of interest involve unsteady aerodynamics due to time varying flow conditions (e.g. in the case of flapping wings, rotorcrafts and wind turbines). In this study, we formulate and apply large eddy simulation (LES) to investigate flow over airfoils at a moderate mean angle of attack with large pitching and surging motions. Current LES methodology entails three features: i) a combined subgrid scale model in the context of stabilized finite element methods, ii) local variational Germano identity (VGI) along with Lagrangian averaging, and iii) arbitrary Lagrangian-Eulerian (ALE) description over deforming unstructured meshes. Several cases are considered with different types of motions including surge only, pitch only and a combination of the two. The flow structures from these cases are analyzed and the numerical results are compared to experimental data when available.

5:48PM L35.00007 Large eddy simulation of tip-leakage flow in an axial flow fan¹, KEUNTAE PARK, HAECHON CHOI, Seoul Natl Univ, SEOKHO CHOI, YONGCHEOL SA, LG Electronics, OH-KYOUNG KWON, Korea Institute of Science and Technology Information — An axial flow fan with a shroud generates a complicated tip-leakage flow by the interaction of the axial flow with the fan blades and shroud near the blade tips. In this study, large eddy simulation is performed for tip-leakage flow in a forward-swept axial flow fan inside an outdoor unit of an air-conditioner, operating at the design condition of the Reynolds number of 547,000 based on the radius of blade tip and the tip velocity. A dynamic global model (Lee et al., 2010, PoF) is used for a subgrid-scale model, and an immersed boundary method in a non-inertial reference frame (Kim & Choi, 2006, JCP) is adopted. The present simulation clearly reveals the generation and evolution of tip-leakage vortex near the blade tip by the leakage flow. At the inception of the leakage vortex near the leading edge of the suction-side of the blade tip, the leakage vortex is composed of unsteady multiple vortices containing high-frequency fluctuations. As the leakage vortex develops downstream along a slant line toward the following blade, large and meandering movements of the leakage vortex are observed. Thus low-frequency broad peaks of velocity and pressure occur near the pressure surface.

¹Supported by the KISTI Supercomputing Center (KSC-2016-C3-0027)

6:01PM L35.00008 Comparative Study of Reynolds Averaged and Embedded Large Eddy Simulations of a High Pressure Turbine Stage, SAM JONES, ALEKSANDAR JEMCOV, THOMAS CORKE¹, University of Notre Dame — An Embedded Large Eddy Simulation (ELES) approach is used to simulate the flow path through a high pressure turbine stage that includes the entry duct, stationary inlet and exit guide vanes, and a rotor. The flowfield around the rotor is simulated using LES. A Reynolds Averaged Simulation (RAS) is used for the rest of the flow domain. The interface between RAS and LES domains uses the RAS turbulence quantities as a means of obtaining length scales that are used in computing the vorticity required to trigger a proper energy cascade within the LES part of the flow field. The objective is to resolve the unsteady vortical motions that emanate from the gap between the rotor tip and duct walls that are presumably under-resolved in a RAS approach. A comparative analysis between RAS and ELES approaches for this turbomachinery problem is then presented.

¹APS Fellow

6:14PM L35.00009 Large-eddy simulation of propeller wake at design operating conditions¹, PRAVEEN KUMAR, KRISHNAN MAHESH, University of Minnesota — Understanding the propeller wake is crucial for efficient design and optimized performance. The dynamics of the propeller wake are also central to physical phenomena such as cavitation and acoustics. Large-eddy simulation is used to study the evolution of the wake of a five-bladed marine propeller from near to far field at design operating condition. The computed mean loads and phase-averaged flow field show good agreement with experiments. The propeller wake consisting of tip and hub vortices undergoes streamtube contraction, which is followed by the onset of instabilities as evident from the oscillations of the tip vortices. Simulation results reveal a mutual induction mechanism of instability where instead of the tip vortices interacting among themselves, they interact with the smaller vortices generated by the roll-up of the blade trailing edge wake in the near wake. Phase-averaged and ensemble-averaged flow fields are analyzed to explain the flow physics.

¹This work is supported by ONR.

Monday, November 21, 2016 4:30PM - 6:27PM – Session L36 Drops: Particle laden And Drops: Wall-bound w/ Transport Portland Ballroom 251 - Michael Miksis, Northwestern University

4:30PM L36.00001 Simulations of electrically induced particle structuring on spherical drop surface, YI HU, Northwestern Univ, PETIA VLAHOVSKA, Brown University, MICHAEL MIKSIS, Northwestern Univ — Recent experiments (Ourjumi 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 electrical field intensity, particles organize into an equatorial belt, pole-to-pole chains, or dynamic vortices. Here we present a model to simulate the collective particle dynamics, which accounts for the electrohydrodynamic flow and particle dielectrophoresis due to the non-uniformity of local electrical field. In stronger electric fields, particles are expected to undergo Quincke rotation, inducing rotating clusters through inter-particle hydrodynamical interaction. We discuss how the field intensity influences the width, orientation and periodicity of the particle clusters. Our results provide insight into the various particle assemblies discovered in the experiments.

4:43PM L36.00002 Snap-in of particles at curved liquid interfaces, CHAO LI, MOMENE

MORADIAFRAPOLI, JEREMY MARSTON, Texas Tech University — The contact of particles with liquid interfaces constitutes the first stage in the formation of a particle-laden interface, the so-called "snap-in effect". Here, we report on an experimental study using high-speed video to directly visualize the snap-in process and the approach to the equilibrium state of a particle at a curved liquid interface (i.e. droplet surface). We image the evolution of the contact line, which is found to follow a power-law scaling in time, and the dynamic contact angle during the snap-in. Both hydrophilic and hydrophobic particles are explored and we match the lift-off stage of the particles with a simple force balance. We also explore some multi-particle experiments, eluding to the dynamics of particle-laden interface formation.

4:56PM L36.00003 Irreversible deformation of particle-coated droplets, MOMENE MORA-

DIAFRAPOLI, JEREMY MARSTON, Texas Tech University — In this presentation, we will explore one particular property of particle-coated droplets, known as liquid marbles - namely - irreversible deformation. Whilst some studies have eluded to certain mechanical properties of particle-laden interfaces such as interfacial rheology and elasticity, the extent of deformation they can sustain has not been well-characterized. We explore this rich area using liquid marbles as the model system for the study. We find that they can withstand a compression of up to approximately 40% without rupturing, but even before this threshold, the deformation can be irreversible. We quantify and attempt to explain this feature for a range of particle sizes. Our observations thus add another layer of hidden complexity and non-linearity to seemingly elemental interfacial structures.

5:09PM L36.00004 Dynamics of Droplet Detachment from a Granular Raft, SUZIE

PROTIERE, CNRS-Institut Jean le Rond d'Alembert, MATTHIEU ROCHE, CNRS-Laboratoire MSC — When we sprinkle dense particles at an oil/water interface these particles self-assemble due to long-range capillary interactions into a monolayer that we call a granular raft. Particles can progressively be added to the raft until it destabilizes due to the balance between the local buoyancy forces and the capillary forces at the border of the raft. When the raft destabilizes it sinks and forms oil-in-water armored droplets. We study the formation of such armored droplets and compare its detachment to the behavior observed in suspensions or in pure viscous fluids. Indeed for pure fluids the radius of the neck of the forming droplet decays linearly. Here, we find that depending on the size and on the density of the particles two types of behaviors are observed during droplet formation. Either the raft sinks and no particles are found along the neck during the armored droplet formation, or an "interfacial granular jet" forms which breaks, due to a Rayleigh-Plateau-like instability, into a multitude of small millimeter-sized armored droplets. We show that since the particles are adsorbed at the interface, those two types of behaviors depend on a dimensionless parameter that takes into account the particle size and density. Moreover we find that the position of the particles during the formation of the drop dramatically modifies the dynamics, proving that the initial conditions are important during droplet breakup.

5:22PM L36.00005 Lattice Boltzmann Simulations of Evaporating Droplets with Nanoparticles, MINGFEI ZHAO, XIN YONG, Binghamton University — Elucidating the nanoparticle dynamics in drying droplets

provides fundamental hydrodynamic insight into the evaporation-induced self-assembly, which is of great importance to materials printing and thin film processing. We develop a free-energy-based multiphase lattice Boltzmann model coupled with Lagrangian particle tracking to simulate evaporating particle-laden droplets on a solid substrate with specified wetting behavior. This work focuses on the interplay between the evaporation-driven advection and the self-organization of nanoparticles inside the droplet and at the droplet surface. For static droplets, the different parameters, fluid-particle interaction strength and particle number, governing the nanoparticle-droplet dynamics are systematically investigated, such as particle radial and circumferential distribution. We clarify the effect of nanoparticle presence on the droplet surface tension and wetting behavior. For evaporating droplets, we observe how droplet evaporation modulates the self-assembly of nanoparticles when the droplet has different static contact angles and hysteresis windows. We also confirm that the number of nanoparticles at the liquid-vapor interface influences the evaporation flux at the liquid-vapor interface.

5:35PM L36.00006 Colloidal Deposition of Ellipsoidal Particles: Competition between Capillary and Hydrodynamic Forces, DONG-OOK KIM, MIN PACK, Drexel University, HYOUNGSOO

KIM, Princeton University, YING SUN, Drexel University — Ellipsoidal particles have previously been shown to suppress the coffee-ring effect in millimeter-size colloidal droplets. Compared to their spherical counterparts, ellipsoidal particles experience stronger adsorption energy to the drop surface where the anisotropy-induced liquid-air interface deformation leads to much greater capillary attractions between particles. Using inkjet-printed colloidal drops of varying drop size, particle concentration, and particle aspect ratio, the present work demonstrates how the suppression of the coffee-ring is not only a function of the particle anisotropy, but rather a competition between the propensity for particles to assemble at the drop surface via capillary interactions and the evaporation-driven particle motion to the contact line. For ellipsoidal particles on the drop surface, the capillary force increases with particle concentration and aspect ratio, while the hydrodynamic force increases with aspect ratio but decreases with drop size. When the capillary force dominates, the surface ellipsoids form a coherent network inhibiting advection and the coffee-ring effect is suppressed, whereas when the hydrodynamic force dominates, the ellipsoids move to the contact line resulting in coffee-ring deposition.

5:48PM L36.00007 Targeting Sessile Droplets with Electrospray to Form Nanoparticle Deposits¹, PAUL CHIAROT, MATTHIAS DAEUMER, SEPEHR MAKHTABI, XIN YONG, State University of New York at Binghamton — The ability to print ordered deposits of nanoparticles has significant implications for electronics and photonics manufacturing.

In this work, electrospray atomization was used to deliver dry nanoparticles to the surface of sacrificial sessile droplets. The particles were subsequently mapped to a glass substrate upon complete evaporation of the target droplet to create a deposit. The influence of the key electrospray operating parameters on the final deposit structure were explored, including: spray time, nanoparticle concentration, and initial sacrificial droplet volume. Once the nanoparticles were delivered to the interface, evaporatively-driven transport of the particles across the surface of the sessile droplet played a significant role in determining the structure of the deposit. When the contact line of the target sessile droplet was pinned during evaporation, the final deposit had greater particle density at the edge and center. The particles were distributed more uniformly across the deposit when the contact line of the target droplet moved during evaporation. The influence of thermal gradients on the final deposit structure was investigated by heating the substrate to increase the sessile droplet temperature. We also conducted computational simulations of evaporating particle-laden droplets and explored the influences of contact line behavior and nanoparticle surface chemistry on the deposit structure.

¹This research supported by the National Science Foundation (Award 1538090).

6:01PM L36.00008 Controlling the Growth Modes of Femtoliter Sessile Droplets Nucleating on Chemically Patterned Surfaces, XUEHUA ZHANG, LEI BAO, ZENON WERBIUK, Royal Melbourne Institute of Technology, DETLEF LOHSE, University of Twente — Femtoliter droplet arrays on immersed substrates are essential elements in a broad range of advanced droplet-based technologies, such as light manipulation, sensing, and high throughput diagnosis. Solvent exchange is a bottom-up approach for producing those droplets from a pulse of oil oversaturation when a good solvent of the droplet liquid is displaced by a poor solvent. The position and arrangement of the droplets are regulated by chemical micropatterns on the substrate. Here we show experimentally and theoretically that the growth modes of droplets confined in planar micropatterns on the surface can be manipulated through the laminar flow of the solvent exchange. The control parameters are the area size of the micropatterns and the flow rate, and the observables are the contact angle and the final droplet volume. For a given pattern size, the Peclet number of the flow determines whether the growing droplets switch from an initial constant contact angle mode to a subsequent constant contact radius mode. Good agreement is achieved between the experimental results and our theoretical model that describes the dependence of the final droplet size on Pe.

6:14PM L36.00009 Dissolution of Droplets on a Substrate with Engraved Concentric Rings, JOSE MANUEL ENCARNACION ESCOBAR, ERIK DIETRICH, PENGYU LV, HAROLD ZANDVLIET, XUEHUA ZHANG, Physic of Fluids Group, University of Twente, STEVE ARSCOTT, Institut d'Electronique, de Microelectronique et de Nanotechnologie of the University of Lille, DETLEF LOHSE, Physic of Fluids Group, University of Twente, UNIVERSITY OF TWENTE TEAM, UNIVERSITY OF LILLE TEAM, MCEC TEAM — The nucleation of nano and micro sized drops and bubbles often occurs on catalytic surfaces lowering its efficiency. The contact angle hysteresis, which is a consequence of the pinning on heterogeneities of the surface, can dramatically affect the stability and lifetime of the drop. The stability of a surface bubble can, in fact, be theoretically calculated thanks to the assumption of the pinning of the bubble [Lohse and Zhang, Lohse, D.; Zhang, X., Phys. Rev. E 2015, 91, 031003]. Our experiments try to shed light on the understanding of the pinning of droplets caused by micro structures during their dissolution. It is possible to predict the depinning angle of a drying drop as a function of the geometry of the defect and the receding contact angle. Additionally, the jump from one defect to another happens fast but is not an immediate change. This dewetting happens showing the so called zipping behavior. We present quantitative data from experiments as well as the experimental techniques used, including confocal microscopy and the first analysis and comparison with the already existent theoretical models.

Monday, November 21, 2016 4:30PM - 6:40PM –

Session L37 Drops: More Impacts Portland Ballroom 252 - Michael Rother, University of Minnesota

4:30PM L37.00001 Gravitational Interactions of Slightly Deformable Drops in a Vertical Temperature Gradient, JOHN STARK, MICHAEL ROTHER, University of Minnesota Duluth — For the case of low Reynolds and Marangoni numbers, collision efficiencies are calculated for two interacting, slightly deformable drops moving due to combined gravitational and thermocapillary driving forces. The solution technique employs ideas borrowed from matched asymptotic expansions. Also, as separate solutions are required for both the outer region, when there is a large separation between the drops, and the inner region, when the drops are in apparent contact, some investigation is made into the approach for the matching or transition region. Using bispherical coordinates to determine the mobility functions along the drops' line of centers, the outer region solution yields the contact force for the inner region solution. The inner region solution utilizes the thin-film equations for drops with fully mobile interfaces. Van der Waals forces are neglected in the outer region but become important in the inner region. This work has possible applications in materials processing and low gravity operations.

4:43PM L37.00002 Modeling Oblique Impact Dynamics of Particle-Laden Nanodroplets, XIN YONG, SHIYI QIN, State University of New York at Binghamton — A fundamental understanding of the impact dynamics of nanoscopic droplets laden with nanoparticles has important implications for materials printing and thin film processing. Using many-body dissipative particle dynamics (MDPD), we model nanometer sized suspension droplets impinging on dry solid substrate with oblique angles, and compare their behavior with pure liquid droplets. Equilibrated floating droplets containing two types of nanoparticles, namely fully-wetted hydrophilic particles and surface-active Janus particles, impact onto the solid surface with varying initial velocities and impact angles. The velocity components in the normal and tangential directions to the substrate defines normal and tangential Reynolds and Weber numbers, which are used to classify impact regimes. Droplets with nanoparticles dispersed in the bulk and covering the droplet surface (resembling liquid marbles) exhibit quite different behavior in the course of impact. We also reveal the influences of substrate wettability and its interaction with nanoparticles on the impact dynamics. In addition, the vapor film beneath an impinging droplet shows no significant effect on the impact dynamics in our MDPD simulations.

4:56PM L37.00003 Air cushioning vs liquid jets in drop impact, CHRISTOPHE JOSSERAND, PASCAL RAY, CNRS, Institut D'Alembert, STEPHANE ZALESKI, UPMC, Institut D'Alembert, INSTITUT D'ALEMBERT TEAM — Drop impact on a liquid thin film is investigated numerically, focusing on the interaction between the air cushioning and the splashing dynamics. We show that a new dimensionless number, balancing the time scale of the lubrication dynamics and that of the jet formation is at the heart of the different mechanisms at play.

5:09PM L37.00004 When sticky fluids don't stick: yield-stress fluid drops on heated surfaces, BRENDAN BLACKWELL, ALEX WU, RANDY EWOLDT, University of Illinois at Urbana-Champaign — 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; this sticky behavior motivates several applications of these rheologically-complex materials. Here we describe experiments with aqueous yield stress fluids that are more 'sticky' than water at room temperature (e.g. supporting larger coating thicknesses), but are less 'sticky' at higher temperatures. Specifically, we study the conditions for aqueous yield stress fluids to bounce and slide on heated surfaces when water sticks. Here we present high-speed imaging and color interferometry to observe the thickness of the vapor layer between the drop and the surface during both stick and non-stick events. We use these data to gain insight into the physics behind the phenomenon of the yield-stress fluids bouncing and sliding, rather than sticking, on hot surfaces.

5:22PM L37.00005 Enhanced droplet retention through in-situ precipitation, MAHER DAMAK, SEYED REZA MAHMOUDI, MD NASIM HYDER, KRIPA VARANASI, Massachusetts Inst of Tech-MIT — Poor retention of agricultural sprays on hydrophobic plants is an important issue, as large quantities of toxic chemicals end up in soils and groundwater after sprayed droplets bounce off leaves. Here we propose to increase liquid retention on hydrophobic surfaces by in-situ formation of hydrophilic surface defects that pin the impacting drops. Defects are formed through simultaneous spraying of solutions containing opposite polyelectrolyte, which combine on the surface and precipitate. We study individual drop-on-drop impact dynamics with high-speed imaging and analyze the surface after impact. Using these results, we elucidate the mechanism of precipitate formation and droplet retention. We derive a physical model to estimate the energy dissipation by the formed defects and predict the transition from bouncing to sticking, which can be used to design effective sprays. We finally show large macroscopic enhancements in retention of sprays on superhydrophobic synthetic surfaces as well as leaves.

5:35PM L37.00006 Impact of ultra-viscous drops on a smooth solid surface, KENNETH LANGLEY, E. Q. LI, S. T. THORODDSEN, King Abdullah Univ of Sci & Tech (KAUST) — As an impacting drop approaches a solid surface, the gas layer between the drop and surface must be pushed aside. The lubrication pressure in this gas layer is sufficient to deform the droplet. For low-viscosity drops (~ 1 cSt) a kink develops at the edge of the deformation, which results in contact being made along a ring, entraining a disc of air inside the drop. At higher viscosities, the kink is less pronounced due to the viscous stresses allowing the drop to glide on a thin layer of air (~ 150 nm) for an extended time. When the thin air layer ruptures, numerous contacts are made that grow substantially faster than the predicted capillary-viscous balance. The evolution of the air layer and the subsequent growth of the contacts are investigated experimentally using two-color interferometry and high-speed imaging for a 7 orders of magnitude range of drop viscosities.

5:48PM L37.00007 Liquid repellency by a moving plate, AMBRE BOUILLANT, Ecole polytechnique (France), ANAIS GAUTHIER, CHRISTOPHE CLANET, DAVID QUERE, LadHyX, Ecole Polytechnique - PMMH, ESPCI — Moving solids can repel impacting drops, owing to their motion. Provided the solid velocity is larger than a threshold value, air entrained at the vicinity of the moving plate prevents the drop from wetting, and makes it bounce. In addition, the rebound is oblique, which enhances the evacuation of liquid. We discuss experiments and models on this theme, and extend them to case of small droplets (such as formed in a spray) found to be even more efficiently repelled by the moving plate.

6:01PM L37.00008 Numerical simulation of drop impact on a controlled falling liquid film¹, ZHIZHAO CHE, Tianjin University, China, IDRIS ADEBAYO, ZHIHUA XIE, DIMITRIOS PAVLIDIS, PABLO SALINAS, OMAR MATAR, Imperial College London — We study the impact process of droplets falling obliquely on controlled films using a numerical simulation approach. This approach is based on a finite element discretisation of the Navier Stokes equations on fully unstructured anisotropic and adaptive meshes, which are capable of representing the underlying physics of multiphase problems accurately while also reducing computational effort. Liquid film control here is applied to ensure that droplet impact occurs on different, targeted regions of a controlled film surface viz. capillary waves preceding a large-amplitude wave, flat film regions, and wave humps. The outcomes of droplet impact on these different regions are then compared and the differences discussed. The effect of varying the film flow rate, droplet speed, and droplet size on a number of droplet impact outcomes is also studied and the results further compared with those from uncontrolled as well as quiescent liquid films.

¹EPSRC UK Programme Grant MEMPHIS (EP/K003976/1)

6:14PM L37.00009 Big Hydrophobic Capillary Fluidics; Basically Water Ping Pong in Space¹, MARK WEISLOGEL, BABAK ATTARI, ANDREW WOLLMAN, KARL CARDIN, JOHN GEILE, THOMAS LINDNER, Portland State University — Capillary surfaces can be enormous in environments where the effects of gravity are small. In this presentation we review a number of interesting examples from demonstrative experiments performed in drop towers and aboard the International Space Station. The topic then focuses on large length scale hydrophobic phenomena including puddle jumping, spontaneous particle ejections, and large drop rebounds akin to water ping pong in space. Unseen footage of NASA Astronaut Scott Kelly playing water ping pong in space will be shown. Quantitative and qualitative results are offered to assist in the design of experiments for ongoing research.

¹NASA NNX12A047A

6:27PM L37.00010 ABSTRACT WITHDRAWN —

Monday, November 21, 2016 4:30PM - 6:40PM —
Session L38 Flow Instability: Jets, Drops, and Bubbles Portland Ballroom 255 - Mickael Bourgoin, LEGI/CNRS

4:30PM L38.00001 Numerical study of three-dimensional liquid jet breakup with adaptive unstructured meshes, ZHIHUA XIE, DIMITRIOS PAVLIDIS, PABLO SALINAS, CHRISTOPHER PAIN, OMAR MATAR, Imperial College London — Liquid jet breakup is an important fundamental multiphase flow, often found in many industrial engineering applications. The breakup process is very complex, involving jets, liquid films, ligaments, and small droplets, featuring tremendous complexity in interfacial topology and a large range of spatial scales. The objective of this study is to investigate the fluid dynamics of three-dimensional liquid jet breakup problems, such as liquid jet primary breakup and gas-sheared liquid jet breakup. An adaptive unstructured mesh modelling framework is employed here, which can modify and adapt unstructured meshes to optimally 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. Numerical examples of some benchmark tests and the dynamics of liquid jet breakup with and without ambient gas are presented to demonstrate the capability of this method.

4:43PM L38.00002 Bubble Impact with a Solid Wall, VISHRUT GARG, SUMEET THETE, OSMAN

BASARAN, Purdue University — In diverse natural and industrial processes, and in particular in process equipment widely used in oil and gas production, bubbles and drops that are immersed in a continuous liquid phase frequently collide with solid walls. In this talk, the impact with a solid wall of a gas bubble that is surrounded by a liquid that is either a Newtonian or a non-Newtonian fluid is analyzed by numerical simulation. Special attention is paid to the thin film that forms between the approaching bubble and the solid wall. Flow regimes that arise as the film thickness decreases are scrutinized and rationalized by comparison of the computational predictions to well-known and new analytical results from lubrication theory based thin film literature. Finally, flow transitions that occur as the lubrication theory breaks down and inertia becomes significant are investigated.

4:56PM L38.00003 Explicit demonstration of the role of Marangoni effect in the breakup of nanoscale liquid filaments¹, IVANA SERIC, New Jersey Institute of Technology, KYLE MAHADY, University of Tennessee, SHAHRIAR AFKHAMI, New Jersey Institute of Technology, CHRIS HARTNETT, JASON FOWLKES, PHILIP RACK, University of Tennessee, LOU KONDIC, New Jersey Institute of Technology — We consider a breakup of bi-metal filaments deposited on a solid substrate.

These filaments are exposed to laser irradiation and, while in the liquid phase, evolve by a process resembling breakup of a liquid jet governed by the Rayleigh-Plateau instability. The novel element is that the Marangoni effect, resulting from a different surface tension of the two metals from which the filament is built, is crucial in understanding the instability development. In particular, Marangoni effect may lead to the inversion of the breakup process, producing droplets at the locations where according to the Rayleigh-Plateau theory dry spots would be expected. We present experimental results carried out with Cu-Ni filaments, as well as direct numerical simulations based on a novel algorithm that includes variable surface tension in a Volume-of-Fluid based Navier-Stokes solver. These results suggest the possibility of using Marangoni effect for the purpose of self- and directed-assembly on the nanoscale.

¹Supported by the NSF grant No. CBET-1604351

5:09PM L38.00004 Three dimensional direct numerical simulation of complex jet flows¹, SEUNGWON SHIN, Hongik University, South Korea, LYES KAHOUADJI, Imperial College London, DAMIR JURIC, JALEL CHERGUI, LIMSI, CNRS, France, RICHARD CRASTER, OMAR MATAR, Imperial College London — We present three-dimensional simulations of two types of very challenging jet flow configurations. The first consists of a liquid jet surrounded by a faster coaxial air flow and the second consists of a global rotational motion. These computations require a high spatial resolution and are performed with a newly developed high performance parallel code, called BLUE, for the simulation of two-phase, multi-physics and multi-scale incompressible flows, tested on up to 131072 threads with excellent scalability performance. The method for the treatment of the fluid interfaces uses a hybrid Front Tracking/Level Set technique that defines the interface both by a discontinuous density field as well as by a local triangular Lagrangian mesh. Coriolis forces are taken into account and solved via an exact time-integration method that ensures numerical accuracy and stability.

¹EPSRC UK Programme Grant EP/K003976/1

5:22PM L38.00005 Numerical simulations of vibrating sessile drop¹, LYES KAHOUADJI,

Imperial College London, JALEL CHERGUI, DAMIR JURIC, LIMSI, CNRS, France, SEUNGWON SHIN, Hongik University, Republic of Korea, RICHARD CRASTER, OMAR MATAR, Imperial College London — A vibrated drop constitutes a very rich physical system, blending both interfacial and volume phenomena. A remarkable experimental study was performed by M. Costalanga (PhD. Universit Paris Diderot, 2015) highlighting sessile drop motion subject to horizontal, vertical and oblique vibration. Several intriguing phenomena are observed such as drop walking and rapid droplet ejection. We perform three-dimensional direct numerical simulations of vibrating sessile drops where the phenomena described above are computed using the massively parallel multiphase code BLUE.

¹EPSRC UK Programme Grant MEMPHIS (EP/K003976/1)

5:35PM L38.00006 Structure build-up and evolution in the drying of sessile blood droplets¹, RICHARD CRASTER, ARAN UPPAL, OMAR MATAR, Imperial College London — Experimental observations have recorded blood undergoing a sol-gel transition during the evaporation process. Consequently, the rheology becomes non-uniform throughout the droplet and exhibits transitional complex phenomena that we must capture if we wish to accurately model the evaporative/cracking process. We propose a model where thixotropy is introduced to capture the evolving rheology as evaporation occurs. Thixotropy is often used to describe fluids which exhibit a decrease in viscosity due to flow and subsequent slow recovery of viscosity after the cessation of the flow. We introduce an additional parameter to describe the internal structure of the fluid at each point and consider a droplet in the limit of the lubrication approximation. We present a discussion of our results that demonstrates the dependence of structure build-up, which accompanies the spatio-temporal evolution of the drop, on system parameters.

¹EPSRC UK Centre for Doctoral Training

5:48PM L38.00007 Formation of surfactant-laden drops: comparison of experimental and numerical results¹, NINA KOVALCHUK, University of Birmingham, LYES KAHOUADJI, Imperial College London, MARK SIMMONS, University of Birmingham, RICHARD CRASTER, OMAR MATAR, Imperial College London, DAMIR JURIC, JALEL CHERGUI, LIMSI, CNRS, France, SEUNGWON SHIN, Hongkin University, South Korea — Drop formation is ubiquitous in many industrial processes, with surfactants being commonly used to stabilise drops. Thus, understanding the regularities of drop formation and accompanying processes, such as formation of satellite droplets in the presence of surfactant is of high importance. Here we present the results of a comparative experimental and numerical study on formation of surfactant-laden drops over a range of flow rates and surfactant concentrations. The precise parameters of the surface tension isotherm for surfactants used in the experimental study are implemented in the numerical code enabling quantitative comparison between the two approaches. It is shown that the effect of surfactant depends not only on concentration, but also on the value of critical micellar concentration (cmc). The transition to the regime where satellite droplets are no longer released was observed when the flow rate exceeded a threshold value depending on surfactant concentration and cmc value.

¹EPSRC UK Programme Grant MEMPHIS (EP/K003976/1)

6:01PM L38.00008 Three-dimensional simulations of thin ferro-fluid films and drops in magnetic fields¹, DEVIN 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, bounded 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, velocity, and magnetic fields inside the film. The potential in the gas phase is solved with 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, and the magnetic field. We consider the three-dimensional evolution of the film to spawise perturbations by solving the non-linear equations numerically. The constant flux configuration is considered, which corresponds to a thin film and drop flowing down an incline, and a parametric study is performed to understand the effect of a magnetic field on the stability and structure of the formed drops.

¹EPSRC UK platform grant MACIPh (EP/L020564/1) and programme Grant MEMPHIS (EP/K003976/1)

6:14PM L38.00009 Evolution of the dynamic Rayleigh-Plateau instability on liquid jets¹, FABIAN DENNER, FABIEN EVRARD, BEREND VAN WACHEM, Imperial College London, ALFONSO ARTURO CASTREJON-PITA, University of Oxford, JOSE RAFAEL CASTREJON-PITA, Queen Mary University of London — The Rayleigh-Plateau instability (RPI) is the dominating mechanism leading to the breakup of surface-tension-dominated liquid jets. Although linear stability analysis has proven to be a powerful tool to study the evolution of the RPI for (quasi-)static liquid jets and filaments, in typical practical applications (e.g. inkjet printing) the inertia of liquid jets is significant, giving rise to nonlinear effects that influence the spatiotemporal evolution of the RPI and which are not captured by linear stability analysis. Using direct numerical simulation and laboratory experiments, we study the evolution of the dynamic RPI on liquid jets with different Weber and Ohnesorge numbers as well as different velocity profiles, perturbation amplitudes and wavenumbers. Our results show how inertia as well as the amplitude/wavenumber of the perturbation change the velocity and pressure fields of the liquid jet, which changes the spatiotemporal growth of the dynamic RPI and, consequently, the breakup length of the jet, with a local reversal of the RPI under certain conditions. We identify the key mechanisms that govern the complex evolution of the dynamic RPI and highlight the main differences between static and dynamic RPI.

¹Financial support from the EPSRC (grant EP/M021556/1), from Petrobras, from the John Fell Oxford University Press Research Fund and from the Royal Society is gratefully acknowledged.

6:27PM L38.00010 Experimental study of splashing mechanisms by an immersed rotating body, MICKAEL BOURGOIN, Laboratoire de Physique, University of Lyon, Ecole Normale Suprieure de Lyon, CNRS, DIEGO RODRIGUEZ, SAKSHAM GAKHAR, LEGI, Universit Grenoble Alpes, France, JEAN-PHILIPPE MATAS, LMFA, University of Lyon, France, REMI BERGER, PSA Peugeot Citroen, Velizy, France — We study the entrainment of water by a rotating wheel, as a function of the rotation frequency, wheel radius and depth of immersion. The entrainment leads to the formation of a liquid sheet on the ascending side, and of ligaments on the descending side. These structures are captured via simple models. We measure via image processing the liquid flux ejected by the wheel.

Monday, November 21, 2016 4:30PM - 6:40PM –
Session L39 Bio: Modeling of Microswimmers Portland Ballroom 256 - Enkeleida Lushi, Brown University

4:30PM L39.00001 On the interactions of micro-swimmers with surfaces, ENKELEIDA LUSHI, Brown University — Solid boundaries alter both motion and spatial distribution of microorganisms in ways that are currently not completely understood. We present novel micro-swimmer models and simulations able to display correct features seen in experiments such as bacteria circling near surfaces or micro-algae scattering from them. For pushers like bacteria we show that the correct flow singularity is more complex than a force dipole. For bi-flagellates like micro-algae we show that their behavior at surfaces results from a nuanced interplay of flagellar contact, hydrodynamics, noise and cell spinning, with the swimmer geometry being a crucial component. Our results compare well with the most recent experimental data and suggest ways of designing multi-swimmer simulations that capture the correct physics.

4:43PM L39.00002 An elastic two-sphere swimmer in Stokes flow, BABAK NASOURI, GWYNN ELFING, University of British Columbia — Swimming at low Reynolds number in Newtonian fluids is only possible through non-reciprocal body deformations due to the kinematic reversibility of the Stokes equations. We consider here a model swimmer consisting of two linked spheres, wherein one sphere is rigid and the other an incompressible neo-Hookean solid. The two spheres are connected by a rod which changes its length periodically. We show that the deformations of the body are non-reciprocal despite the reversible actuation and hence, the elastic two-sphere swimmer propels forward. Our results indicate that even weak elastic deformations of a body can qualitatively alter swimming dynamics and should not be neglected in analyzing swimming in Stokes flows.

4:56PM L39.00003 Optimal control, optimization and asymptotic analysis of Purcell's microswimmer model, OREN WIEZEL, YIZHAR OR, Technion, Israel Institute of Technology — Purcell's swimmer (1977) is a classic model of a three-link microswimmer that moves by performing periodic shape changes. Becker et al. (2003) showed that the swimmer's direction of net motion is reversed upon increasing the stroke amplitude of joint angles. Tam and Hosoi (2007) used numerical optimization in order to find optimal gaits for maximizing either net displacement or Lighthill's energetic efficiency. In our work, we analytically derive leading-order expressions as well as next-order corrections for both net displacement and energetic efficiency of Purcell's microswimmer. Using these expressions enables us to explicitly show the reversal in direction of motion, as well as obtaining an estimate for the optimal stroke amplitude. We also find the optimal swimmer's geometry for maximizing either displacement or energetic efficiency. Additionally, the gait optimization problem is revisited and analytically formulated as an optimal control system with only two state variables, which can be solved using Pontryagin's maximum principle. It can be shown that the optimal solution must follow a "singular arc". Numerical solution of the boundary value problem is obtained, which exactly reproduces Tam and Hosoi's optimal gait.

5:09PM L39.00004 The stresslet induced by active swimmers , SEBASTIEN MICHELIN, LadHyX

- Ecole Polytechnique, ERIC LAUGA, DAMTP - University of Cambridge — Active particles such as self-propelled cells and catalytic swimmers disturb the fluid around them as stresslets, symmetric force dipoles whose flow field decays as the inverse distance squared. The characteristics of the stresslet govern their collective dynamics and their contribution to the suspension bulk stress. Unlike swimming speeds, the stresslets of active particles are rarely determined due to the lack of a suitable theoretical framework, since it combines information on both fluid velocity and forces at the surface of the active particle. We propose a new method, based on the reciprocal theorem of Stokes flows, to compute stresslets as integrals of the velocities on the particle's surface exclusively. This method can be efficiently used to determine the stresslet of spheroidal chemically-active particles. This approach will help tuning the stresslet of artificial swimmers and tailor their collective motion in complex environments.

5:22PM L39.00005 Swimming in a suspension of rod-like molecules , JUAN SHI, THOMAS POWERS, Brown University

— In nature, it is common for microorganisms to swim in fluids with microstructure, such as mucus. Motivated by this fact, there have been many recent theoretical, computational, and experimental studies of idealized swimmers in a dilute solution of flexible polymers. Here we study this problem from a different point of view by considering swimmers in a dilute suspension of rigid rod-like polymers. We study the prescribed swimming problem of Taylor's sheet in a dilute suspension of non-Brownian rods. Using a simple continuum constitutive law for the suspension that describes the stress in terms of velocity gradient and local rod orientation, we calculate swimming speed to second order in the amplitude of the wave. Due to stresses induced by the presence of the rods, the first-order flow field differs from that of the Newtonian case. We find that the swimming speed increases linearly with rod concentration: the presence of the rods always makes the swimmer go faster. We also consider the problem of a finite swimmer by studying a two-dimensional circular squirmer. The squirmer is defined as a circle with a prescribed tangential slip velocity that leads to propulsion. By varying the prescribed slip boundary condition, we study both pushers and pullers.

5:35PM L39.00006 Elastohydrodynamics of flagellated microorganisms , GAOJIN LI, ARE-ZOO ARDEKANI, Purdue University

— The swimming motion of many microorganisms and cells are achieved by the waving deformation of their cilia and flagella. The typical structure of flagella and cilia contains nine doublets of parallel microtubules in a cylindrical arrangement surrounding one pair of microtubules in the center. The dynein molecular motors internally drive the sliding motion between the neighboring microtubules and cause the bending motion of the flagella and cilia and drive the microorganism swimming motion. In this work, we develop a numerical model for a microorganism swimming by an internally self-driven filament. Our numerical method captures the interaction between the elasticity of the flagellum and the surround fluid. The no-slip boundary conditions are satisfied by an iterative distributed Lagrangian multiplier method. We also investigate the effects of the non-Newtonian fluid rheology on the motion of an elastic flagellum near a wall.

5:48PM L39.00007 What causes periodic beating in sperm flagella: synchronized internal forcing or fluid-structure interaction? , RANGANATHAN PRABHAKAR, Dept. of Mech. & Aerosp.

Eng., Monash University, ASHWIN NANDAGIRI, IITB-Monash Research Academy, SAMEER JADHAV, Dept. of Chem. Eng., Indian Institute of Technology Bombay — Eucaryotic cells such as sperm propel themselves using internally driven flagella. Two different models for the origin of the whip-like beating observed in such flagella are compared. The first model assumes that internal protein motors actuate synchronously to cause a traveling active-force wave within the filament. The forcing wave is chosen such that its resultant and the total torque on the filament are zero. In contrast, the second model assumes that forces and torques exerted by the motors locally sum to zero across the scale of the filament diameter. The only effect of the motor activity is to give rise to a stresslet distribution across the filament length. In either model, the slender filament is modeled as a bead-spring chain with hydrodynamic interactions. Flagellar waveforms and trajectory patterns obtained are compared systematically while keeping the dissipation rates the same for the two models. Periodic beating emerges in freely swimming filaments with the second model without any imposed periodicity in the stresslet distribution. This suggests that periodic waveforms in eucaryotic flagella can emerge by fluid-structure interactions alone without significant internal synchronization of protein motor activity.

6:01PM L39.00008 Vortex arrays and ciliary tangles underlie the feeding-swimming tradeoff in starfish larvae , WILLIAM GILPIN, Department of Applied Physics, Stanford University, VIVEK N. PRAKASH,

MANU PRAKASH, Department of Bioengineering, Stanford University — Many marine invertebrates have larval stages covered in linear arrays of beating cilia, which propel the animal while simultaneously entraining planktonic prey. These bands are strongly conserved across taxa spanning four major superphyla, and they are responsible for the unusual morphologies of many invertebrates. However, few studies have investigated their underlying hydrodynamics. Here, we study the ciliary bands of starfish larvae, and discover a beautiful pattern of slowly-evolving vortices that surrounds the swimming animals. Closer inspection of the bands reveals unusual ciliary tangles analogous to topological defects that break-up and re-form as the animal adjusts its swimming stroke. Quantitative experiments and modeling demonstrate that these vortices create a physical tradeoff between feeding and swimming in heterogenous environments, which manifests as distinct flow patterns or "eigenstrokes" representing each behavior—potentially implicating neuronal control of cilia. This quantitative interplay between larval form and hydrodynamic function generalizes to other invertebrates, and illustrates the potential effects of active boundary conditions in other biological and synthetic systems.

6:14PM L39.00009 Sorting of Sperm by Morphology , JAMES KOH, MARCOS MARCOS¹, Nanyang Technological University

— Many studies have proven that the percentage of morphologically normal sperm is a significant factor in determining the success of assisted reproduction. The velocity of sperm in a microchannel with shear flow subjected to an external field will be explored theoretically. The difference in response between morphologically normal and abnormal sperm will be computed from a statistical approach, to study the feasibility and effectiveness of sorting by an external field to remove abnormal sperm.

¹The full name of this author is Marcos.

6:27PM L39.00010 Stability transitions and directional flipping in a microswimmer with superparamagnetic links , YUVAL HARDUF, YIZHAR OR, Technion, Israel Institute of Technology

— The famous work of Dreyfus et al (2005) introduced a microswimmer composed of a chain of superparamagnetic beads and actuated by a planar oscillating magnetic field. Further numerical simulations of the swimmer model by Gauger & Stark (2006) revealed that for large enough oscillation amplitude of the magnetic field's direction, the swimmer's mean orientation and net swimming direction both flip from the mean direction of the magnetic field to a direction perpendicular to it. This observation has been confirmed experimentally in Roper et al (2008). In our work, we analyze this phenomenon theoretically by studying the simplest possible microswimmer model: two slender links connected by an elastic joint, while one link is superparamagnetic. The dynamic equations of motion are formulated explicitly, and approximated by a second-order system which resembles the well-known Kapitza pendulum with an oscillating pivot. Conditions for stability transitions induced by the system's parametric excitation are obtained numerically and analytically by using Hill's equation and infinite determinant. Remarkably, it is also found that there exist intermediate parameter regions of dynamic bistability where the aligned and perpendicular directions are both stable under different initial conditions.

Monday, November 21, 2016 4:30PM - 6:40PM –

Session L40 Porous Media Flows: Mixing, Turbulence, and Petroleum Applications

Portland Ballroom 253-258-254-257 - Joachim Moortgat, Ohio State University

4:30PM L40.00001 Thermodynamically Consistent Fluid Mixing in Porous Media Induced by Viscous Fingering and Channeling of Multiphase Flow, MOHAMMAD AMIN AMOOIE, MOHAMMAD REZA SOLTANIAN , JOACHIM MOORTGAT, The Ohio State University — Fluid mixing and its interplay with viscous fingering as well as flow channeling through heterogeneous media have been traditionally studied for fully (im)miscible conditions in which a (two-) single-phase system is represented by two components, e.g. a solvent and a solute, with (zero) infinite mutual solubility. However, many subsurface problems, e.g. gas injection/migration in hydrocarbon reservoirs, involve multiple species transfer. Multicomponent fluid properties behave non-linearly, through an equation of state, as a function of temperature, pressure, and compositions. Depending on the minimum miscibility pressure, a two-phase region with finite, non-zero mutual solubility may develop, e.g. in a partially-miscible system. Here we study mixing of fluids with partial mutual solubility, induced by viscous flow fingering, channeling, and species transport within and between phases. We uncover non-linear mixing dynamics of a finite-size slug of a less viscous fluid attenuated by a carrier fluid during rectilinear displacement. We perform accurate numerical simulations that are thermodynamically-consistent to capture fingering patterns and complex phase behavior of mixtures. The results provide a broad perspective into how multiphase flow can alter fluid mixing in porous media.

4:43PM L40.00002 Upscaling of Mixing Processes using a Spatial Markov Model¹, DIOGO BOLSTER, University of Notre Dame, NICOLE SUND, Desert Research Institute, GIOVANNI PORTA, Politecnico de Milano — The Spatial Markov model is a model that has been used to successfully upscale transport behavior across a broad range of spatially heterogeneous flows, with most examples to date coming from applications relating to porous media. In its most common current form the model predicts spatially averaged concentrations. However, many processes, including for example chemical reactions, require an adequate understanding of mixing below the averaging scale, which means that knowledge of subscale fluctuations, or closures that adequately describe them, are needed. Here we present a framework, consistent with the Spatial Markov modeling framework, that enables us to do this. We apply and present it as applied to a simple example, a spatially periodic flow at low Reynolds number. We demonstrate that our upscaled model can successfully predict mixing by comparing results from direct numerical simulations to predictions with our upscaled model. To this end we focus on predicting two common metrics of mixing: the dilution index and the scalar dissipation. For both metrics our upscaled predictions very closely match observed values from the DNS.

¹This material is based upon work supported by NSF grants EAR-1351625 and EAR-1417264.

4:56PM L40.00003 Multiscale Lagrangian Statistics of Curvature Angle in Pore-Scale Turbulence¹, BRYAN HE, Oregon State University, Corvallis, USA, BENJAMIN KADOCH, Aix-Marseille Université, Marseille, France, SOURABH APTE, Oregon State University, Corvallis, USA, MARIE FARGE, LMD-IPSL-CNRS, Ecole Normale Supérieure, Paris, France, KAI SCHNEIDER, Aix-Marseille Université, Marseille, France — Porescale turbulent flow physics are investigated using Direct Numeric Simulation (DNS) of flow through a periodic face centered cubic (FCC) unit cell at Reynolds numbers of 300, 500 and 1000. The simulations are performed using a fictitious domain approach [Apte et al, J. Comp. Physics 2009], which uses non-body conforming Cartesian grids. Lagrangian statistics of scale dependent curvature angle and acceleration are calculated by tracking a large number of fluid particle trajectories. For isotropic turbulence, it has been shown [Bos et al. 2015, PRL] that the mean curvature angle varies linearly with time initially, reaches an inertial range and asymptotes to a value of $\pi/2$ at long times, corresponding to the decorrelation and equipartition of the cosine of the curvature angle. Similar trends are observed at early times for turbulence in porous medium; however, the mean curvature angle asymptotes to a value larger than $\pi/2$, due to the effect of confinement on the fluid particle trajectories that result in preferred directions at large times. A Monte-Carlo based stochastic model to predict the long-time behavior of curvature angles is developed and shown to correctly predict an angle larger than $\pi/2$ at large times.

¹NSF project numbers 1336983, 1133363.

5:09PM L40.00004 ABSTRACT WITHDRAWN –

5:22PM L40.00005 Turbulent flow characteristics over anisotropic porous media, KAZUHIKO SUGA, UNDE HO, SEITARO NAKAMURA, MASAYUKI KANEDA, Osaka Prefecture Univ — Planar PIV measurements of turbulence over anisotropic porous media are carried out. Three kinds of anisotropic porous media are considered to form a bottom wall of fully developed turbulent channel flows. Their wall normal component of the permeability is larger than the other components by the factor of 1.3-170. The range of the measured Reynolds number is $Re_b=1300-13000$. It is found that the streamwise component significantly more affects the turbulent flow characteristics than the other components. It is considered that the streamwise permeability controls the strength of traveling waves over the porous surface which are originated by the Kelvin-Helmholtz instability and drive transverse roll vortices. Considering the present data and our previous data of isotropic porous wall flows, an effective permeability Reynolds number that is a parameter to characterize the turbulence over porous walls is proposed. This effective permeability Reynolds number is a tensorially consistent expression including both streamwise and wall normal components of the permeability. It is shown that the zero-plane displacement, equivalent roughness height and Karman constant variations of mean velocity profiles well correlate with the effective permeability Reynolds number.

5:35PM L40.00006 Fluid-driven fractures in brittle hydrogels, NIALL O'KEEFFE, PAUL LINDEN, DAMTP, University of Cambridge — Hydraulic fracturing is a process in which fluid is injected deep underground at high pressures that can overcome the strength of the surrounding matrix. This results in an increase of surface area connected to the well bore and thus allows extraction of natural gas previously trapped in a rock formation. We experimentally study the physical mechanisms of these fluid-driven fractures in low permeability reservoirs where the leak-off of fracturing fluid is considered negligible. This is done through the use of small scale experiments on transparent and brittle, heavily cross-linked hydrogels. The propagation of these fractures can be split into two distinct regimes depending on whether the dominant energy dissipation mechanism is viscous flow or material toughness. We will analyse crack growth rates, crack thickness and tip shape in both regimes. Moreover, PIV techniques allow us to explore the flow dynamics within the fracture, which is crucial in predicting transport of proppants designed to prevent localisation of cracks.

5:48PM L40.00007 Model of fluid flow and internal erosion of a porous fragile medium¹, ARSHAD KUDROLLI, XAVIER CLOTET, Clark Univ — We discuss the internal erosion and transport of particles leading to heterogeneity and channelization of a porous granular bed driven by fluid flow by introducing a model experimental system which enables direct visualization of the evolution of porosity from the single particle up to the system scale [1]. Further, we develop a hybrid hydrodynamic-statistical model to understand the main ingredients needed to simulate our observations. A uniqueness of our study is the close coupling of the experiments and simulations with control parameters used in the simulations derived from the experiments. Understanding this system is of fundamental importance to a number of geophysical processes, and in the extraction of hydrocarbons in the subsurface including the deposition of proppants used in hydraulic fracturing. We provide clear evidence for the importance of curvature of the interface between high and low porosity regions in determining the flux rate needed for erosion and the spatial locations where channels grow. [1]: Evolution of Porosity and Channelization of an Erosive Medium Driven by Fluid Flow, Arshad Kudrolli and Xavier Clotet, Phys. Rev. Lett. 117, 028001 (2016).

¹This material is based upon work supported by the U.S. Department of Energy Office of Science, Office of Basic Energy Sciences program under DE-SC0010274.

6:01PM L40.00008 A model to determine the petroleum pressure in a well using fractional differential equations, BEATRIZ BRITO MARTINEZ, FERNANDO BRAMBILA PAZ, Facultad de Ciencias, Universidad Nacional Autonoma de Mexico, CARLOS FUENTES RUIZ, Instituto Mexicano de Tecnologia del Agua — A noninvasive method was used to determine the pressure of petroleum leaving a well. The mathematical model is based on nonlinear fractional differential equations. This model comes from the fractal dimension of the porous medium. The problem is solved in three stages. In the first stage the fractal dimension of the porous medium is determined. We show that microwaves reflected and transmitted through soil have a fractal dimension which is correlated with the fractal dimension of the porous medium. The fractal signature of microwave scattering correlates with certain physical and mechanical properties of soils (porosity, permeability, conductivity, etc.). In the second stage we use three partial fractional equations as a mathematical model to study the diffusion inside the porous medium. In this model sub-diffusive phenomenon occurs if fractal derivative is between zero and one and supra-diffusive occurs if the derivative is greater than 1 and less than 2. Finally in the third stage the mathematical model is used to determinate the petroleum pressure output in a Mexican oil field, which contains three partial fractional equations with triple porosity and permeability.

6:14PM L40.00009 An advective volume-balance model for flow in porous media¹, CARLOS MALAGA, FRANCISCO MANDUJANO, Physics Department. School of Science. Universidad Nacional Autonoma de Mexico, JULIAN BECERRA, Abacus, CINVESTAV — Volume-balance models are used by petroleum engineers to simulate 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. Preliminary numerical tests of phase separation due to gravity suggest the model reproduces qualitatively the physical phenomena.

¹Fondo Sectorial CONACYT-SENER grant number 42536 (DGAJ-SPI-34-170412-217)

6:27PM L40.00010 Outer boundary effects in a petroleum reservoir, RHODRI NELSON, DARREN CROWDY, EVERETT KROPF, Imperial College London, LIHUA ZUO, RUUD WEIJERMARS, Texas AM University — A new toolkit for potential theory based on the Schottky-Klein prime function is first introduced. This potential theory toolkit is then applied to study the fluid flow structures in bounded 2D petroleum reservoirs. In the model, reservoirs are assumed to be heterogeneous and isotropic porous medium and can thus be modelled using Darcys equation. First, computations of flow contours are carried out on some test domains and benchmarked against results from the ECLIPSE reservoir simulator. Following this, a case study of the Quitman oil field in Texas is presented.

Tuesday, November 22, 2016 8:00AM - 9:44AM –
Session M1 Mini-Symposium - Multiphase Flows in Biomedicine I A105 - Michael Calvisi, University of Colorado, Colorado Springs

8:00AM M1.00001 Theory of margination and cell-free layer thickness in blood flow, MICHAEL GRAHAM, Univ of Wisconsin, Madison — A mechanistic model is developed to describe segregation in confined multicomponent suspensions such as blood during Couette or plane Poiseuille flow. We focus attention on the case of a binary suspension with a deformable primary component (e.g. red blood cells) that completely dominates the collision dynamics in the system. The model captures the phenomena of depletion layer formation and margination observed in confined multicomponent suspensions of deformable particles. The depletion layer thickness of the primary component is predicted to follow a master curve relating it in a specific way to confinement ratio and volume fraction. Results from experiments and detailed simulations with different parameters (flexibility, viscosity ratio, confinement) collapse onto this curve with only one adjustable parameter. In a binary suspension, several regimes of segregation arise, depending on the value of a "margination parameter" M . Most importantly, in both Couette and Poiseuille flows there is a critical value of M below which a sharp "drainage transition" occurs: one component is completely depleted from the bulk flow to the vicinity of the walls. Direct simulations also exhibit this transition as the size or flexibility ratio of the components changes.

8:26AM M1.00002 Two problems in multiphase biological flows: Blood flow and particulate transport in microvascular network, and pseudopod-driven motility of amoeboid cells¹, PROSENJIT BAGCHI, Rutgers University — In this talk, two problems in multiphase biological flows will be discussed. The first is the direct numerical simulation of whole blood and drug particulates in microvascular networks. Blood in microcirculation behaves as a dense suspension of heterogeneous cells. The erythrocytes are extremely deformable, while inactivated platelets and leukocytes are nearly rigid. A significant progress has been made in recent years in modeling blood as a dense cellular suspension. However, many of these studies considered the blood flow in simple geometry, e.g., straight tubes of uniform cross-section. In contrast, the architecture of a microvascular network is very complex with bifurcating, merging and winding vessels, posing a further challenge to numerical modeling. We have developed an immersed-boundary-based method that can consider blood cell flow in physiologically realistic and complex microvascular network. In addition to addressing many physiological issues related to network hemodynamics, this tool can be used to optimize the transport properties of drug particulates for effective organ-specific delivery. Our second problem is pseudopod-driven motility as often observed in metastatic cancer cells and other amoeboid cells. We have developed a multiscale hydrodynamic model to simulate such motility. We study the effect of cell stiffness on motility as the former has been considered as a biomarker for metastatic potential.

¹Funded by the National Science Foundation

8:52AM M1.00003 Acoustic droplet vaporization of vascular droplets in gas embolotherapy, JOSEPH BULL, Univ of Michigan - Ann Arbor — This work is primarily motivated by a developmental gas embolotherapy technique for cancer treatment. In this methodology, infarction of tumors is induced by selectively formed vascular gas bubbles that arise from the acoustic vaporization of vascular droplets. Additionally, micro- or nano-droplets 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*. Functionalized droplets that are targeted to tumor vasculature are examined. The influence of fluid mechanical and acoustic parameters, as well as droplet functionalization, is explored. This work was supported by NIH grant R01EB006476.

9:18AM M1.00004 Temperature measurements in cavitation bubbles.¹, OLIVIER COUTIER-DELGOSHA², Arts et Metiers ParisTech / Virginia Tech — Cavitation is usually a nearly isothermal process in the liquid phase, but in some specific flow conditions like hot water or cryogenic fluids, significant temperature variations are detected. In addition, a large temperature increase happens inside the cavitation bubbles at the very end of their collapse, due to the fast compression of the gas at the bubble core, which is almost adiabatic. This process is of primary interest in various biomedical and pharmaceutical applications, where the mechanisms of bubble collapse plays a major role. To investigate the amplitude and the spatial distribution of these temperature variations inside and outside the cavitation bubbles, a system based on cold wires has been developed. They have been tested in a configuration of a single bubble obtained by submitting a small air bubble to a large amplitude pressure wave. Some promising results have been obtained after the initial validation tests.

¹This work is funded by the Office of Naval Research Global under grant N62909-16-1-2116, Dr. Salahuddin Ahmed & Ki-Han Kim program managers

²Two other authors should be added : Merouane Hamdi & Sylvie Fuzier, from Arts et Metiers ParisTech

Tuesday, November 22, 2016 8:00AM - 10:10AM –
Session M2 Industrial Application: Marine Hydrokinetic Turbines A106 - Martin Wosnik,
University of New Hampshire

8:00AM M2.00001 Cross-flow turbines: progress report on physical and numerical model studies at large laboratory scale¹, MARTIN WOSNIK, PETER BACHANT, University of New Hampshire — Cross-flow turbines show potential in marine hydrokinetic (MHK) applications. A research focus is on accurately predicting device performance and wake evolution to improve turbine array layouts for maximizing overall power output, i.e., minimizing wake interference, or taking advantage of constructive wake interaction. Experiments were carried with large laboratory-scale cross-flow turbines $D \sim O(1m)$ using a turbine test bed in a large cross-section tow tank, designed to achieve sufficiently high Reynolds numbers for the results to be Reynolds number independent with respect to turbine performance and wake statistics, such that they can be reliably extrapolated to full scale and used for model validation. Several turbines of varying solidity were employed, including the UNH Reference Vertical Axis Turbine (RVAT) and a 1:6 scale model of the DOE-Sandia Reference Model 2 (RM2) turbine. To improve parameterization in array simulations, an actuator line model (ALM) was developed to provide a computationally feasible method for simulating full turbine arrays inside Navier-Stokes models. Results are presented for the simulation of performance and wake dynamics of cross-flow turbines and compared with experiments and body-fitted mesh, blade-resolving CFD.

¹supported by NSF-CBET grant 1150797, Sandia National Laboratories

8:13AM M2.00002 Advancing marine hydrokinetic turbine arrays towards large-scale deployments in sandy rivers: a laboratory study.¹, MIRKO MUSA, St. Anthony Falls Lab, CEGE, University of Minnesota, CRAIG HILL, University of Washington, MICHELE GUALA, St. Anthony Falls Lab, CEGE, University of Minnesota — A staggered array of twelve axial-flow marine hydrokinetic (MHK) turbine models was investigated at the St. Anthony Falls Laboratory under live-bed sediment transport conditions. In particular, the interaction between the MHK power plant and the complex migrating bedforms was monitored using a state-of-the-art high-resolution submersible laser scanning device able to provide spatio(x,y)-temporally(t) resolved channel bathymetry $z(x,y,t)$. Results revealed both a local signature of each individual turbine and a cumulative array effect that extends farther from the site. Single turbine localized scour results from the blockage effect of the operating rotor and the consequent flow acceleration between the lower rotor tip and the erodible bed. The resultant shear stress enhancement around the device protects the turbine during extreme sediment transport conditions, ultimately preventing the blades from impacting the incoming bedforms. A turbine failure case was simulated to illustrate the consequence of such event, which can irreversibly bury and damage the turbine. Additionally, velocity and turbine performance estimates provided a preliminary description of the power plant energy output, revealing similar features already observed in experimental wind farm models.

¹NSF Career Grant Geophysical Flow Control, Dr. Michele Guala

8:26AM M2.00003 Field scale simulation of axial hydrokinetic turbines in a natural marine environment¹, SAURABH CHAWDHARY, St. Anthony Falls Laboratory, Department of Mechanical Engineering, University of Minnesota, DIONYSIOS ANGELIDIS, St. Anthony Falls Laboratory, University of Minnesota, LIAN SHEN, St. Anthony Falls Laboratory, Department of Mechanical Engineering, University of Minnesota, FOTIS SOTIROPOULOS, Department of Civil Engineering, Stony Brook University — Commercialization of marine and hydrokinetic (MHK) energy technologies is still in the development stage. Existing technologies need fundamental research to enable efficient energy extraction from identified MHK sites. We propose a large eddy simulation (LES)-based framework to investigate the site-specific flow dynamics past MHK arrays in a real-life marine environment. To this end, we use advanced computational tools developed at the Saint Anthony Falls Laboratory (SAFL) to resolve the vast range of scales present in the flow. The new generation unstructured Cartesian flow solver, coupled with a sharp interface immersed boundary method for 3D incompressible flows, is used to numerically investigate New York City's East River, where an array of MHK turbines is to be deployed as part of the Roosevelt Island Tidal Energy (RITE) Project. Multi-resolution simulations on locally refined grids are used to simulate the flow in a section of the East River with detailed river bathymetry and inset turbines at field scale. The results are analyzed in terms of the wake recovery, overall wake dynamics, and the power produced by the turbines. These results will help develop design guidelines for the site-specific turbine array configuration.

¹This work was supported by NSF grant IIP-1318201.

8:39AM M2.00004 Performance Characteristics of a Vertical Axis Hydrokinetic Turbine, BENJAMIN BAILIN, KAREN FLACK, ETHAN LUST, US Naval Academy — Performance characteristics are presented for a vertical axis hydrokinetic turbine designed for use in a riverine environment. The test turbine is a 1:6 scale model of a three-bladed device (9.5 m span, 6.5 m diameter) that has been proposed by the Department of Energy. Experiments are conducted in the large towing tank (116 m long, 7.9 m wide, 5 m deep) at the United States Naval Academy. The large scale facility allows for scale independent results. The turbine is towed beneath a moving carriage at a constant speed in combination with a shaft brake to achieve the desired tip speed ratio (TSR) range. The measured quantities of turbine thrust, torque and RPM result in power and thrust coefficients for a range of TSR. Results will be presented for cases with quiescent flow and flow with mild surface waves, representative of riverine environments.

8:52AM M2.00005 Wake Survey of a Marine Current Turbine Under Steady Conditions, ETHAN LUST, LUKSA LUZNIK, KAREN FLACK, US Naval Academy — A submersible particle image velocimetry (PIV) system was used to study the wake of a horizontal axis marine current turbine. The turbine was tested in a large tow tank facility at the United States Naval Academy. 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. Separate wind tunnel testing has shown the foil section used on the turbine to be Reynolds number independent with respect to lift at the experimental parameters of tow carriage speed ($U_{\text{tow}} = 1.68 \text{ m/s}$) and tip speed ratio (TSR = 7). The wake survey was conducted over an area extending 0.25D forward of the turbine tip path to 2.0D aft, and 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 into a single field of investigation. Results include streamwise and vertical ensemble average velocity fields averaged over approximately 1,000 realizations, as well as higher-order statistics. Turbine tip vortex centers were identified and plotted showing increasing aperiodicity with wake age. keywords: horizontal axis marine current turbine, particle image velocimetry, towing tank, wake survey

9:05AM M2.00006 Influence of surface gravity waves on near wake development behind a towed model horizontal axis marine current turbine¹, LUKSA LUZNIK, KAREN FLACK, ETHAN LUST, US Naval Academy — 2D PIV measurements in the near wake flow field ($x/D < 2$) are presented for a 1/25 scale, 0.8 m diameter (D) two bladed horizontal axis tidal turbine. All measurements were obtained in the USNA 380 ft tow tank with turbine towed at a constant carriage speed ($U_{\text{tow}} = 1.68 \text{ m/s}$), at the nominal tip speed ratio (TSR) of 7 and incoming regular waves with a period of 2.3 seconds and 0.18 m wave height. Near wake mapping is accomplished by "tiling" phase locked individual 2D PIV fields of view (nominally 30x30 cm²) with approximately 5 cm overlap. The discussion will focus on the downstream evolution of coherent tip vortices shed by the rotor blades and their vertical/horizontal displacements by the wave induced fluctuations. This observed phenomena ultimately results in significantly increased downstream wake expansion in comparison with the same conditions without waves.

¹Office of Naval Research

9:18AM M2.00007 The effect of active control on the performance and wake characteristics of an axial-flow Marine Hydrokinetic turbine, CRAIG HILL, KATHERINE VANNESS, Department of Mechanical Engineering, University of Washington, ANDY STEWART, Applied Physics Lab, University of Washington, BRIAN POLAGYE, ALBERTO ALISEDA, Department of Mechanical Engineering, University of Washington — Turbulence-induced unsteady forcing on turbines extracting power from river, tidal, or ocean currents will affect performance, wake characteristics, and structural integrity. A laboratory-scale axial-flow turbine, 0.45 m in diameter, incorporating rotor speed sensing and independent blade pitch control has been designed and tested with the goal of increasing efficiency and/or decreasing structural loading. Laboratory experiments were completed in a 1 m wide, 0.75 m deep open-channel flume at moderate Reynolds number ($Re_c = 6 \cdot 10^4 - 2 \cdot 10^5$) and turbulence intensity ($T.I. = 2 - 10\%$). A load cell connecting the hub to the shaft provided instantaneous forces and moments on the device, quantifying turbine performance under unsteady inflow and for different controls. To mitigate loads, blade pitch angles were controlled via individual stepper motors, while a six-axis load cell mounted at the root of one blade measured instantaneous blade forces and moments, providing insights into variable loading due to turbulent inflow and blade-tower interactions. Wake characteristics with active pitch control were compared to fixed blade pitch and rotor speed operation. Results are discussed in the context of optimization of design for axial-flow Marine Hydrokinetic turbines.

9:31AM M2.00008 Experimental Validation of a Theory for a Variable Resonant Frequency Wave Energy Converter (VRFWEC), MINOK PARK, LOUIS VIREY, University of California, Berkeley, ZHONGFEI CHEN, Harbin Engineering University, SIMO MKIHARJU, University of California, Berkeley — A point absorber wave energy converter designed to adapt to changes in wave frequency and be highly resilient to harsh conditions, was tested in a wave tank for wave periods from 0.8 s to 2.5 s. The VRFWEC consists of a closed cylindrical floater containing an internal mass moving vertically and connected to the floater through a spring system. The internal mass and equivalent spring constant are adjustable and enable to match the resonance frequency of the device to the exciting wave frequency, hence optimizing the performance. In a full scale device, a Permanent Magnet Linear Generator will convert the relative motion between the internal mass and the floater into electricity. For a PMLG as described in Yeung et al. (OMAE2012), the electromagnetic force proved to cause dominantly linear damping. Thus, for the present preliminary study it was possible to replace the generator with a linear damper. While the full scale device with 2.2 m diameter is expected to generate $O(50 \text{ kW})$, the prototype could generate $O(1 \text{ W})$. For the initial experiments the prototype was restricted to heave motion and data compared to predictions from a newly developed theoretical model (Chen, 2016).

9:44AM M2.00009 Results from the field test of two 1 kW oscillating hydrofoil generators in a tidal canal¹, MICHAEL MILLER, JENNIFER CARDONA, LEANNE BLOCK, KENTA KONDO, MICHAEL LEE, REBECCA LORICK, MICHAEL MANNING, ISABEL SCHERL, FILIP SIMESKI, ARIANE SPAULDING, YUNXING SU, Brown University, DAVID ELLERBY, Wellesly College, ERIKA SUDDERTH, KRISTEN LEWIS, Volpe - The National Transportation Authority, JAMES KIDD, WILLIAM HUBBARD, HUNG TOM PHAM, Massachusetts Maritime Academy, TOM DERECKTOR, STEVE WINCKLER, BluSource Energy, Inc, ALICE FAWZI, JENNIFER FRANCK, KENNETH BREUER, SHREYAS MANDRE, Brown University — We present results from field tests of two 1 kW hydrokinetic energy capture devices operating in the Cape Cod Canal, in Bourne, MA. Each device consists of two oscillating hydrofoils with a chord of 0.24 m and span of 1.35 m, operating 90° out of phase with each other and driving a single generator. The pitch of each hydrofoil is mechanically coupled to the heave, also with a 90° phase difference. The two devices are arranged in tandem with a stream-wise separation of 1 span. We find that depending on the operating conditions, the hydrofoil oscillations may synchronize with each other through hydrodynamic interactions. Furthermore, in their optimized operation, the trailing device generates 60-80

¹ARPA-e DE AR0000318

9:57AM M2.00010 Wingtip Devices for Marine Applications¹, IVAYLO NEDYALKOV, TIMOTHY BARRETT, ALEKSANDRA WOJTOWICZ, MARTIN WOSNIK, University of New Hampshire — Wingtip devices are widely used in aeronautics, and have been gaining popularity in wind and marine turbine applications. Although the principles of operation of the devices in air and water are similar, one major difference in the marine environment is the presence of cavitation. In an integrated numerical and experimental study, three wingtip devices were attached to an elliptical foil and compared to a reference case (no wingtip). Lift, drag, and cavitation characteristics were obtained both numerically (in OpenFOAM) and experimentally (in the University of New Hampshire High-Speed Cavitation Tunnel). As expected, with the addition of wingtip devices, the maximum lift/drag ratio increases and tip vortex cavitation is suppressed. The next step in the study is to develop a theoretical relationship between tip-vortex cavitation inception and flow parameters for foils with non-elliptical load distribution, such as foils with wingtips.

¹The authors would like to acknowledge Ian Gagnon, Benjamin Mitchell, and Alexander Larson for their help in conducting experiments.

Tuesday, November 22, 2016 8:00AM - 10:10AM – Session M4 Energy: Thermodynamics and Combustion B112 - Amy Lang, University of Alabama

8:00AM M4.00001 High Speed Imaging of Diesel Fuel Sprays¹, JA'KIRA JACKSON, Alabama Agricultural & Mechanical University, JOSHUA BITTLE, The University of Alabama — Fuel sprays primarily serve as methods for fuel distribution, fuel/air mixing, and atomization. In this research, a constant pressure flow rig vessel is being tested at various pressures and temperatures using n-heptane. The experiment requires two imaging techniques: color Schlieren and Mie-scatter. Schlieren captures density gradients in a spray which includes both liquid and vapor phases while Mie-scatter is only sensitive to the liquid phase of the fuel spray. Essentially, studies are mainly focused on extracting the liquid boundary from the Schlieren to possibly eliminate the need for acquiring the Mie-Scatter technique. Four test conditions (combination of low and high pressure and temperatures) are used in the application to attempt to find the liquid boundary independent of the Mie-scatter technique. In this pursuit the following methods were used: a color threshold, a value threshold, and the time variation in color. All methods provided some indication of the liquid region but none were able to capture the full liquid boundary as obtained by the Mie-scatter results.

¹Funding from NSF REU site grant EEC 1358991 is greatly appreciated.

8:13AM M4.00002 Study of ethanol and gasoline fuel sprays using mie-scatter and schlieren imaging¹, LAUREN BOUCHARD, Smith College, JOSHUA BITTLE, PAUL PUZINAUSKAS, The University of Alabama — Many cars today are capable of running on both gasoline and ethanol, however it is not clear how well optimized the engines are for the multiple fuels. This experiment looks specifically at the fuel spray in a direct injection system. The length and angle of direct injection sprays were characterized and a comparison between ethanol and gasoline sprays was made. Fuels were tested using a modified diesel injector in a test chamber at variable ambient pressures and temperatures in order to simulate both high and low load combustion chamber conditions. Rainbow schlieren and mie-scatter imaging were both used to investigate the liquid and vapor portions of the sprays. The sprays behaved as expected with temperature and pressure changes. There was no noticeable fuel effect on the liquid portion of the spray (mie-scatter), though the gasoline vapor spray angles were wider than ethanol spray angles (possible a result of the distillation curves of the two fuels).

¹Funding from NSF REU site grant EEC 1358991 is greatly appreciated.

8:26AM M4.00003 Effects of porous insert on flame dynamics in a lean premixed swirl-stabilized combustor¹, MARCUS BROWN, Hastings College, AJAY AGRAWAL, JAMES ALLEN, JOHN KORNEGAY, The University of Alabama — In this study, we investigated different methods of determining the effect a porous insert has on flame dynamics during lean premixed combustion. A metallic porous insert is used to mitigate instabilities in a swirl-stabilized combustor. Thermoacoustic instabilities are seen as negative consequences of lean premixed combustion and eliminating them is the motivation for our research. Three different diagnostics techniques with high-speed Photron SA5 cameras were used to monitor flame characteristics. Particle image velocimetry (PIV) was used to observe vortical structures and recirculation zones within the combustor. Using planar laser induced fluorescence (PLIF), we were able to observe changes in the reaction zones during instabilities. Finally, utilizing a color high-speed camera, visual images depicting a flame's oscillations during the instability were captured. Using these monitoring techniques, we are able to support the claims made in previous studies stating that the porous insert in the combustor significantly reduces the thermoacoustic instability.

¹Funding for this research was provided by the NSF REU site grant EEC 1358991 and NASA grant NNX13AN14A

8:39AM M4.00004 Effects of Heat Loss and Subgrid-Scale Models on Large Eddy Simulations of a Premixed Jet Combustor Using Flamelet-Generated Manifolds, FRANCISCO E. HERNANDEZ PEREZ, BOK JIK LEE, HONG G. IM, King Abdullah University of Science and Technology, ALESSIO FANCELLO, ANDREA DONINI, JEROEN A. VAN OIJEN, L. PHILIP H. DE GOEY, Eindhoven University of Technology — Large eddy simulations (LES) of a turbulent premixed jet flame in a confined chamber are performed using the flamelet-generated manifold technique for tabulation of chemical kinetics and the OpenFOAM framework for computational fluid dynamics. The configuration is characterized by an off-center nozzle having an inner diameter of 10 mm, feeding a lean methane-air mixture with an equivalence ratio of 0.71 and mean velocity of 90 m/s, at 573 K and atmospheric pressure. Conductive heat loss is accounted for in the manifold via burner-stabilized flamelets and the subgrid-scale (SGS) turbulence-chemistry interaction is modeled via presumed filtered density functions. The effects of heat loss inclusion as well as SGS modeling for both the SGS stresses and SGS variance of progress variable on the numerical predictions are all systematically investigated. Comparisons between numerical results and measured data show a considerable improvement in the prediction of temperature when heat losses are incorporated into the manifold, as compared to the adiabatic one. In addition, further improvements in the LES predictions are achieved by employing SGS models based on transport equations.

8:52AM M4.00005 Effects of pressure on syngas/air turbulent nonpremixed flames, BOK JIK LEE, HONG G. IM, King Abdullah University of Science and Technology, PIETRO PAOLO CIOTTOLI, MAURO VALORANI, Sapienza University of Rome — Large eddy simulations (LES) of turbulent non-premixed jet flames were conducted to investigate the effects of pressure on the syngas/air flame behavior. The software to solve the reactive Navier-Stokes equations was developed based on the OpenFOAM framework, using the YSLFM library for the flamelet-based chemical closure. The flamelet tabulation is obtained by means of an in-house code designed to solve unsteady flamelets of both ideal and real fluid mixtures. The validation of the numerical setup is attained by comparison of the numerical results with the Sandia/ETH-Zurich experimental database of the CO/H₂/N₂ non-premixed, unconfined, turbulent jet flame, referred to as Flame A. Two additional simulations, at pressure conditions of 2 and 5 atm, are compared and analyzed to unravel computational and scientific challenges in characterizing turbulent flames at high pressures. A set of flamelet solutions, representative of the jet flames under review, are analyzed following a CSP approach. In particular, the Tangential Stretching Rate (TSR), representing the reciprocal of the most energetic time scale associated with the chemical source term, and its extension to reaction-diffusion systems (extended TSR), are adopted.

9:05AM M4.00006 Coupling Between Turbulent Boundary Layer and Radiative Heat Transfer Under Engine-Relevant Conditions¹, A SIRCAR, C PAUL, S FERREYRO, A IMREN, D C HAWORTH, Penn State, S ROY, W GE, M F MODEST, UC Merced — The lack of accurate submodels for in-cylinder radiation and heat transfer has been identified as a key shortcoming in developing truly predictive CFD models that can be used to develop combustion systems for advanced high-efficiency, low-emissions engines. Recent measurements of wall layers in engines show discrepancies of up to 100% with respect to standard CFD boundary-layer models. And recent analysis of in-cylinder radiation based on recent spectral property databases and high-fidelity radiative transfer equation (RTE) solvers has shown that at operating conditions typical of heavy-duty CI engines, radiative emission can be as high as 40% of the wall heat losses, that molecular gas radiation can be more important than soot radiation, and that a significant fraction of the emitted radiation can be reabsorbed before reaching the walls. That is, radiation changes the in-cylinder temperature distribution, which in turn affects combustion and emissions. The goal of this research is to develop models that explicitly account for the potentially strong coupling between radiative and turbulent boundary layer heat transfer. For example, for optically thick conditions, a simple diffusion model might be formulated in terms of an absorption-coefficient-dependent turbulent Prandtl number.

¹NSF, DOE

9:18AM M4.00007 An Experimental Study on Burning Characteristics of n-Heptane/Ethanol Mixture Pool Fires in a Reduced Scaled Tunnel, AHMET YOZGATLIGIL, SINA SHAFEE, Middle East Technical University — Fire accidents in recent decades have drawn attention to safety issues associated with the design, construction and maintenance of tunnels. A reduced scale tunnel model constructed based on Froude scaling technique is used in the current work. Mixtures of n-heptane and ethanol are burned with ethanol volumetric fraction up to 30 percent and the longitudinal ventilation velocity varying from 0.5 to 2.5 m/s. The burning rates of the pool fires are measured using a precision load cell. The heat release rates of the fires are calculated according to oxygen calorimetry method and the temperature distributions inside the tunnel are also measured. Results of the experiments show that the ventilation velocity variation has a significant effect on the pool fire burning rate, smoke temperature and the critical ventilation velocity. With increased oxygen depletion in case of increased ethanol content of blended pool fires, the quasi-steady heat release rate values tend to increase as well as the ceiling temperatures while the combustion duration decreases.

9:31AM M4.00008 Radiative Heat Transfer and Turbulence-Radiation Interactions in a Heavy-Duty Diesel Engine¹, C PAUL, A. SIRCAR, S FERREYRO, A IMREN, D.C. HAWORTH, Penn State, S. ROY, W. GE, M.F. MODEST, U.C. Merced — Radiation in piston engines has received relatively little attention to date. Recently, it is being revisited in light of current trends towards higher operating pressures and higher levels of exhaust-gas recirculation, both of which enhance molecular gas radiation. Advanced high-efficiency engines also are expected to function closer to the limits of stable operation, where even small perturbations to the energy balance can have a large influence on system behavior. Here several different spectral radiation property models and radiative transfer equation (RTE) solvers have been implemented in an OpenFOAM-based engine CFD code, and simulations have been performed for a heavy-duty diesel engine. Differences in computed temperature fields, NO and soot levels, and wall heat transfer rates are shown for different combinations of spectral models and RTE solvers. The relative importance of molecular gas radiation versus soot radiation is examined. And the influence of turbulence-radiation interactions is determined by comparing results obtained using local mean values of composition and temperature to compute radiative emission and absorption with those obtained using a particle-based transported probability density function method.

¹DOE, NSF

9:44AM M4.00009 Thermodynamic Control System for cryogenic propellant storage : experimental and analytical performance assessment¹, SAMUEL MER, JEAN-PAUL THIBAULT, CNRS - UGA - LEGI, CHRISTOPHE CORRE, Ecole Centrale de Lyon - LMFA — Future operations in space exploration require to store cryogens for long duration. Residual heat loads induce cryogenic propellant vaporization and tank self-pressurization (SP), eventually leading to storage failure for large enough mission duration. The present study focuses on the Thermodynamic Venting System (TVS) control strategy : liquid propellant is pumped from the tank, cooled down by a heat exchanger and re-injected, as a jet, inside the tank. The injection is followed by vapor condensation and liquid bath destratification due to mixing. The system cold source is created thanks to a Vented Branch where a liquid fraction is withdrawn from the tank and expanded through a Joule-Thomson valve. The vented branch vaporization permits to cool down the injection loop. Quantitative analyses of SP and TVS control have been experimentally performed using a 110 L tank and a simulant fluid. A database of accurate temperature and pressure dynamics has been gathered and used to validate a homogeneous thermodynamic model which provides a fast prediction of the tank dynamics. The analytical model has been coupled with a multi-objective optimizer to identify system components and regulation strategies that maximize the tank storage duration for various mission types.

¹The authors acknowledge the joint support of the Centre National d'Etudes Spatiales and Air Liquide Advanced Technologies.

9:57AM M4.00010 Experimental Investigation of A Heat Pipe-Assisted Latent Heat Thermal Energy Storage System¹, SAEED TIARI, MAHBOOBE MAHDAVI, Mechanical Engineering Department, Gannon University, Erie, PA, SONGGANG QIU, Department of Mechanical and Aerospace Engineering, West Virginia University, Morgantown, WV — In the present work, different operation modes of a latent heat thermal energy storage system assisted by a heat pipe network were studied experimentally. Rubitherm RT55 enclosed by a vertical cylindrical container was used as the Phase Change Material (PCM). The embedded heat pipe network consisting of a primary heat pipe and an array of four secondary heat pipes were employed to transfer heat to the PCM. The primary heat pipe transports heat from the heat source to the heat sink. The secondary heat pipes transfer the extra heat from the heat source to PCM during charging process or retrieve thermal energy from PCM during discharging process. The effects of heat transfer fluid (HTF) flow rate and temperature on the thermal performance of the system were investigated for both charging and discharging processes. It was found that the HTF flow rate has a significant effect on the total charging time of the system. Increasing the HTF flow rate results in a remarkable increase in the system input thermal power. The results also showed that the discharging process is hardly affected by the HTF flow rate but HTF temperature plays an important role in both charging and discharging processes.

¹The authors would like to acknowledge the financial supports by Temple University for the project.

Tuesday, November 22, 2016 8:00AM - 9:44AM – Session M5 Compressible Flow: General B113 - Carlo Scalo, Purdue University

8:00AM M5.00001 Local and global stability analysis of compressible channel flow over wall impedance, IMAN RAHBARI, CARLO SCALO, Purdue University — The stability properties of compressible channel flow over porous walls is investigated via Local (LSA) and Global Stability Analysis (GSA) for laminar and turbulent base flows at $Re_b = 6900$ and $M_b = 0.85, 1.5, 3.5$. Linearized Navier-Stokes equations are discretized via a sixth-order fully collocated Padé scheme leading to a Generalized Eigenvalue Problem (GEVP) solved using a parallel sparse eigenvalue solver based on the shift-invert Arnoldi method. The adopted discretization guarantees spectral-like spatial resolution. Fully sparsity of the system is retained via implicit calculation of the numerical derivatives ensuring computational efficiency on multi-processor platforms. The global eigen-spectrum exhibits various sets of modes grouped by streamwise wavenumbers, which are captured via LSA, as well as global acoustic modes. Consistently with the findings of C. Scalo, et al. Phys. Fluids (2015), two unstable local modes are found for sufficiently high wall permeability: one standing-wave-like and one representing a bulk pressure mode, both generating additional Reynolds shear stresses concentrated in the viscous sublayer region. Stability properties of the flow over non-modal streamwise impedance distributions are also discussed.

8:13AM M5.00002 A Multi-Fidelity Surrogate Model for Handling Real Gas Equations of State¹, FREDERICK OUELLET, CHANYOUNG PARK, BERTRAND ROLLIN, S. "BALA" BALACHANDAR, Center for Compressible Multiphase Turbulence - Univ. of Florida — The explosive dispersal of particles is an example of a complex multiphase and multi-species fluid flow problem. This problem has many engineering applications including particle-laden explosives. In these flows, the detonation products of the explosive cannot be treated as a perfect gas so a real gas equation of state is used to close the governing equations (unlike air, which uses the ideal gas equation for closure). As the products expand outward from the detonation point, they mix with ambient air and create a mixing region where both of the state equations must be satisfied. One of the more accurate, yet computationally expensive, methods to deal with this is a scheme that iterates between the two equations of state until pressure and thermal equilibrium are achieved inside of each computational cell. This work strives to create a multi-fidelity surrogate model of this process. We then study the performance of the model with respect to the iterative method by performing both gas-only and particle laden flow simulations using an Eulerian-Lagrangian approach with a finite volume code. Specifically, the model's (i) computational speed, (ii) memory requirements and (iii) computational accuracy are analyzed to show the benefits of this novel modeling approach.

¹This 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, under Contract No. DE-NA00023

8:26AM M5.00003 Solving the Guderley Implosion Problem with a Gruneisen-Like Equation of State, JENNIFER LILIEHOLM, EMMA SCHMIDT, SCOTT RAMSEY, ZACHARY BOYD, Los Alamos National Laboratory — The Guderley problem is a solution to the inviscid Euler Equations which models a strong shock originating at infinity in both space and time. This shock converges to the one-dimensional symmetric origin, where the resulting infinite pressure causes a weaker diverging shock. The solutions to the converging and diverging shocks are self-similar, and thus the results can be scaled. This scaling means that the solutions are independent of unit choice, and can then be freely transformed across time, space, and material state. However, this requirement of self-similarity limits the equations of state (EoS) that can be used with the problem, because the EoS must exhibit scaling behavior as well. The Gruneisen equation of state is one such closure model that does not feature scaling properties. The EoS describes the behavior of crystalline solids, making it desirable for use in studies of shock propagation in solid materials. Our work endeavored to find coefficients for the Virial EoS (which consists of a power series expansion about the ideal gas in density) to approximate the Gruneisen EoS. This was done because Virial EoS is inherently scalable, and thus can be used to solve the Guderley problem. With our new form of the Virial EoS, we were able to solve the Guderley problem for a material with Gruneisen-like qualities.

8:39AM M5.00004 Symmetries of the Gas Dynamics Equations Using the Differential Form Method, JOE SCHMIDT, SCOTT RAMSEY, ROY BATY, Los Alamos National Laboratory — A brief review of the theory of exterior differential systems and isovector symmetry analysis methods is presented in the context of the one-dimensional inviscid compressible flow equations. These equations are formulated as an exterior differential system with equation of state (EOS) closure provided in terms of an adiabatic bulk modulus. The scaling symmetry generators and corresponding EOS constraints otherwise appearing in the existing literature are recovered through the application of and invariance under Lie derivative dragging operations.

8:52AM M5.00005 The Radially Symmetric Euler Equations as an Exterior Differential System, ROY BATY, SCOTT RAMSEY, JOSEPH SCHMIDT, Los Alamos National Laboratory — This work develops the Euler equations as an exterior differential system in radially symmetric coordinates. The Euler equations are studied for unsteady, compressible, inviscid fluids in one-dimensional, converging flow fields with a general equation of state. The basic geometrical constructions (for example, the differential forms, tangent planes, jet space, and differential ideal) used to define and analyze differential equations as systems of exterior forms are reviewed and discussed for converging flows. Application of the Frobenius theorem to the question of the existence of solutions to radially symmetric converging flows is also reviewed and discussed. The exterior differential system is further applied to derive and analyze the general family of characteristic vector fields associated with the one-dimensional inviscid flow equations.

9:05AM M5.00006 Effects of spanwise instabilities on the suppression of wake mode in flow over a long rectangular cavity¹, YIYANG SUN, KUNIHIKO TAIRA, LOUIS CATTAFESTA, Florida State University, LAWRENCE UKEILEY, University of Florida — Direct numerical simulation (DNS) and biglobal stability analysis are performed to examine the spanwise effects on the appearance of the so-called wake mode in the flow over long rectangular cavities. The wake mode has been reported to exhibit high-amplitude fluctuations and eject large spanwise vortices in numerical studies, despite its lack of observation in experiments, leaving its existence an open question. The present study focuses on a rectangular cavity flow with aspect ratio of $L/D = 6$, free stream Mach number of $M_\infty = 0.6$ and $Re_D = 502$. The properties of the wake mode are revealed via 2D DNS. From the biglobal stability analysis, the wake mode can be captured with a zero spanwise wavenumber. Furthermore, 3D eigenmodes are calculated with spanwise wavelength $\lambda/D \in [0.5, 2]$. With the knowledge of the features of the wake mode and the 3D eigenmodes, 3D DNS are performed with width-to-depth ratio of $W/D = 1$ and 2. We find the flow exhibits the wake mode with $W/D = 1$ but presents a moderate shear-layer mode with $W/D = 2$. Based on the findings, we argue that the spanwise instabilities in flows over wide cavities redistribute energy from spanwise vortices to streamwise vortical structures, which suppresses the emergence of the wake mode in the 3D cavity flows.

¹This work was supported by the US Air Force Office of Scientific Research (Grant FA9550-13-1-0091).

9:18AM M5.00007 Effects of the Mach number on the evolution of vortex-surface fields in compressible Taylor-Green flows¹, NAIFU PENG, YUE YANG, Peking Univ — We investigate the evolution of vortex-surface fields (VSFs) in viscous compressible Taylor-Green flows. The VSF is applied to the direct numerical simulation of the Taylor-Green flows at a range of Mach numbers from $Ma = 0.6$ to $Ma = 2.2$ for characterizing the Mach-number effects on evolving vortical structures. We find that the dilatation and baroclinic force strongly influence the geometry of vortex surfaces and the energy dissipation rate in the transitional stage. The vortex tubes in compressible flows are less curved than those in incompressible flows, and the maximum dissipation rate occurs earlier in high-Mach-number flows perhaps owing to the conversion of kinetic energy into heat. Moreover, the relations between the evolutionary geometry of vortical structures and flow statistics are discussed.

¹This work has been supported in part by the National Natural Science Foundation of China (Grant Nos. 11522215 and 11521091), and the Thousand Young Talents Program of China.

9:31AM M5.00008 Nonmodal Growth Of Kelvin-Helmholtz Instability In Compressible Flows, MONA KARIMI, SHARATH GIRIMAJI, Texas A&M Univ. — Kelvin-helmholtz instability (khi) is central to the vertical mixing in shear flows and is known to be suppressed in compressible flows. To understand the inhibition of mixing under the influence of compressibility, we analyze the linear growth of khi in the short-time limit using initial value analysis. The evolution of perturbations is studied from a nonmodal standpoint. As the underlying suppression mechanism can be understood by considering primarily linear physics, the effect of compressibility on khi is scrutinized by linear analysis. Then its inferences are verified against direct numerical simulations. It has been demonstrated that compressibility forces the dominance of dilatational, rather than shear, dynamics at the interface of two fluids of different velocities. Within the dilatational interface layer, pressure waves cause the velocity perturbation to become oscillatory [karimi and girimaji, 2016]. Thereupon, the focus is to examine the effect of the initial perturbation wavenumber on the formation of this layer and eventually the degree of khi suppression in compressible flows. We demonstrate that the degree of suppression decreases with the increase the wavenumbers of the initial perturbation. Within the dilatational interface layer, pressure waves cause the velocity perturbation to become oscillatory [karimi and girimaji, 2016]. Thereupon, the focus is to examine the effect of the initial perturbation wavenumber on the formation of this layer and eventually the degree of khi suppression in compressible flows. We demonstrate that the degree of suppression decreases with the increase the wavenumbers of the initial perturbation.

Tuesday, November 22, 2016 8:00AM - 10:10AM – Session M6 Aerodynamics: Flutter, Vibration and Morphing Membranes B114 - Amir Danish-Yazdi, Pennsylvania State University

8:00AM M6.00001 Aerodynamic Coupling between Two Side-by-Side Piezoelectric Harvesters in Grid Turbulence¹, AMIR DANESH-YAZDI, Penn State Univ, Erie, YIANNIS ANDREOPoulos, NIELL ELVIN, 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 piezoelectric beams 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 passive turbulence-generating grid is used to excite various piezoelectric cantilever beam configurations positioned parallel to the flow with different gap widths between the beams at various distances from the grids and for different flow velocities. We observe that the aerodynamic coupling is stronger at higher velocities and when longer beams are paired together and decays exponentially with increasing gap width between the beams. More importantly, however, it is observed that the aerodynamic coupling due to the presence of a second beam greatly improves the energy harvesting process, so much so that the average power generated per beam increases by up to 20 times, potentially allowing for significant power extraction from a random, non-resonant phenomenon such as turbulence.

¹NSF Grant: CBET 1033117

8:13AM M6.00002 Heat Transfer Enhancement in High Performance Heat Sink Channels by Autonomous, Aero-Elastic Reed Fluttering¹, SOURABH JHA, THOMAS CRITTENDEN, ARI GLEZER, Georgia Inst of Tech — Heat transport within high aspect ratio, rectangular mm-scale channels that model segments of a high-performance, air-cooled heat sink is enhanced by the formation of unsteady small-scale vortical motions induced by autonomous, aeroelastic fluttering of cantilevered planar thin-film reeds. The flow mechanisms and scaling of the interactions between the reed and the channel flow are explored to overcome the limits of forced convection heat transport from air-side heat exchangers. High-resolution PIV measurements in a testbed model show that undulations of the reed's surface lead to formation and advection of vorticity concentrations, and to alternate shedding of spanwise CW and CCW vortices. These vortices scale with the reed motion amplitude, and ultimately result in motions of decreasing scales and enhanced dissipation that are reminiscent of a turbulent flow. The vorticity shedding lead to strong enhancement in heat transfer that increases with the Reynolds number of the base flow (e.g., the channel's thermal coefficient of performance is enhanced by 2.4-fold and 9-fold for base flow $Re = 4,000$ and $17,400$, respectively, with corresponding decreases of 50 and 77% in the required channel flow rates). This is demonstrated in heat sinks for improving the thermal performance of low-Re thermoelectric power plant air-cooled condensers, where the global air-side pressure losses can be significantly reduced by lowering the required air volume flow rate at a given heat flux and surface temperature.

¹AFOSR and NSF-EPRI

8:26AM M6.00003 Interaction of Gap Flow With Flapping Dynamics of Two Side-by-Side Elastic Foils, PARDHA S GURUGUBELLI, RAJEEV K JAIMAN, Natl Univ of Singapore — We present a numerical analysis on the interaction between two side-by-side elastic foils with their leading edges clamped and the gap flow between them. We perform systematic parametric direct numerical simulations as a function of nondimensional bending rigidity, $K_B \in [1 \times 10^{-4}, 3 \times 10^{-3}]$ and mass-ratio, $m^* \in [0.05, 0.2]$, for a fixed gap, $d_p = 0.2/L$, at the leading edge and Reynolds number, $Re = 1000$ to explain the underlying physical mechanism behind the in-phase and out-of-phase coupled flapping modes. The parametric simulations show that the parallel foil system exhibits predominant out-of-phase coupling for low mass-ratio $m^* \leq 0.1$ and in-phase coupling for higher mass-ratios $m^* > 0.1$. We also show that the two side-by-side elastic foils always exhibit out-of-phase coupling initially irrespective of whether the fully developed flapping show out-of-phase or in-phase coupled mode. Finally, we show that the transition from the initial out-of-phase to stable in-phase is characterized by loss of gap flow symmetric stability to undergo oscillations at the gap exit.

8:39AM M6.00004 Comparison of driven and simulated “free” stall flutter in a wind tunnel¹, ETHAN CULLER, JOHN FARNSWORTH, Univ of Colorado - Boulder, CASEY FAGLEY, JURGEN SEIDEL, United States Air Force Academy — Stall flutter and dynamic stall have received a significant amount of attention over the years. To experimentally study this problem, the body undergoing stall flutter is typically driven at a characteristic, single frequency sinusoid with a prescribed pitching amplitude and mean angle of attack offset. This approach allows for testing with repeatable kinematics, however it effectively decouples the structural motion from the aerodynamic forcing. Recent results suggest that this driven approach could misrepresent the forcing observed in a “free” stall flutter scenario. Specifically, a dynamically pitched rigid NACA 0018 wing section was tested in the wind tunnel under two modes of operation: (1) Cyber-Physical where “free” stall flutter was physically simulated through a custom motor-control system modeling a torsional spring and (2) Direct Motor-Driven Dynamic Pitch at a single frequency sinusoid representative of the cyber-physical motion. The time-resolved pitch angle and moment were directly measured and compared for each case. It was found that small deviations in the pitch angle trajectory between these two operational cases generate significantly different aerodynamic pitching moments on the wing section, with the pitching moments nearly 180° out of phase in some cases.

¹This work is supported by the Air Force Office of Scientific Research through the Flow Interactions and Control Program and by the National Defense Science and Engineering Graduate Fellowship Program.

8:52AM M6.00005 Water channel experiments of a novel fully-passive flapping-foil turbine¹, MATTHIEU BOUDREAU, GUY DUMAS, Univ of Laval, MOSTAFA RAHIMPOUR, PETER OSHKAI, Univ of Victoria — Experiments have been conducted to assess the performances of a fully-passive flapping-foil hydrokinetic turbine for which the blade's motions are stemming from the interaction between the blade's elastic supports (springs and dampers) and the flow field. Previous numerical studies conducted by Peng & Zhu (2009) and Zhu (2012) have proved that a simplified version of such a turbine can extract a substantial amount of energy from the flow while offering the potential to greatly simplify the complex mechanical apparatus needed to constrain and link the blade's pitching and heaving motions in the case of the more classical flapping-foil turbine (e.g., Kinsey et al., 2011). Based on the promising numerical investigations of Veilleux (2014) and Veilleux & Dumas (2016), who proposed a more general version of this novel concept, a prototype has been built and tested in a water channel at a chord Reynolds number of 17,000. Periodic motions of large amplitudes have been observed leading to interesting energy harvesting efficiencies reaching 25% for some specific sets of structural parameters. The sensitivity of the turbine's dynamics to each of the seven structural parameters appearing in the equations of motion has been experimentally evaluated around a case close to the optimal one.

¹Financial support from the Natural Sciences and Engineering Research Council of Canada (NSERC) is gratefully acknowledged by the authors.

9:05AM M6.00006 Coupled-Mode Flutter of Bending-Bending Type in Highly-Flexible Uniform Airfoils¹ , PARIYA POURAZARM, YAHYA MODARRES-SADEGHI, University of Massachusetts Amherst —

We study the behavior of a highly flexible uniform airfoil placed in wind both numerically and experimentally. It is shown that for a non-rotating highly-flexible cantilevered airfoil, placed at very small angles of attack (less than 1 degree), the airfoil loses its stability by buckling. For slightly higher angles of attack (more than 1 degree) a coupled-mode flutter in which the first and the second flapwise modes coalesce toward a flutter mode is observed, and thus the observed flutter has a bending-bending nature. The flutter onset and frequency found experimentally matched the numerical predictions. If the same airfoil is forced to rotate about its fixed end, the static deflection decreases and the observed couple-mode flutter becomes of flapwise-torsional type, same as what has already been observed for flutter of rotating wind turbine blades.

¹The support provided by the National Science Foundation, CBET-1437988, is greatly acknowledged.

(Institute of Fluid Mechanics of Toulouse)

UMR5502, France

(Laboratory on plasma and conversion of energy)

9:18AM M6.00007 Experimental investigation of the dynamics of a hybrid morphing wing: time resolved particle image velocimetry and force measures , GURVAN JODIN

IMFT, JOHANNES SCHELLER, IMFT - MIT, JEAN-FRANÇOIS ROUCHON, Laplace, MARIANNA BRAZA, IMFT, MIT COLLABORATION, LAPLACE COLLABORATION — A quantitative characterization of the effects obtained by high frequency amplitude trailing edge actuation is performed. Particle image velocimetry, as well as pressure and aerodynamic force measurements, are carried out on an airfoil model. This hybrid morphing wing model is equipped with both trailing edge piezoelectric-actuators and camber control memory alloy actuators. It will be shown that this actuation allows for an effective manipulation of the wake turbulent structures. Proper domain analysis and proper orthogonal decomposition show that proper actuating reduces the energy dissipation by favoring more vortical structures. This modification in the airflow dynamics eventually allows for a tapering of the wake thickness compared to the baseline configuration. Hence, drag reductions relative to the non-actuated trailing edge configuration are observed.

UMR5213, France ¹Massachusetts Institute of Technology

9:31AM M6.00008 Unsteady aerodynamics of membrane wings with adaptive compliance , JILLIAN KISER¹, KENNETH BREUER, Brown University —

Membrane wings are known to provide superior aerodynamic performance at low Reynolds numbers ($Re = 10^4 - 10^5$), primarily due to passive shape adaptation to flow conditions. In addition to this passive deformation, active control of the fluid-structure interaction and resultant aerodynamic properties can be achieved through the use of dielectric elastomer actuators as the wing membrane material. When actuated, membrane pretension is decreased and wing camber increases. Additionally, actuation at resonance frequencies allows additional control over wing camber. We present results using synchronized (i) time-resolved particle image velocimetry (PIV) to resolve the flow field, (ii) 3D direct linear transformation (DLT) to recover membrane shape, (iii) lift/drag/torque measurements and (iv) near-wake hot wire anemometry measurements to characterize the fluid-structure interactions. Particular attention is paid to cases in which the vortex shedding frequency, the membrane resonance, and the actuation frequency coincide. In quantitatively examining both flow field and membrane shape at a range of actuation frequencies and vortex shedding frequencies, this work seeks to find actuation parameters that allow for active control of boundary layer separation over a range of flow conditions.

¹Also at Naval Undersea Warfare Center, Division Newport

9:44AM M6.00009 Unsteady fluid-structure interactions with a heaving compliant membrane wing , GALI ALON TZEZANA, KENNETH BREUER, Brown University —

Membrane wings have been shown to provide some benefits over rigid wings at the low Reynolds number regime ($Re \sim 10^3$ to 10^5), specifically improved thrust in flapping flight. Here we present results from a theoretical framework used to characterize the unsteady aeroelastic behavior of compliant membrane wings executing a heaving motion. An analytical model is developed using 2D unsteady thin airfoil theory, coupled with an unsteady membrane equation. Chebyshev collocation methods are used to solve the coupled system efficiently. The model is used to explore the effects of wing compliance, inertia (including added mass effect) and flapping kinematics on the aerodynamic performance, identifying optimal conditions for maximum thrust and propulsive efficiency. A resonant frequency of the coupled system is identified and characterized for different fluid-structure interaction regimes. Extensions to pitching kinematics are also discussed.

9:57AM M6.00010 Combining spanwise morphing, inline motion and model based optimization for force magnitude and direction control , JOHANNES SCHELLER, MIT, MARIANNA BRAZA, IMFT, MICHAEL TRIANTAFYLLOU, MIT —

Bats and other animals rapidly change their wingspan in order to control the aerodynamic forces. A NACA0013 type airfoil with dynamically changing span is proposed as a simple model to experimentally study these biomimetic morphing wings. Combining this large-scale morphing with inline motion allows to control both force magnitude and direction. Force measurements are conducted in order to analyze the impact of the 4 degree of freedom flapping motion on the flow. A blade-element theory augmented unsteady aerodynamic model is then used to derive optimal flapping trajectories.

Tuesday, November 22, 2016 8:00AM - 9:57AM –
Session M7 Flow Control: Plasma, Actuators and Synthetic Jets B115 - Michael Amitay,
Rensselaer Polytechnic Institute

8:00AM M7.00001 More Insight of Piezoelectric-based Synthetic Jet Actuators¹

KEVIN HOUSLEY, MICHAEL AMITAY, Rensselaer Polytechnic Institute — Increased understanding of the internal flow of piezoelectric-based synthetic jet actuators is needed for the development of specialized actuator cavity geometries to increase jet momentum coefficients and tailor acoustic resonant frequencies. Synthetic jet actuators can benefit from tuning of the structural resonant frequency of the piezoelectric diaphragm(s) and the acoustic resonant frequency of the actuator cavity such that they experience constructive coupling. The resulting coupled behavior produces increased jet velocities. The ability to design synthetic jet actuators to operate with this behavior at select driving frequencies allows for them to be better used in flow control applications, which sometimes require specific jet frequencies in order to utilize the natural instabilities of a given flow field. A parametric study of varying actuator diameters was conducted to this end. Phase-locked data were collected on the jet velocity, the cavity pressure at various locations, and the three-dimensional deformation of the surface of the diaphragm. These results were compared to previous analytical work on the interaction between the structural resonance of the diaphragm and the acoustic resonance of the cavity.

¹Funded by the Boeing Company

8:13AM M7.00002 Vectoring of parallel synthetic jets: A parametric study , TIM BERK,

GUILLAUME GOMIT, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton — The vectoring of a pair of parallel synthetic jets can be described using five dimensionless parameters: the aspect ratio of the slots, the Strouhal number, the Reynolds number, the phase difference between the jets and the spacing between the slots. In the present study, the influence of the latter four on the vectoring behaviour of the jets is examined experimentally using particle image velocimetry. Time-averaged velocity maps are used to study the variations in vectoring behaviour for a parametric sweep of each of the four parameters independently. A topological map is constructed for the full four-dimensional parameter space. The vectoring behaviour is described both qualitatively and quantitatively. A vectoring mechanism is proposed, based on measured vortex positions.

We acknowledge the financial support from the European Research Council (ERC grant agreement no. 277472).

8:26AM M7.00003 Experimental and Computational Investigation of a Dual-Throat Thrust Vectoring Nozzle¹ , JOHN FARNSWORTH, NAVEEN PENMETSA, RYAN STARKEY, Univ of Colorado - Boulder —

The dual-throat fluidic thrust vectoring nozzle is of particular interest because of its ability to provide large vector angles with minimal losses in thrust. This work investigated the performance of a dual-throat fluidic thrust vectoring nozzle for three secondary injection geometries: two spanwise oriented rectangular slots of two thicknesses, and a single spanwise oriented array of circular holes. Initial testing of the nozzles at a nozzle pressure ratio of two showed that the presence of the injection geometry alone influenced the baseline vector angle of the flow. With the introduction of secondary injection, the thinner rectangular slot was found to outperform the two other configurations at low injection percentages, while secondary injection through an array of holes trended higher at higher injection percentages. Using the experimental and computational data collected during this study, a method was developed to predict vector angle from the wall static-pressure distributions internal to the nozzle. The predicted thrust-vector angle matched the angles measured from schlieren photographs to within the measurement uncertainty across the range of injection mass flow rates tested.

¹This work was supported by the University of Colorado Boulder Engineering Excellence Fund.

8:39AM M7.00004 Towards In-Flight Applications? - Requirements on the Dielectric Barrier Discharge (DBD) Plasma Actuator (PA) , JOCHEN KRIEGSEIS, Karlsruhe Institute of Technology, BERNHARD SIMON, Technische Universität Darmstadt, SVEN GRUNDMANN, University of Rostock —

Most of todays flow control (FC) efforts with DBD show a rather one-sided picture. Typically, either the discharge properties are discussed extensively or FC achievements are reported. The former group of contributions only pays limited attention to implications and consequences of most characteristics with respect to subsequent control steps for successful DBD-based FC - the latter group mostly ignores changing discharge properties, thus varying control authority for the respective applications when changes of environment, PA health state or simply a varied angle-of-attack are to be considered. In addition, there still remains a fair bit of uncertainty regarding a universal PA-evaluation metric, such that some of the most promising quantities/characteristics for successful controller operation remain largely untouched from the community. The purpose of the present work is to outline the requirement profile of PAs in one coherent story starting from electrical issues all the way down the road to in-flight FC success, where particular emphasis is placed on the interplay of the involved subtopics. It is hypothesized that such a clear guideline is the only way to advance beyond the present level of lab studies, where there still is an obvious lack of real flight applications.

8:52AM M7.00005 Study of shock shape and strength as a function of plasma energy using background oriented schlieren and shadowgraph. , BHAVINI SINGH, LALIT RAJENDRAN, MATTHEW GIARRA, SALLY BANE, PAVLOS VLACHOS, None —

The formation of a spark is a random, chaotic process. The flow field generated by this spark can be used in flow control and plasma assisted combustion applications. In order to understand the flow field some time after spark discharge (approximately 1 microsecond), it is important to observe the shape and strength of the shockwave immediately following the plasma discharge. It is also important to understand the effect that the energy deposited in the spark gap has on the shock strength and shock shape. We therefore propose a background oriented schlieren (BOS) technique to measure density gradients associated with the spark discharge and hence quantify shock strength. Simultaneous shadowgraph measurements will be used to observe the shape of the shock and compare it with the reconstructed density gradients obtained from BOS measurements.

9:05AM M7.00006 Pulsed-DC DBD Plasma Actuators¹ , ALAN DUONG, RYAN MCGOWAN, KATHERINE DISSER, THOMAS CORKE², ERIC MATLIS³, University of Notre Dame —

A new powering system for dielectric barrier discharge (DBD) plasma actuators that utilizes a pulsed-DC waveform is presented. The plasma actuator arrangement is identical to most typical AC-DBD designs with staggered electrodes that are separated by a dielectric insulator. However instead of an AC voltage input to drive the actuator, the pulsed-DC utilizes a DC voltage source. The DC source is supplied to both electrodes, and remains constant in time for the exposed electrode. The DC source for the covered electrode is periodically grounded for very short instants and then allowed to rise to the source DC level. This process results in a plasma actuator body force that is significantly larger than that with an AC-DBD at the same voltages. The important characteristics used in optimizing the pulsed-DC plasma actuators are presented. Time-resolved velocity measurements near the actuator are further used to understand the underlying physics of its operation compared to the AC-DBD.

¹Supported by NASA Glenn RC

²APS Fellow

³APS Member

9:18AM M7.00007 Turbulent Mixing Layer Control using Ns-DBD Plasma

Actuators¹, ASHISH SINGH, JESSE LITTLE, University of Arizona — A low speed turbulent mixing layer ($Re_{\theta_0}=1282$, $U_1/U_2 = 0.28$ and $U_2 = 11.8m/s$) is subject to nanosecond pulse driven dielectric barrier discharge (ns-DBD) plasma actuation. The forcing frequency corresponds to a Strouhal number (St) of 0.032 which is the most amplified frequency based on stability theory. Flow response is studied as a function of the pulse energy, the energy input time scale (carrier frequency) and the duration of actuation (duty cycle). It is found that successful actuation requires a combination of forcing parameters. An evaluation of the forcing efficacy is achieved by examining different flow quantities such as momentum thickness, vorticity and velocity fluctuations. In accordance with past work, a dependence is found between the initial shear layer thickness and the energy coupled to the flow. More complex relationships are also revealed such as a limitation on the maximum pulse energy which yields control. Also, the pulse energy and the carrier frequency (inverse of period between successive pulses) are interdependent whereby an optimum exists between them and extreme values of either parameter is inconsonant with the control desired. These observations establish a rich and complex process behind ns-DBD plasma actuation.

¹Air Force Office of Scientific Research (FA9550-12-1-0044)

9:31AM M7.00008 Transitory Control of the Aerodynamic Loads on an Airfoil in Dynamic Pitch and Plunge¹, YUEHAN TAN, THOMAS CRITTENDEN, ARI GLEZER, Georgia Inst of Tech — Transitory

control and regulation of trapped vorticity concentrations are exploited in wind tunnel experiments for control of the aerodynamic loads on an airfoil moving in time-periodic 2-DOF (pitch and plunge) beyond the dynamic stall margin. Actuation is effected using a spanwise array of integrated miniature chemical (combustion based) high-impulse actuators that are triggered intermittently relative to the airfoil's motion. Each actuation pulse has sufficient control authority to alter the global aerodynamic performance throughout the motion cycle on a characteristic time scale that is an order of magnitude shorter than the airfoil's convective time scale. The effects of the actuation on the aerodynamic characteristics of the airfoil are assessed using time-dependent measurements of the lift force and pitching moment coupled with time-resolved particle image velocimetry that is acquired phased-locked to the motion of the airfoil. It is shown that the aerodynamic loads can be significantly altered using actuation programs based on multiple actuation pulses during the time-periodic pitch/plunge cycle. Superposition of such actuation programs leads to enhancement of cycle lift and pitch stability, and reduced cycle hysteresis and peak pitching moment.

¹Supported by GT-VLRCOE

9:44AM M7.00009 Closed-Loop Flow Control of the Coupled Wake Dynamics and Aerodynamic Loads of a Freely-Pivoting 3-DOF Bluff Body¹, T. LAMBERT, B. VUKASINOVIC, A.

GLEZER, Georgia Institute of Technology — The motion of an axisymmetric bluff body model that is free to pivot in pitch, yaw, and roll in a uniform stream in response to flow-induced aerodynamic loads is controlled in wind tunnel experiments using fluidic actuation. The model is attached to an upstream, wire-supported short streamwise sting through a low-friction hinge, and each of the support wires is individually-controlled by a servo actuator through an in-line load cell. The aerodynamic loads on the body, and thereby its motion, are controlled through fluidic modification of its aerodynamic coupling to its near wake using four independent aft mounted synthetic jet actuators that effect azimuthally-segmented flow attachment over the model's tail end. The effects of actuation-induced, transitory changes in the model's aerodynamic loads are measured by its motion response using motion tracking, while the coupled evolution of the near-wake is captured by high-speed stereo PIV. Flow control authority is demonstrated by feedback-controlled manipulation of the model's dynamic response, and dynamic mode decomposition (DMD) of the wake is used to characterize changes in the wake structure and stability. It is shown that this flow control approach can modify the stability and damping of the model's motion (e.g., suppression or amplification of its natural oscillations), and impose desired directional attitude.

¹Supported by the ARO.

Tuesday, November 22, 2016 8:00AM - 10:10AM –

Session M9 General Fluid Dynamics: Viscous Flows B117 - Randy Ewoldt, University of Illinois, Urbana-Champaign

8:00AM M9.00001 A validated computational model for the design of surface textures in full-film lubricated sliding, JONATHON SCHUH, YONG HOON LEE, JAMES ALLISON, RANDY EWOLDT, University of Illinois at Urbana-Champaign — Our recent experimental work showed that asymmetry is needed for surface textures to decrease friction in full-film lubricated sliding (thrust bearings) with Newtonian fluids; textures reduce the shear load and produce a separating normal force. The sign of the separating normal force is not predicted by previous 1-D theories. Here we model the flow with the Reynolds equation in cylindrical coordinates, numerically implemented with a pseudo-spectral method. The model predictions match experiments, rationalize the sign of the normal force, and allow for design of surface texture geometry. To minimize sliding friction with angled cylindrical textures, an optimal angle of asymmetry β exists. The optimal angle depends on the film thickness but not the sliding velocity within the applicable range of the model. The model has also been used to optimize generalized surface texture topography while satisfying manufacturability constraints.

8:13AM M9.00002 Viscous erosion at low Reynolds number, WILLIAM MITCHELL, SAVERIO SPAGNOLIE, University of Wisconsin - Madison — We study the shape evolution of immersed particles in a viscous fluid under several flow configurations, including uniform background flows and shear flows in wall-bounded or free domains. The surface recedes proportionally to local shear stress, which we compute using a new traction integral formulation of Newtonian Stokes flow. This opens the door to efficient numerical simulation of the evolving particle geometry. Analytical predictions from reduced-order models are then compared against the numerical simulations. For the case of particles held fixed against an oncoming background flow, the theory predicts the finite time required for complete particle dissolution as well as the emergence and locations of sharp corners on the eroding bodies. Simulations involving force- and torque-free particles and multibody systems are also presented.

8:26AM M9.00003 Measurement of steady and transient liquid coiling with high-speed video and digital image processing, FRANK AUSTIN MIER, RAJ BHAKTA, NICOLAS CASTANO, JOSHUA THACKRAH, TYLER MARQUIS, JOHN GARCIA, MICHAEL HARGATHER, New Mexico Tech — Liquid coiling occurs as a gravitationally-accelerated viscous fluid flows into a stagnant reservoir causing a localized accumulation of settling material, commonly designated as stack. This flow is broadly characterized by a vertical rope of liquid, the tail, flowing into the stack in a coiled motion with frequency defined parametrically within four different flow regimes. These regimes are defined as viscous, gravitational, inertial-gravitational, and inertial. Relations include parameters such as flow rate, drop height, rope radius, gravitational acceleration, and kinematic viscosity. While previous work on the subject includes high speed imaging, only basic and often averaged measurements have been taken by visual inspection of images. Through the implementation of additional image processing routines in MATLAB, time resolved measurements are taken on coiling frequency, tail diameter, stack diameter and height. Synchronization between a high speed camera and stepper motor driven syringe pump provides accurate correlation with flow rate. Additionally, continuous measurement of unsteady transition between flow regimes is visualized and quantified. This capability allows a deeper experimental understanding of processes involved in the liquid coiling phenomenon.

8:39AM M9.00004 Angular dynamics of small crystals in viscous flows¹, JOHAN FRIES, JONAS EINARSSON, BERNHARD MEHLIG, Department of Physics, Gothenburg University — The angular dynamics of a very small ellipsoidal particle in a viscous flow decouples from its translational dynamics, and the particle angular velocity is given by Jeffery's theory. It is known that cuboid particles share these properties. In the literature a special case is most frequently discussed, that of axisymmetric particles, with a continuous rotational symmetry. Here we compute the angular dynamics of crystals that possess a discrete rotational symmetry and certain mirror symmetries, but that do not have a continuous rotational symmetry. We give examples of such particles that nevertheless obey Jeffery's theory. But there are other examples where the angular dynamics is determined by a more general equation of motion.

¹Vetenskapsrådet [grant number 2013-3992], Formas [grant number 2014-585], Bottlenecks for particle growth in turbulent aerosols from the Knut and Alice Wallenberg Foundation, Dnr. KAW 2014.0048, MPNS COST Action MP1305 Flowing matter.

8:52AM M9.00005 Reverse Fluid Transport Due to Boundary Pulsations, MIKHAIL COLOMA, DAVID SCHAFER, PAUL CHIAROT, PETER HUANG, Binghamton University — We investigate a reverse fluid transport mechanism consisting of peristaltic flow and boundary wave reflections. The reverse flow occurs in a rectangular conduit aligned in parallel between two cylindrical channels embedded in an elastic PDMS medium. The pulsating flow in the cylindrical channels, driven by a peristaltic pump, deform the PDMS medium and induce a pulsating flow in the rectangular conduit. Waveforms along the conduit boundaries, and their transmission and reflections, can be controlled by changing the PDMS rigidity. Our results show that while the overall wave propagation direction is in the forward direction, a reverse flow in the rectangular conduit can be preferentially induced by varying the elastic rigidity in one of the cylindrical channels. We study the overall flow velocity and direction under various PDMS rigidities. The identified set of experimental parameters that leads to a reverse flow will provide insights in understanding metabolic waste transport within the arterial walls in the brain.

9:05AM M9.00006 Oscillatory slip flow past a spherical inclusion embedded in a Brinkman medium, D. PALANIAPPAN, Texas AM University, Corpus Christi — Non-steady flow past an impermeable sphere embedded in a porous medium is investigated based on Brinkman model with Navier slip conditions. Exact analytic solution for the stream-function - involving modified Bessel function of the second kind - describing the slow oscillatory flow around a rigid spherical 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 control the flow fields and physical quantities in the entire flow domain. Local streamlines for fixed times demonstrate the variations in flow patterns. Closed form expressions for the tangential velocity profile, wall shear stress, and the force acting on the sphere are computed and compared with the existing results. It is noted that the slip parameter in the range $0 \leq \xi \leq 0.5$ has a significant effect in reducing the stress and force. The steady-state velocity overshoot behavior in the vicinity of the sphere is re-iterated. 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.

9:18AM M9.00007 Buckling and stretching of thin viscous sheets, DOIREANN O'KIELY, CHRIS BREWARD, IAN GRIFFITHS, PETER HOWELL, Mathematical Institute, University of Oxford, ULRICH LANGE, Schott AG — Thin glass sheets are used in smartphone, battery and semiconductor technology, and may be manufactured by producing a relatively thick glass slab and subsequently redrawing it to a required thickness. The resulting sheets commonly possess undesired centerline ripples and thick edges. We present a mathematical model in which a viscous sheet undergoes redraw in the direction of gravity, and show that, in a sufficiently strong gravitational field, buckling is driven by compression in a region near the bottom of the sheet, and limited by viscous resistance to stretching of the sheet. We use asymptotic analysis in the thin-sheet, low-Reynolds-number limit to determine the centerline profile and growth rate of such a viscous sheet.

9:31AM M9.00008 A reciprocal theorem for convective heat and mass transfer in Stokes and potential flows, HASSAN MASOUD, VAHID VANDADI, SAEED JAFARI KANG, University of Nevada, Reno — In the study of convective heat and mass transfer from a particle, key quantities of interest are usually the average rate of transfer and the mean distribution of the scalar (i.e., temperature or concentration) at the particle surface. Calculating these quantities using conventional equations requires detailed knowledge of the scalar field, which is available predominantly for problems involving uniform scalar and flux boundary conditions. Here, we derive a reciprocal relation between two diffusing scalars that are advected by oppositely driven Stokes or potential flows whose streamline configurations are identical. This relation leads to alternative expressions for the aforementioned average quantities based on the solution of the scalar field for uniform surface conditions.

9:44AM M9.00009 ABSTRACT WITHDRAWN —

9:57AM M9.00010 ABSTRACT WITHDRAWN —

Tuesday, November 22, 2016 8:00AM - 10:10AM —
Session M10 DFD GPC: Convection and Buoyancy Driven Flows: Confined Flows
B118-119 - Elizabeth Camp, Portland State University
mixing" (Log Number DFD16-2016-000953).

8:00AM M10.00001 Stratified shear flow in an inclined duct: near-instantaneous velocity and density measurements¹, JAMIE PARTRIDGE, ADRIEN LEFAUVE, STUART DALZIEL, Univ of Cambridge — We present results from a new experimental setup to study the exchange flow in an inclined square duct connecting two reservoirs containing fluids of different densities. This system can exhibit stratified shear wave motions, and has a distinct parameter regime above which turbulence is triggered and progressively fills a larger fraction of the duct. To probe these intrinsically 3D flows, we have developed a setup in which a traversing laser sheet allows us to obtain near-instantaneous 3D velocity and density fields. Three components of the flow are measured on successive 2D planes using stereo particle image velocimetry (PIV) with density information obtained simultaneously via laser induced fluorescence (LIF).

Thank you

¹Supported by EPSRC Programme Grant EP/K034529/1 entitled Mathematical Underpinnings of Stratified Turbulence

8:13AM M10.00002 Stratified shear flow in an inclined duct: coherent structures and mixing¹, ADRIEN LEFAUVE, JAMIE PARTRIDGE, STUART DALZIEL, PAUL LINDEN, Univ of Cambridge — We present laboratory experiments on the exchange flow in an inclined square duct connecting two reservoirs at different densities. This system generates and maintains a stratified shear flow, which can be laminar, wavy or turbulent depending on the density difference and inclination angle. It is believed that the mean dissipation is set by the angle, and that high buoyancy Reynolds numbers (i.e. turbulent intensity) can be maintained, making this system suited for the study of continuously forced stratified turbulence. The talk will focus on the analysis of time-resolved, near-instantaneous 3D velocity and density data obtained by stereo particle image velocimetry (PIV) and laser induced fluorescence (LIF). This data allow for the visualisation of 3D coherent structures as well as turbulent mixing properties, which are key in understanding the dynamics of stratified turbulence.

¹Supported by EPSRC Programme Grant EP/K034529/1 entitled Mathematical Underpinnings of Stratified Turbulence

8:26AM M10.00003 Harmonic Forcing on the Stratified Square Lid Driven Cavity, JASON YALIM, BRUNO WELFERT, JUAN LOPEZ, STEPHANIE TAYLOR, Arizona State University — Stratified fluids that are driven at an interface, such as oceans or seas, can be periodically driven by wind. As a canonical flow, the square lid driven cavity with a harmonic forcing and a linear temperature gradient serves as a idealized model. Resonances of the harmonic forcing with the internal modes of the system aide energy transfer from the surface to the bulk, leading to interesting dynamics. Using a numerical spectral collocation method, the internal waves of the system are investigated, including their possible interaction and annihilation.

8:39AM M10.00004 Numerical investigation of the onset of centrifugal buoyancy in a rotating cavity¹, DIOGO B. PITZ, OLAF MARXEN, JOHN CHEW, University of Surrey — Buoyancy-induced flows in a differentially heated rotating annulus present a multitude of dynamics when control parameters such as rotation rate, temperature difference and Prandtl number are varied. Whilst most of the work in this area has been motivated by applications involving geophysics, the problem of buoyancy-induced convection in rotating systems is also relevant in industrial applications such as the flow between rotating disks of turbomachinery internal air systems, in which buoyancy plays a major role and poses a challenge to accurately predict temperature distributions and heat transfer rates. In such applications the rotational speeds involved are very large, so that the centrifugal accelerations induced are much higher than gravity. In this work we perform direct numerical simulations and linear stability analysis of flow induced by centrifugal buoyancy in a sealed rotating annulus of finite gap with flat end-walls, using a canonical setup representative of an internal air system rotating cavity. The analysis focuses on the behaviour of small-amplitude disturbances added to the base flow, and how those affect the onset of Rossby waves and, ultimately, the transition to a fully turbulent state where convection columns no longer have a well-defined structure.

¹Diogo B. Pitz acknowledges the financial support from the Capes foundation through the Science without Borders program.

8:52AM M10.00005 The oscillation modes of large-scale circulation in turbulent Rayleigh-Bnard convection in a cubic container, DANDAN JI, KUNLUN BAI, ERIC BROWN, Yale University — We present measurement of the large-scale circulation (LSC) of turbulent Rayleigh- Bnard convection of a cubic cell. We measured the LSC orientation angle θ_0 and off-center displacement angle α . We found the LSC oscillates around one corner. The oscillation frequency matches the turnover time and the natural frequency based on the geometry of the cell predicted by the stochastic model presented by Brown and Ahlers (Phys. Fluids, 2008), however the model with advection which was used to predict oscillations in cylinders predicts the system is over-damped. The structure of the LSC breaks the symmetry of the cube.

9:05AM M10.00006 Statistical convergence and the effect of large-scale motions on turbulent Rayleigh-Bnard convection in a cylindrical domain with 6.3 aspect ratio¹, PHILIP SAKIEVICH, Arizona State University, YULIA PEET, RONALD ADRIAN, Arizona State University — At high Rayleigh numbers in moderate aspect-ratio cylindrical domains turbulent Rayleigh-Bnard convection (RBC) exhibits coherent large-scale motions with patterns like some of those found in laminar flow. In this work we show how the patterns of the largest scales in turbulent RBC affect the bias and convergence of the flow statistics at aspect-ratio 6.3 (diameter/ height). Large scale motions influence two of the finite-time statistical means inherent properties: 1) the orientation of the patterns changes so slowly that it may appear almost fixed during a finite averaging time interval, thereby imbedding a preferred azimuthal direction in the sampled data; 2) they also have at least two states associated with the occurrence of up and down motions near the center of the convection cell. We will present a novel technique for triggering additional states of RBC in DNS simulations that are targeted for improving the statistical convergence of the flow. This technique gently perturbs the flow so that the new variations of the large scale patterns can be sampled.

¹Funding through U. S. National Science Foundation Grants CBET-1335731, CMMI-1250124 and XSEDE research allocation TG-CTS150039

9:18AM M10.00007 Aspect-ratio dependence of the large-scale circulation in Rayleigh-Bénard convection with weak rotation¹, GUENTER AHLERS, Department of Physics, UCSB, PING WEI, School of Aerospace engineering and applied mechanics, Tongji U, Shanghai, China — We report measurements for slowly rotating turbulent thermal convection in cylindrical samples with aspect ratios $\Gamma = 1.0$ and 2.0 for a Prandtl number $Pr = 12.3$. The results are for the large-scale circulation (LSC) strength δ , Fourier-energy E_{tot} , and relative flow strength S , as well as for two Reynolds numbers Re_{ret} and Re_{sl} , for the Nusselt number Nu , and for the vertical temperature gradient $\partial\Theta/\partial z$ at the sample center. They cover the Rayleigh-number range $3 \times 10^{10} \leq Ra \leq 4 \times 10^{11}$ and the inverse Rossby-number range $0 \leq 1/Ro \leq 1/Ro_c$. Nu , E_{tot} , S , and $\partial\Theta/\partial z$ showed sharp transitions at $1/Ro_c$. The LSC underwent retrograde rotation with period τ_{ret} and showed sloshing oscillations with period $\tau_{sl} \ll \tau_{ret}$. At constant Ra and $1/Ro$ δ grew and decayed with a period equal to τ_{ret} . We found that $Re_{ret} \equiv 4L^2/\tau_{ret}\nu \propto Ra^{0.65}$ ($\propto Ra^{0.50}$) for $\Gamma = 1.0$ ($\Gamma = 2.0$) (ν is the kinematic viscosity and L the sample height) and $Re_{sl} \equiv 4L^2/\tau_{sl}\nu \propto Ra^{0.50}$ ($\propto Ra^{0.42}$) for $\Gamma = 1.0$ ($\Gamma = 2.0$).

¹Supported by NSF grant DMR11-58514

9:31AM M10.00008 Aspect-ratio dependence of small-scale temperature properties in turbulent Rayleigh-Bénard convection¹, PING WEI, School of Aerospace engineering and applied mechanics, Tongji U, Shanghai, GUENTER AHLERS, Department of Physics, UCSB, CA, USA — We report measurements of the variance σ^2 , skewness S , and kurtosis K of temperature fluctuations in turbulent Rayleigh-Bénard convection of a fluid with Prandtl number $Pr = 12.3$ in cylindrical samples with aspect ratios $\Gamma = D/L$ (D is the diameter and L the height) of $0.50, 1.00$ and 2.00 in the Rayleigh-number range $6 \times 10^9 \leq Ra \leq 2 \times 10^{12}$. The measurements were primarily for the radial positions $\xi = 1.00$ (along the sample center line) and $\xi = 0.063$ (near the side wall) at several vertical locations z/L . For all Γ we found that σ^2 could be fitted by $\sigma^2 \sim (z/L)^{-\zeta}$ with $\zeta \simeq 0.7$ near the side wall and $\zeta \simeq 1.0$ along the sample center line ($\xi = 1.00$). At the sample center and for $\Gamma = 1$, the temperature probability distribution was very close to a Laplace distribution, with K close to 6 independent of Ra . However, for $\Gamma = 0.5$ the distribution was intermediate between Gaussian and Laplace, with K close to 4 and also independent of Ra . For $\Gamma = 2$ the distribution was close to Gaussian near the peak but had exponential tails, yielding K values that decreased from about 6 to about 4 as Ra increased.

¹Supported by NSF grant DMR11-58514

9:44AM M10.00009 Experimental container shape dependence and heat transport scaling of Rayleigh-Bénard convection of high-Prandtl-number fluids, STEPHEN JOHNSTON, Georgia Institute of Technology, ENRICO FONDA, KATEPALLI R. SREENIVASAN, New York University, DEVESH RANJAN, Georgia Institute of Technology — Both experiments and simulations on Rayleigh-Bénard convection with fluids of Prandtl numbers 5 and below have shown that the container shape influences the flow structure. Here, we investigate similar dependences of convection of fluids with Prandtl numbers of up to 10^4 . The convection cells have aspect ratio of order unity, and we use cubic and cylindrical shapes. Visual analysis using a noninvasive photochromic dye technique indicates the distinct large-scale flow patterns in both square and cylindrical test cells. The stability of these flow patterns is explored. Also presented are results on the Nusselt-Rayleigh scaling for moderate Rayleigh numbers. References: Z. A. Daya and R. E. Ecke, Phys. Rev. Lett. 87, 184501 (2001) N Foroozani, JJ Niemela, V Armenio, KR Sreenivasan, Phys. Rev. E 90, 063003 (2014)

9:57AM M10.00010 Different motion modes of a mobile plate on top of a thermally convecting fluid, YADAN MAO, Institute of Geophysics and Geomatics, China University of Geosciences, Wuhan, China, JIN-QIANG ZHONG, School of Physics Sciences and Engineering, Tongji University, Shanghai, China, JUN ZHANG, Courant Institute, New York University, New York , USA and NYU Shanghai, China — Numerical simulations are conducted to model the dynamics of a mobile, insulating plate floating on top of a Rayleigh-Bénard convecting fluid with infinite Prandtl number in a two dimensional rectangular domain, which is roughly analogous to the geological model of continent drift over mantle. We focus on the effect of plate size on the dynamic feedback between the plate and the underlying convection. Four different modes of coupling are revealed as plate size varies. Among them, two transient stable modes are identified: 1. a very small plate tends to linger for long time over a cold downwelling bordering two counter-rotating convection cells; 2. a relatively small plate sometimes lingers over an upwelling plume bordering two convection cells with cold downwellings on the edges of the plate. A relatively large plate rides on a moving convection cell and oscillates periodically between the two end walls. A very large plate executes only small excursions in response to the competition between the two neighbouring cells underneath and no longer touches the end walls. These modes are well related to different continent motions since the breakup of the Pangaea supercontinent.

Tuesday, November 22, 2016 8:00AM - 10:10AM –
Session M12 Multiphase Flows: Aerodynamics and Hydrodynamics C123 - Ali Tohidi,
 University of Maryland at College Park

8:00AM M12.00001 Experimental & Numerical Modeling of Non-combusting Model Firebrands' Transport¹, ALI TOHIDI, University of Maryland College Park, NIGEL KAYE, Clemson University — Fire spotting is one of the major mechanisms of wildfire spread. Three phases of this phenomenon are firebrand formation and break-off from burning vegetation, lofting and downwind transport of firebrands through the velocity field of the wildfire, and spot fire ignition upon landing. The lofting and downwind transport phase is modeled by conducting large-scale wind tunnel experiments. Non-combusting rod-like model firebrands with different aspect ratios are released within the velocity field of a jet in a boundary layer cross-flow that approximates the wildfire velocity field. Characteristics of the firebrand dispersion are quantified by capturing the full trajectory of the model firebrands using the developed image processing algorithm. The results show that the lofting height has a direct impact on the maximum travel distance of the model firebrands. Also, the experimental results are utilized for validation of a highly scalable coupled stochastic & parametric firebrand flight model that, couples the LES-resolved velocity field of a jet-in-nonuniform-cross-flow (JINCF) with a 3D fully deterministic 6-degrees-of-freedom debris transport model. The validation results show that the developed numerical model is capable of estimating average statistics of the firebrands' flight.

¹Authors would like to thank support of the National Science Foundation under Grant No. 1200560. Also, the presenter (Ali Tohid) would like to thank Dr. Michael Gollner from the University of Maryland College Park for the conference participation support

8:13AM M12.00002 Simultaneous velocity measurements of particle and gas phase in particle-laden co-flowing pipe jets¹, ISAAC SARIDAKIS, TIMOTHY LAU, Centre for Energy Technology, School of Mechanical Engineering, University of Adelaide, Australia, LYAZID DJENIDI, School of Engineering, University of Newcastle, Australia, GRAHAM NATHAN, Centre for Energy Technology, School of Mechanical Engineering, University of Adelaide, Australia — Simultaneous planar velocity measurements of both the carrier gas and particles are reported of well-characterized particle-laden co-flowing pipe jets. It is proposed to present measurements that were obtained through application of a median-filter discrimination technique to separate the Particle Image Velocimetry (PIV) signals of the $0.5\mu\text{m}$ diameter fluid tracers from those of the larger particles of diameter $20\mu\text{m}$ and $40\mu\text{m}$. Instantaneous particle and fluid planar velocity distributions were measured for three Reynold's numbers ranging from 10,000 to 40,000 and five Stokes numbers from 1 to 22, at a jet bulk fluid velocity to co-flow velocity ratio of 12. Selected results will be presented which show that the slip velocity is dependent on the local Stokes number. These are the first simultaneous carrier gas and particle velocity measurements in particle-laden jets and provide new understanding of fluid-particle interactions.

¹Financial support from Australian Research Council and Australian Renewable Energy Agency

8:26AM M12.00003 Pairwise Interaction Extended Point Particle (PIEP) Model for a Random Array of Spheres., GEORGES AKIKI, Univ of Florida - Gainesville, THOMAS JACKSON, SIVARAMAKRISHNAN BALACHANDAR, University of Florida, CENTER FOR COMPRESSIBLE MULTIPHASE TURBULENCE TEAM — This study investigates a flow past random array of spherical particles. The understanding of the governing forces within these arrays is crucial for obtaining accurate models used in particle-laden simulations. These models have to faithfully reflect the sub-grid interactions between the particles and the continuous phase. The models being used today assumes an average force on all particles within the array based on the mean volume fraction and Reynolds number. Here, we develop a model which can compute the drag and lateral forces on each particle by accounting for the precise location of few surrounding neighbors. A pairwise interaction is assumed where the perturbation flow induced by each neighbor is considered separately, then the effect of all neighbors are linearly superposed to obtain the total perturbation. Faxén correction is used to quantify the force perturbation due to the presence of the neighbors. The single neighbor perturbations are mapped in the vicinity of a reference sphere and stored as libraries. We test the Pairwise Interaction Extended Point-Particle (PIEP) model for random arrays at two different volume fractions of $\phi = 0.1$ and 0.21 and Reynolds number in the range $16 \leq Re \leq 170$. The PIEP model predictions are compared against drag and lift forces obtained from fully-resolved DNS performed using immersed boundary method. We observe the PIEP model prediction to correlate much better with the DNS results than the classical mean drag model prediction.

8:39AM M12.00004 Generalized Faxén's theorem: Evaluating first-order (hydrodynamic drag) and second-order (acoustic radiation) forces on finite-sized rigid particles, bubbles and droplets in arbitrary complex flows¹, SUBRAMANIAN ANNAMALAI, S BALACHANDAR, University of Florida Gainesville, FL — In recent times, study of complex disperse multiphase problems involving several million particles (e.g. volcanic eruptions, spray control etc.) is garnering momentum. The objective of this work is to present an accurate model (termed generalized Faxén's theorem) to predict the hydrodynamic forces on such inclusions (particles/bubbles/droplets) without having to solve for the details of flow around them. The model is developed using acoustic theory and the force obtained as a summation of infinite series (monopole, dipole and higher sources). The first-order force is the time-dependent hydrodynamic drag force arising from the dipole component due to interaction between the gas and the inclusion at the microscale level. The second-order force however is a time-averaged differential force (contributions arise both from monopole and dipole), also known as the acoustic radiation force primarily used to levitate particles. In this work, the monopole and dipole strengths are represented in terms of particle surface and volume averages of the incoming flow properties and therefore applicable to particle sizes of the order of fluid length scale and subjected to any arbitrary flow. Moreover, this model can also be used to account for inter-particle coupling due to neighboring particles.

¹U.S. DoE, NNSA, Advanced Simulation and Computing Program, Cooperative Agreement under PSAAP-II, Contract No. DE-NA0002378

8:52AM M12.00005 Fully-resolved simulation of particle rotation in a turbulent flow¹, YAYUN WANG, ADAM J. SIERAKOWSKI, Department of Mechanical Engineering, Johns Hopkins University, ANDREA PROSPERETTI², Department of Mechanical Engineering, University of Houston — Some results on the statistics of particle rotation induced by hydrodynamic stresses in a weakly turbulent flow, with a Taylor Reynolds number of about 32, are presented. Two particle Reynolds numbers, 80 and 150, and two different particle moments of inertia, are considered. The particle center is held fixed so that the particle statistics can be compared with those of the fluid vorticity in the absence of the particle. It is found that the particle essentially responds only to vortex structures with a scale comparable to, or larger than, its diameter, thus acting as a low-pass filter for the incident turbulent vorticity. An analysis of the flatness of the PDF's of the angular acceleration and angular velocity shows that the former is mildly non-Gaussian, while the latter is very close to Gaussian. The numerical method used, Physalis, is particularly suited for this problem due to the high accuracy with which the couple is calculated.

¹Supported by NSF grant CBET 1335965

²Also: Department of Science and Technology, University of Twente, Enschede, The Netherlands

9:05AM M12.00006 Angular velocity of a spheroid log rolling in a simple shear at small Reynolds number, JAN MEIBOHM, Department of Physics, Gothenburg University, FABIEN CANDELIER, University of Aix-Marseille, TOMAS ROSEN, KTH Mechanics, Royal Institute of Technology, JONAS EINARSSON, Department of Physics, Gothenburg University, FREDRIK LUNDELL, KTH Mechanics, Royal Institute of Technology, BERNHARD MEHLIG, Department of Physics, Gothenburg University — We analyse the angular velocity of a small neutrally buoyant spheroid log rolling in a simple shear. When the effect of fluid inertia is negligible the angular velocity $\bar{\omega}$ equals half the fluid vorticity. We compute by singular perturbation theory how weak fluid inertia reduces the angular velocity in an unbounded shear, and how this reduction depends upon the shape of the spheroid (on its aspect ratio). In addition we determine the angular velocity by direct numerical simulations. The results are in excellent agreement with the theory at small but not too small values of the shear Reynolds number, for all aspect ratios considered. For the special case of a sphere we find $\omega/s = -1/2 + 0.0540 Re^{3/2}$ where s is the shear rate and Re is the shear Reynolds number. This result differs from that derived by Lin et al. [J. Fluid Mech. 44 (1970) 1] who obtained a numerical coefficient roughly three times larger.

9:18AM M12.00007 Influence of lubrication forces in direct numerical simulations of particle-laden flows¹, ROHIT MAITRI, FRANK PETERS, JOHAN PADDING, HANS KUIPERS, Eindhoven Univ of Tech — Accurate numerical representation of particle-laden flows is important for fundamental understanding and optimizing the complex processes such as proppant transport in fracking. Liquid-solid flows are fundamentally different from gas-solid flows because of lower density ratios (solid to fluid) and non-negligible lubrication forces. In this interface resolved model, fluid-solid coupling is achieved by incorporating the no-slip boundary condition implicitly at particles surfaces by means of an efficient second order ghost-cell immersed boundary method. A fixed Eulerian grid is used for solving the Navier-Stokes equations and the particle-particle interactions are implemented using the soft sphere collision and sub-grid scale lubrication model. Due to the range of influence of lubrication force on a smaller scale than the grid size, it is important to implement the lubrication model accurately. In this work, different implementations of the lubrication model on particle dynamics are studied for various flow conditions. The effect of a particle surface roughness on lubrication force and the particle transport is also investigated. This study is aimed at developing a validated methodology to incorporate lubrication models in direct numerical simulation of particle laden flows.

¹This research is supported from grant 13CSER014 of the Foundation for Fundamental Research on Matter (FOM), which is part of the Netherlands Organisation for Scientific Research (NWO).

9:31AM M12.00008 Numerical investigation of drag characteristics of spherical particles under non-isothermal conditions, JUNGWOO KIM, YEONG EUN YIM , Seoul National University of Science and Technology — In predicting particle-laden flows related to particle transport and dispersion, better understanding and accurate parameterization of the hydrodynamic forces on the particles are one of the important subjects. Heat transfer between dispersed particle and fluid is often observed in nature and engineering applications. However, existing analytical expressions and empirical correlations used in point particle approaches are made based on the assumption that the particle and surrounding ambient flow are under thermal equilibrium conditions. So, the effect of thermal non-equilibrium state of particle motion remains an unresolved issue. Therefore, we perform three-dimensional numerical simulations for the flow around a finite-sized spherical particle in order to investigate its drag characteristics under non-isothermal conditions (heated or cooled particles). In this study, the working fluids are considered to be water and air as typical cases of liquids and gases. The heated particle experiences larger drag in air and smaller drag in water than that in the isothermal case. On the other hand, the impact of cooling is to decrease drag in air and to increase it in water. These behaviors of the drag coefficient in air and water mainly depend on the variation of the viscosity in terms of the temperature. Those results would provide useful information in understanding the particle motion in heated or cooled conditions.

9:44AM M12.00009 Drag of Spherical Particles in a Periodic Lattice: Heat Transfer, Buoyancy and Non-Boussinesq Effects¹, SWETAVA GANGULI, SANJIVA LELE, Stanford University — What are the forces that act on a particle as it moves in a fluid? How do they change in the presence of significant heat transfer from the particle, a variable density fluid or gravity? Last year, we demonstrated and quantified these effects on a single particle. We further our study by adding interactions between particles placed in a periodic lattice whose parameters we control. Our particle resolved simulations use a fully unstructured, node-based, low-Mach variable density solver to study the low-Mach response. Let the Boussinesq parameter λ as the ratio of the difference of the particle temperature and the far-field fluid temperature to the far-field fluid temperature. The heating of the fluid near the particle affects the drag significantly which can be characterized in a parameter space where the variation in Reynolds number, λ and Froude number can be collapsed to a single parameter. Despite the large drag changes, the pressure and viscous fractional contributions do not vary with λ . In the low Re limit, a semi-analytical low Mach perturbation expansion has significant explanatory power. For a single particle, these variations can be captured with 95% accuracy by developing correlations based on physical insights from the semi-analytical model. When particles are placed within a lattice, depending on the lattice parameter, the individual wakes of the particles interact and the drag increases or decreases based on the lattice position.

¹PSAAP II at Stanford University

9:57AM M12.00010 Heat transfer in suspensions of rigid particles¹, LUCA BRANDT, MEHDI NIAZI ARDEKANI, Linne FLOW Centre, KTH, Stockholm, OMID ABOUALI, Shiraz University, Mechanical Engineering Department — We study the heat transfer in laminar Couette flow of suspensions of rigid neutrally buoyant particles by means of numerical simulations. An Immersed Boundary Method is coupled with a VOF approach to simulate the heat transfer in the fluid and solid phase, enabling us to fully resolve the heat diffusion. First, we consider spherical particles and show that the proposed algorithm is able to reproduce the correlations between heat flux across the channel, the particle volume fraction and the heat diffusivity obtained in laboratory experiments and recently proposed in the literature, results valid in the limit of vanishing inertia. We then investigate the role of inertia on the heat transfer and show an increase of the suspension diffusivity at finite particle Reynolds numbers. Finally, we vary the relativity diffusivity of the fluid and solid phase and investigate its effect on the effective heat flux across the channel. The data are analyzed by considering the ensemble averaged energy equation and decomposing the heat flux in 4 different contributions, related to diffusion in the solid and fluid phase, and the correlations between wall-normal velocity and temperature fluctuations. Results for non-spherical particles will be examined before the meeting.

¹Supported by the European Research Council Grant No. ERC-2013- CoG-616186, TRITOS. The authors acknowledge computer time provided by SNIC (Swedish National Infrastructure for Computing)

Tuesday, November 22, 2016 8:00AM - 10:10AM –
Session M13 DFD GPC: Geophysical Fluid Dynamics: Atmospheric Dynamics C124 -
Harindra Fernando, University of Notre Dame

8:00AM M13.00001 Turbulence in the Stable Atmospheric Boundary Layer¹ , HARINDRA

FERNANDO, University of Notre Dame, ELIEZER KIT, Tel Aviv University, PATRICK CONRY, University of Notre Dame, CHRISTOPHER HOCUT, US Army Research Laboratory, DAN LIBERZON, Technion — During the field campaigns of the Mountain Terrain Atmospheric Modeling and Observations (MATERHÖRN) Program, fine-scale measurements of turbulence in the atmospheric boundary layer (ABL) were made using a novel sonic and hot-film anemometer dyad (a combo probe). A swath of scales, from large down to Kolmogorov scales, was covered. The hot-film was located on a gimbal within the sonic probe volume, and was automated to rotate in the horizontal plane to align with the mean flow measured by sonic. This procedure not only helped satisfy the requirement of hot-film alignment with the mean flow, but also allowed in-situ calibration of hot-films. This paper analyzes a period of nocturnal flow that was similar to an idealized stratified parallel shear flow. Some new phenomena were identified, which included the occurrence of strong bursts in the velocity records indicative of turbulence generation at finer scales that are not captured by conventional sonic anemometers. The spectra showed bottleneck effect, but its manifestation did not fit into the framework of previous bottleneck-effect theories and was unequivocally related to bursts of turbulence. The measurements were also used to evaluate the energetics of stratified shear flows typical of the environment.

¹ONR N00014-11-1-0709; NSF AGS-1528451; ISF 408/15

8:13AM M13.00002 Katabatic flow: a closed-form solution with spatially-varying eddy diffusivities , MARCO G. GIOMETTO, École Polytechnique Fédérale de Lausanne, Lausanne, Switzerland; University of British

Columbia, Vancouver, BC, Canada, RICCARDO GRANDI, JIANNONG FANG, PETER A. MONKEWITZ, École Polytechnique Fédérale de Lausanne, Lausanne, Switzerland, MARC B. PARLANGE, University of British Columbia, Vancouver, BC, Canada — The Nieuwstadt closed-form solution for the stationary Ekman layer is generalized for katabatic flows within the conceptual framework of the Prandtl model. The proposed solution is valid for spatially-varying eddy diffusivities (O'Brien type) and constant Prandtl number (Pr). Variations in the velocity and buoyancy profiles will be discussed as a function of the dimensionless model parameters $z_0 \equiv \hat{z}_0 \hat{N}^2 Pr \sin(\alpha) |\hat{b}_s|^{-1}$ and $\lambda \equiv \hat{u}_{\text{ref}} \hat{N} \sqrt{Pr} |\hat{b}_s|^{-1}$, where \hat{z}_0 is the hydrodynamic roughness length, \hat{N} is the buoyancy frequency, α is the surface sloping angle, \hat{b}_s is the imposed surface buoyancy, and \hat{u}_{ref} is a reference velocity scale used to define eddy diffusivities. Profiles show significant variations in both phase and amplitude of extrema with respect to the classic constant K model and with respect to a recent approximate analytic solution based on the Wentzel-Kramers-Brillouin theory, hence shedding new light on the problem.

8:26AM M13.00003 Deviations from Equilibrium in Daytime Atmospheric Boundary Layer Turbulence arising from Nonstationary Mesoscale Forcing¹ , BALAJI JAYARAMAN, Oklahoma

St. U. , JAMES BRASSEUR, U. Colorado, SUE HAUPt, JARED LEE, NCAR — LES of the “canonical” daytime atmospheric boundary layer (ABL) over flat topography is developed as an equilibrium ABL with steady surface heat flux, Q_0 and steady unidirectional “geostrophic” wind vector V_g above a capping inversion. A strong inversion layer in daytime ABL acts as a “lid” that sharply separates 3D “microscale” ABL turbulence at the O(10) m scale from the quasi-2D “mesoscale” turbulent weather eddies (O(100) km scale). While “canonical” ABL is equilibrium, quasi-stationary and characterized statistically by the ratio of boundary layer depth (z_i) to Obukhov length scale ($-L$), the real mesoscale influences (U_g and Q_0) that force a true daytime ABL are nonstationary at both diurnal and sub-diurnal time scales. We study the consequences of this non-stationarity on ABL dynamics by forcing ABL LES with realistic WRF simulations over flat Kansas terrain. Considering horizontal homogeneity, we relate the mesoscale and geostrophic winds, U_g and V_g , and systematically study the ABL turbulence response to non-steady variations in Q_0 and U_g . We observe significant deviations from equilibrium, that manifest in many ways, such as the formation of “roll” eddies purely from changes in mesoscale wind direction that are normally associated with increased surface heat flux.

¹Support from DOE. Compute resources from Penn State ICS

8:39AM M13.00004 Turbulence dynamics in unsteady atmospheric flows¹ , MOSTAFA

MOMEN, ELIE BOU-ZEID, Princeton University — Unsteady pressure-gradient forcing in geophysical flows challenges the quasi-steady state assumption, and can strongly impact the mean wind and higher-order turbulence statistics. Under such conditions, it is essential to understand when turbulence is in quasi-equilibrium, and what are the implications of unsteadiness on flow characteristics. The present study focuses on the unsteady atmospheric boundary layer (ABL) where pressure gradient, Coriolis, buoyancy, and friction forces interact. We perform a suite of LES with variable pressure-gradient. The results indicate that the dynamics are mainly controlled by the relative magnitudes of three time scales: T_{inertial} , $T_{\text{turbulence}}$, and T_{forcing} . It is shown that when $T_f \approx T_t$, the turbulence is no longer in a quasi-equilibrium state due to highly complex mean-turbulence interactions; consequently, the log-law and turbulence closures are no longer valid in these conditions. However, for longer and, surprisingly, for shorter forcing times, quasi-equilibrium is maintained. Varying the pressure gradient in the presence of surface buoyancy fluxes primarily influences the buoyant destruction in the stable ABLs, while under unstable conditions it mainly influences the transport terms.

¹NSF-PDM under AGS-10266362. Cooperative Institute for Climate Science, NOAA-Princeton University under NA08OAR4320752. Simulations performed at NCAR, and Della server at Princeton University.

8:52AM M13.00005 Influence of cross-flow on the entrainment of bending plumes

, GRAHAM FREEDLAND, Portland State University, LARRY MASTIN, United States Geological Survey, SOLOVITZ STEVEN, Washington State University, RAUL CAL, Portland State University — Volcanic eruption columns inject high concentrations of ash into the atmosphere. Some of this ash is carried downwind forming ash clouds in the atmosphere that are hazardous for private and commercial aviation. Current models rely on inputs such as plume height, duration, eruption rate, and meteorological wind fields. Eruption rate is estimated from plume height using relations that depend on the rate of air entrainment into the plume, which is not well quantified. A wind tunnel experiment has been designed to investigate these models by injecting a vertical air jet into a cross-flow. The ratio of the cross-flow and jet velocities is varied to simulate a weak plume, and flow response is measured using particle image velocimetry. The plumes are characterized and profile data is examined to measure the growth of weak plumes and the entrainment velocity along its trajectory. This allows for the study of the flow field, mean, and second order moments, and obtain information to improve models of volcanic ash concentrations in the atmosphere.

9:05AM M13.00006 Dynamic roughness model for LES of turbulent flow over multiscale urban-like topography¹, XIAOWEI ZHU, WILLIAM ANDERSON, UT Dallas — Urban-like topographies are composed of a wide spectrum of topographic elements, which results in multiscale, fractal-like distributions. This has important implications for microscale numerical weather prediction in urban environments, or urban meteorology: the range of scales inhibits the use of numerical schemes where the topography is fully resolved, but the self-similar nature of the topography inspires development of closures that leverage such self-similarity to parameterize unresolved information. That is, a natural urban landscape can be low-pass filtered at the large-eddy simulation grid scale, thereby removing details between the grid scale and the smallest scale of the landscape, but the effects of these truncated topographic modes can be modeled based on details of the large scale. LES has been used to investigate the effects of subgrid-scale (SGS) topography on the roughness length of multiscale urban-like topographies. First, high-resolution multiscale urban-like topographies were generated with random distribution function. Then, the high-resolution multiscale topography was filtered and separated into large- and small-scale topographies with the Reynolds decomposition. Thus, the topography was decomposed into resolved (scale larger than the grid scale) and SGS part (scale smaller than the filter scale). The resolved part was resolved in LES, while the SGS terrain must be parameterized. New models for urban roughness will be used to parameterize SGS topography.

¹Army Research Office, Grant W911NF-15-1-0231

9:18AM M13.00007 Modulation of energetic coherent motions by large-scale topography., WING LAI, TSI Incorporated, ALI M. HAMED, University of Illinois at Urbana-Champaign, DAN TROOLIN, TSI Incorporated, LEONARDO P. CHAMORRO, University of Illinois at Urbana-Champaign — The distinctive characteristics and dynamics of the large-scale coherent motions induced over 2D and 3D large-scale wavy walls were explored experimentally with time-resolved volumetric PIV, and selected wall-normal high-resolution stereo PIV in a refractive-index-matching channel. The 2D wall consists of a sinusoidal wave in the streamwise direction with amplitude to wavelength ratio $a/\lambda x = 0.05$, while the 3D wall has an additional wave in the spanwise direction with $a/\lambda y = 0.1$. The flow was characterized at $Re \approx 8000$, based on the bulk velocity and the channel half height. The walls are such that the amplitude to boundary layer thickness ratio is $a/\delta_99 \approx 0.1$, which resemble geophysical-like topography. Insight on the dynamics of the coherent motions, Reynolds stress and spatial interaction of sweep and ejection events will be discussed in terms of the wall topography modulation.

9:31AM M13.00008 A nonlinear self-similar solution to barotropic flow over rapidly varying topography, RUY IBANEZ, JOSEPH KUEHL, Baylor University — Beginning from the Shallow Water Equations (SWE), a nonlinear self-similar analytic solution is derived for barotropic flow over rapidly varying topography. We study conditions relevant to the ocean slope where the flow is dominated by Earth's rotation and topography. Attention is paid to the northern Gulf of Mexico slope with application to pollutant dispersion and the Norwegian Coastal Current which sheds eddies into the Lofoten Basin that are believed to influence deep water formation. The solution is found to extend the topographic β -plume solution (Kuehl 2014, GRL) in two ways: 1) The solution is valid for intensifying jets. 2) The influence of nonlinear advection is included. The SWE are scaled to the case of a topographically controlled jet, then solved by introducing a similarity variable $\eta = Cxy$. The nonlinear solution, valid for topographies $h = h_0 - \alpha xy^3$, takes the form of the Lambert W Function for velocity. The linear solution, valid for topographies $h = h_0 - \alpha xy^\gamma$, takes the form of the Error Function for transport. Kuehl's results considered the case $-1 \leq \gamma < 1$ which admits expanding jets, while the new result consider the case $\gamma < -1$ which admits intensifying jets.

9:44AM M13.00009 Interactions between intermittent gravity waves and infrasounds, BRUNO RIBSTEIN, CMLA, ENS, F-94230, Cachan, France and LMD, ENS, F-75005, Paris, France, CHRISTOPHE MILLET, CMLA, ENS, F-94230, Cachan, France and CEA, DAM, DIF, F91297, Arpajon, France, FRANCOIS LOTT, LMD, ENS, F-75005, Paris, France, ALVARO DE LA CAMARA, NCAR, Boulder, USA — Even though the accuracy of atmospheric specifications is constantly improving, it is well known that the main part of gravity waves is still yet not resolved in the available data. In most infrasound modeling studies, the unresolved gravity wave field is often represented as a deterministic field that is superimposed on a given average background state. Direct observations in the lower stratosphere show, however, that the gravity wave field is very intermittent, and is often dominated by rather well defined wave packets. In this study we sample the gravity wave spectrum by launching few monochromatic waves and choose their properties stochastically to mimic the intermittency. The statistics of acoustic signals are computed by decomposing the original signal into a sum of modal pulses. Owing to the disparity of the gravity and acoustic lengthscales, the interaction can be described using a multiscale analysis and the appropriate amplitude evolution equation involves certain random terms that are related to the gravity wave sources. More specifically, it is shown how the unpredictable low level small-scale dynamics triggers multiple random stratospheric waveguides in which high frequency infrasound components can propagate efficiently.

9:57AM M13.00010 Modeling Radiation Fog¹, SREENIVAS K R², RAFIUDDIN MOHAMMAD³, JNCASR, Bangalore — Predicting the fog-onset, its growth and dissipation helps in managing airports and other modes of transport. After sunset, occurrence of fog requires moist air, low wind and clear-sky conditions. Under these circumstances radiative heat transfer plays a vital role in the NBL. Locally, initiation of fog happens when the air temperature falls below the dew-point. Thus, to predict the onset of fog at a given location, one has to compute evolution of vertical temperature profile. Earlier, our group has shown that the presence of aerosols and vertical variation in their number density determines the radiative-cooling and hence development of vertical temperature profile¹. Aerosols, through radiation in the window-band, provides an efficient path for air layers to lose heat to the cold, upper atmosphere. This process creates cooler air layer between warmer ground and upper air layers and resulting temperature profile facilitate the initiation of fog. Our results clearly indicates that accounting for the presence of aerosols and their radiative-transfer is important in modeling micro-meteorological process of fog formation and its evolution. [1] Field and laboratory experiments on aerosol-induced cooling in the nocturnal boundary layer. Q.J.R.Meteorol.Soc. 140.678 2014, 151-169.

¹DST, Govt. INDIA

²Professor, Engineering Mechanics Unit

³Research Scholar

Tuesday, November 22, 2016 8:00AM - 9:57AM –
Session M14 Free Surface Flows: General C125-126 - Alfredo Soldati, University of Udine

8:00AM M14.00001 Experimental studies of the streaming flow due to the adsorption of particles at a liquid surface¹, PUSHPENDRA SINGH, NAGA MUSUNURI, IAN FISCHER, New Jersey Institute of Technology — The particle image velocimetry (PIV) technique is used to study the streaming flow that is induced when particles are adsorbed at a liquid surface. The flow develops within a fraction of second after the adsorption of the particle and persists for several seconds. The fluid directly below the particle rises upward, and near the surface, it moves away from the particle. The flow causes powders sprinkled on a liquid surface to disperse on the surface. 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

8:13AM M14.00002 Steady streaming in standing waves, JEAN RAJCHENBACH, ENRICA SAGGESE, Laboratoire de Physique de la Matière Condensée, CNRS UMR 7336, Université de Nice - Sophia Antipolis, DIDIER CLAMOND, Laboratoire J. A. Dieudonné, CNRS UMR 7351, Université de Nice - Sophia Antipolis — We report the existence of recirculating eddies existing in the bulk of a liquid under the action of standing surface waves. This phenomenon results from the combined action of the nonlinearity and viscosity. The period of these secondary flows can be, say, one hundred times that of the wave, depending on the amplitude. Our experimental results reveal strong disagreements with theoretical predictions devised hitherto. In order to account for our data, we propose a new mechanism playing a major role in the formation of these rolls.

8:26AM M14.00003 Experimental investigation of the interaction between turbulent boundary layers and near-surface wave-induced forcing., OWEN WILLIAMS, University of Washington — Free-surface waves can have a significant impact on sub-surface turbulent boundary layers that are present on undersea vehicles or on the bottom of flowing bodies of water such as estuaries. This problem has a wide parameter space and resultant changes to boundary layer structure due to wave forcing still require investigation. Here, preliminary experimental measurements within the newly commissioned wave channel at the University of Washington are detailed. Particle image velocimetry (PIV) is used to examine velocity statistics across the water column. In an effort to more readily identify changes in underlying boundary layer structure, a range of flow decompositions, such as snapshot partial orthogonal decomposition (POD) are evaluated in an effort to separate turbulent motions from the forcing, which to first order is a traveling wave. The effect of the relative difference between water depth and boundary layer thickness will be examined, as well as the Froude number of the surface waves. Ongoing efforts to examine the full parameter space will be discussed, as dimensional analysis and linear wave theory suggest there are up to seven parameters relevant to either inner or outer layers.

8:39AM M14.00004 Numerical Study of Interactions between Surface Waves and Turbulence Underneath Using Phase-Resolved Simulations, ANQING XUAN, LIAN SHEN, Univ of Minnesota - Twin Cities — It has been known that waves can substantially modify the turbulent flow underneath, for example, leading to Langmuir turbulence. To elucidate the effects of surface waves on turbulence, we perform direct numerical and large-eddy simulations using a dynamically-evolving wave-surface-fitted grid with fully nonlinear kinematic and dynamic free-surface boundary conditions. Our simulations have the capability of explicitly resolving the distortion of turbulence by instantaneous phase-resolved wave motions, in addition to the averaged Stokes drift effects. The numerical results show that the effects of waves on turbulence have been successfully captured in our simulations. In the cases of wind-driven shear turbulence interacting with waves, the transition from shear turbulence to Langmuir turbulence occurs in our simulations as the turbulent Langmuir number $\text{Lat} = (u_*/u_s)^{1/2}$, the ratio of the friction velocity to the surface Stokes drift velocity, is decreased. Counter-rotating vortices and enhancement of the mixing due to Langmuir circulations have been observed. Our simulations also reveal detailed information on the underlying mechanisms of the interactions between turbulence and surface waves.

8:52AM M14.00005 Vertical vorticity at a free surface¹, PAUL W. FONTANA, Seattle University — The concept of surface vorticity is developed as a necessary consequence of the discontinuity of flow at the fluid surface. The construct provides the proper boundary conditions for a vortex-dynamical description of surface waves. It is shown that the perturbed free surface in general possesses vertical vorticity, even when the underlying flow is irrotational and the fluid is ideal. This resolves a paradox pointed out by Umeki, who discovered irrotational surface waves with surface rotation in the horizontal plane [Phys. Fluids A 4, 1968 (1992)]. A dynamical equation for vertical vorticity at the free surface is derived and interpreted physically. The traditional idea that vortex lines terminate at fluid boundaries is shown to be unphysical and is amended to include surface vorticity. The extension of vertical surface vorticity into the bulk is connected with particular topological structures, such as plughole vortices, breaking waves, and Klein's Kaffeelöffel. This analysis generalizes boundary-layer vorticity theory to the free surface in the ideal limit. The analogy between surface vorticity on an ideal liquid and sheet currents at the surface of a superconductor is described.

¹Work done as a Visiting Fellow at the Australian National University.

9:05AM M14.00006 ABSTRACT WITHDRAWN —

9:18AM M14.00007 Very-large-scale coherent motions in open channel flows¹, QIANG ZHONG, Tsinghua University, FAZLE HUSSAIN, Texas Tech University, DAN-XUN LI, Tsinghua University — Very-large-scale coherent structures (VLSSs) - whose characteristic length is of the order of $10h$ (h is the water depth) - are found to exist in the log and outer layers near the bed of open channel flows. For decades researchers have speculated that large coherent structures may exist in open channel flows. However, conclusive evidence is still lacking. The present study employed pre-multiplied velocity power spectral and co-spectral analyses of time-resolved PIV data obtained in open channel flows. In all cases, two modes - large-scale structures (of the order of h) and VLSSs - dominate the log and outer layers of the turbulent boundary layer. More than half of TKE and 40% of the Reynolds shear stress in the log and outer layers are contributed by VLSSs. The strength difference of VLSSs between open and closed channel flows leads to pronounced redistribution of TKE near the free surface of open channel flows, which is a unique phenomenon that sets the open channel flows apart from other wall-bounded turbulent flows.

¹Funded by China Postdoctoral Science Foundation (No.2015M580105), National Natural Science Foundation of China (No.51127006).

9:31AM M14.00008 Dynamics of motile micro-organisms in stably-stratified turbulence, SALVATORE LOVECCHIO, IMFT Toulouse, FRANCESCO ZONTA, TU Wien, CRISTIAN MARCHIOLI, University of Udine, ALFREDO SOLDATI, University of udine; TU Wien — Motile micro-organisms populating terrestrial water bodies swim upward towards the air-water interface to capture light and activate photosynthesis. These micro-organisms have the center of mass displaced below the center of buoyancy and are usually called gyrotactic swimmers. Gyrotactic swimmers (which are almost neutrally-buoyant) are extremely sensitive to the local flow field, which is often stably stratified (due to solar heating at the water surface). Stable stratification has a deep influence on the transport processes of mass, momentum, heat and chemical species at the water surface. In this work we use Direct Numerical Simulation (DNS) and Lagrangian Particle Tracking (LPT) to analyze the dynamics of gyrotactic swimmers in stratified turbulence. Our results show that swimmers surfacing and clustering at the surface depend strongly on the re-orientation time of swimmers and on the level of stratification. Obtaining accurate predictions of the surfacing time for gyrotactic swimmers is extremely important to estimate the global CO_2 exchange across the air-water interface.

9:44AM M14.00009 Micro-swimmer dynamics in free-surface turbulence subject to wind stress, CRISTIAN MARCHIOLI, Univ of Udine, SALVATORE LOVECCHIO, IMFT Toulouse, ALFREDO SOLDATI, University of Udine & TUWien — 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 re-orient swimmers, reducing their vertical migration and causing entrapment 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 reo-rientation 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.

Tuesday, November 22, 2016 8:00AM - 10:10AM –
Session M15 Bio: Red Blood Cells and Thrombosis Formation E143/144 - Shawn Shadden, University of California, Berkeley

8:00AM M15.00001 An immersed-boundary method for modeling flow of deformable blood cells in complex geometry¹, PETER BALOGH, PROSENJIT BAGCHI, Rutgers University — We present a computational methodology for simulating blood flow at the cellular scale in highly complex geometries, such as microvascular networks. Immersed boundary methods provide the foundation for our approach, as they allow modeling flows in arbitrary geometries, in addition to resolving the large deformation and dynamics of individual blood cell with high fidelity. Different simulation components are seamlessly integrated into the present methodology that can simultaneously model stationary rigid boundaries of arbitrary and complex shape, moving rigid bodies, and highly deformable interfaces of blood cells that are governed by non-linear elasticity. This permits physiologically realistic simulations of blood cells flowing in complex microvascular networks characterized by multiple bifurcating and merging vessels. The methodology is validated against analytical theory, experimental data, and previous numerical results. We then demonstrate the capabilities of the methodology by simulating deformable blood cells and heterogeneous cell suspensions flowing in both physiologically realistic microvascular networks and geometrically intricate microfluidic devices. The methodology offers the potential of scaling up to large microvascular networks at organ levels.

¹Funded by NSF CBET 1604308

8:13AM M15.00002 Axial dispersion in flowing red blood cell suspensions¹, THOMAS PODGORSKI, SYLVAIN LOSSERAND, GWENNOU COUPIER, LIPhy, CNRS - Université Grenoble Alpes — A key parameter in blood microcirculation is the transit time of red blood cells (RBCs) through an organ, which can influence the efficiency of gas exchange and oxygen availability. A large dispersion of this transit time is observed *in vivo* and is partly due to the axial dispersion in the flowing suspension. In the classic Taylor-Aris example of a solute flowing in a tube, the combination of molecular diffusion and parabolic velocity profile leads to enhanced axial dispersion. In suspensions of non-Brownian deformable bodies such as RBCs, axial dispersion is governed by a combination of shear induced migration and shear-induced diffusion arising from hydrodynamic interactions. We revisit this problem in the case of RBC pulses flowing in a microchannel and show that the axial dispersion of the pulse eventually saturates with a final extension that depends directly on RBC mechanical properties. The result is especially interesting in the dilute limit since the final pulse length depends only on the channel width, exponent of the migration law and dimensionless migration velocity. In continuous flow, the dispersion of transit times is the result of complex cell-cell and cell-wall interactions and is strongly influenced by the polydispersity of the blood sample.

¹The authors acknowledge support from LabEx TEC21 and CNES.

8:26AM M15.00003 Modeling malaria infected cells in microcirculation, AMIR HOSSEIN RAFFIEE, SADEGH DABIRI, AREZOO MOTAVALIZADEH ARDEKANI, Purdue Univ — Plasmodium (*P.*) falciparum is one of the deadliest types of malaria species that invades healthy red blood cells (RBC) in human blood flow. This parasite develops through 48-hour intra-RBC process leading to significant morphological and mechanical (e.g., stiffening) changes in RBC membrane. These changes have remarkable effects on blood circulation such as increase in flow resistance and obstruction in microcirculation. In this work a computational framework is developed to model RBC suspension in blood flow using front-tracking technique. The present study focuses on blood flow behavior under normal and infected circumstances and predicts changes in blood rheology for different levels of parasitemia and hematocrit. This model allows better understanding of blood flow circulation up to a single cell level and provides us with realistic and deep insight into hematologic diseases such as malaria.

8:39AM M15.00004 Blood flow simulation on a role for red blood cells in platelet adhesion.

KAZUYA SHIMIZU, Univ of Tokyo, KAZUYASU SUGIYAMA, Osaka Univ, SHU TAKAGI, Univ of Tokyo — Large-scale blood flow simulations were conducted and a role for red blood cells in platelet adhesion was discussed. The flow conditions and hematocrit values were set to the same as corresponding experiments, and the numerical results were compared with the measurements. Numerical results show the number of platelets adhered on the wall is increased with the increase in hematocrit values. The number of adhered platelets estimated from the simulation was approximately 28 (per 0.01 square millimeter per minute) for the hematocrit value of 20 %. These results agree well with the experimental results qualitatively and quantitatively, which proves the validity of the present numerical model including the interaction between fluid and many elastic bodies and the modeling of platelet adhesion. Numerical simulation also reproduces the behavior of red blood cells in the blood flow and their role in platelet adhesion. Red blood cells deform to a flat shape and move towards channel center region. In contrast, platelets are pushed out and have many chances to contact with the wall. As a result, the large number of adhered platelets is observed as hematocrit values becomes high. This result indicates the presence of red blood cells plays a crucial role in platelet adhesion.

8:52AM M15.00005 Seamless particle-based modeling of blood clotting¹

ALIREZA YAZDANI, GEORGE KARNIADAKIS, Brown University — We propose a new multiscale framework that seamlessly integrate four key components of blood clotting namely, blood rheology, cell mechanics, coagulation kinetics and transport of species and platelet adhesive dynamics. We use transport dissipative particle dynamics (tDPD) which is an extended form of original DPD as the base solver to model both blood flow and the reactive transport of chemical species in the coagulation cascade. Further, we use a coarse-grained representation of blood cell's membrane that accounts for its mechanics; both red blood cells and platelets are resolved at sub-cellular resolution, and stochastic bond formation/dissociation are included to account for platelet adhesive dynamics at the site of injury. Our results show good qualitative agreement with *in vivo* experiments. The numerical framework allows us to perform systematic analysis on different mechanisms of blood clotting. In addition, this new multiscale particle-based methodology can open new directions in addressing different biological processes from sub-cellular to macroscopic scales.

¹NIH Grant No. U01HL116323

9:05AM M15.00006 Computational reconstruction and fluid dynamics of *in vivo* thrombi from the microcirculation

MEHRAN MIRRAZANI, Mechanical Engineering, University of California, Berkeley, MAURIZIO TOMAIUOLO, TIMOTHY STALKER, Department of Medicine, University of Pennsylvania, SHAWN SHADDEN, Mechanical Engineering, University of California, Berkeley — Blood flow and mass transfer can have significant effects on clot growth, composition and stability during the hemostatic response. We integrate *in vivo* data with CFD to better understand transport processes during clot formation. By utilizing electron microscopy, we reconstructed the 3D thrombus structure formed after a penetrating laser injury in a mouse cremaster muscle. Random jammed packing is used to reconstruct the microenvironment of the platelet aggregate, with platelets modeled as ellipsoids. In our 3D model, Stokes flow is simulated to obtain the velocity field in the explicitly meshed gaps between platelets and the lumen surrounding the thrombus. Based on *in vivo* data, a clot is composed of a core of highly activated platelets covered by a shell of loosely adherent platelets. We studied the effects of clot size (thrombus growth), gap distribution (consolidation), and vessel blood flow rate on mean intrathrombus velocity. The results show that velocity is smaller in the core as compared to the shell, potentially enabling higher concentration of agonists in the core contributing to its activation. In addition, our results do not appear to be sensitive to the geometry of the platelets, but rather gap size plays more important role on intrathrombus velocity and transport.

9:18AM M15.00007 A Fictitious Domain Method for Resolving the Interaction of Blood Flow with Clot Growth¹

DEBANJAN MUKHERJEE, SHAWN SHADDEN, University of California - Berkeley — Thrombosis and thrombo-embolism cause a range of diseases including heart attack and stroke. Closer understanding of clot and blood flow mechanics provides valuable insights on the etiology, diagnosis, and treatment of thrombotic diseases. Such mechanics are complicated, however, by the discrete and multi-scale phenomena underlying thrombosis, and the complex interactions of unsteady, pulsatile hemodynamics with a clot of arbitrary shape and microstructure. We have developed a computational technique, based on a fictitious domain based finite element method, to study these interactions. The method can resolve arbitrary clot geometries, and dynamically couple fluid flow with static or growing clot boundaries. Macroscopic thrombus-hemodynamics interactions were investigated within idealized vessel geometries representative of the common carotid artery, with realistic unsteady flow profiles as inputs. The method was also employed successfully to resolve micro-scale interactions using a model driven by *in-vivo* morphology data. The results provide insights into the flow structures and hemodynamic loading around an arbitrarily grown clot at arterial length-scales, as well as flow and transport within the interstices of platelet aggregates composing the clot.

¹The work was supported by AHA Award No: 16POST27500023

9:31AM M15.00008 Spatial dependence of thrombolysis

HARI HARA SUDHAN LAKSHMANAN, Department of Chemical and Biomedical Engineering, West Virginia University, JEVGENIA ZILBERMAN-RUDENKO, OWEN MCCARTY, Department of Biomedical Engineering, Oregon Health & Science University, JEEVAN MADDALA, Department of Chemical and Biomedical Engineering, West Virginia University — Thrombolysis under hemodynamic conditions is affected by both transport processes and reactions, thus profoundly dependent on the geometry of blood vessels or vasculature. Although thrombosis has long been observed clinically, a systematic and quantitative understanding has not been established in complex geometries such as vasculature, where various factors would affect thrombogenesis and its stability. A thrombus's location determines the effect of hydrodynamic forces on it and rate of tPA diffusion, that would result in either embolization or formation of micro-aggregates. Preliminary experiments have shown that thrombolysis is not uniform across an entire network with different locations lysing at different rates. Numerical simulations of thrombolysis under hemodynamics in a microfluidic geometry such as a ladder network with a focus on parameters such as reaction rate, shear gradient, velocity and diffusion established the lysis's dependence on geometry. Finite element simulations of blood flow coupled with reactions have been performed in COMSOL and the results were used to develop quantifiable metrics for thrombolysis in a complex geometry.

9:44AM M15.00009 Evaluating How Circle of Willis Topology Affects Embolus Distribution in the Brain

NEEL JANI, DEBANJAN MUKHERJEE, SHAWN SHADDEN, Univ of California - Berkeley — Embolic stroke occurs when fragmented cellular or acellular material (emboli) travels to the brain to occlude an artery. Understanding the transport of emboli across unsteady, pulsatile flow in complex arterial geometries is challenging and influenced by a range of factors, including patient anatomy. The work herein develops the modeling and mechanistic understanding of how embolus transport is affected by the arterial connections at the base of the brain known as the Circle of Willis (CoW). A majority of the human population has an incomplete CoW anatomy, with connections either missing or ill-developed. We employ numerical simulations combining image-based modeling, computational fluid dynamics, discrete particle dynamics, and a sampling based analysis to compare collateral flow through the most prevalent CoW topologies, to determine embolus distribution fractions among vessels in the CoW, and to investigate the role of inertial effects in causing differences in flow and embolus distribution. The computational framework developed enables characterization of the complex interplay of anatomy, hemodynamics, and embolus properties in the context of embolic stroke as well as statistical analysis of embolism risks across common CoW variations.

9:57AM M15.00010 The influence of inflow cannula malposition and left ventricle size on intraventricular thrombosis risk for Left Ventricle Assist Device, V. KESHAV CHIVUKULA, PATRICK MCGAH, Mechanical Engineering, U. Washington, ANTHONY PRISCO, Dept. Medicine, U. Minnesota, JENNIFER BECKMAN, Div. Cardiology, NANUSH MOKADAM, Div. Cardiothoracic Surgery, CLAUDIUS MAHR, Div. Cardiology, ALBERTO ALISEDA, Mechanical Engineering, U. Washington — Patients with left ventricular assist devices (LVAD) have high incidence of thrombosis and stroke. Patient-specific left ventricle 3D models were created with different LVAD inflow cannula angles with the ventricle axis (in increments of $\pm 7^\circ$). Left ventricle sizes ranging from 4 – 7.5 cm in diameter were studied. The aim is optimizing inflow cannula selection and alignment in LVAD patients by understanding the roles that misalignment of the cannula and ventricle size variability play in platelet activation, residence time and agglomeration. Unsteady CFD simulations with patient-specific boundary conditions were performed, and particle tracking was conducted for 10 cardiac cycles to compute the residence time and shear stress histories of platelets for different configurations. Eulerian and Lagrangian metrics, as well as a newly-developed thrombogenic potential score were calculated and used to assess the thrombogenic risk associated with the inflow cannula. Results indicate that inflow cannula misalignment can significantly increase the risk of thrombosis. Cannula sizing without ventricle size consideration affects thrombogeneity for patients outside the normal range (5-6 cm). A methodology is outlined for minimization of thrombotic potential in LVAD implantation strategies

Tuesday, November 22, 2016 8:00AM - 10:10AM –

Session M16 Drops: Electric Field Effects D133/134 - David Saintillan, University of California San Diego

8:00AM M16.00001 Electrohydrodynamics of drops in strong electric fields: Simulations and theory¹, DAVID SAINTILLAN, DEBASISH DAS, University of California San Diego — Weakly conducting dielectric liquid drops suspended in another dielectric liquid exhibit a wide range of dynamical behaviors when subject to an applied uniform electric field contingent on field strength and material properties. These phenomena are best described by the much celebrated Maylor-Taylor leaky dielectric model that hypothesizes charge accumulation on the drop-fluid interface and prescribes a balance between charge relaxation, the jump in Ohmic currents and charge convection by the interfacial fluid flow. Most previous numerical simulations based on this model have either neglected interfacial charge convection or restricted themselves to axisymmetric drops. In this work, we develop a three-dimensional boundary element method for the complete leaky dielectric model to systematically study the deformation and dynamics of liquid drops in electric fields. The inclusion of charge convection in our simulation permits us to investigate drops in the Quincke regime, in which experiments have demonstrated symmetry-breaking bifurcations leading to steady electrorotation. Our simulation results show excellent agreement with existing experimental data and small deformation theories.

¹ACSPRF Grant 53240-ND9

8:13AM M16.00002 Simulating the electrohydrodynamics of a viscous droplet, MAXIME THEILLARD, DAVID SAINTILLAN, University of California, San Diego — We present a novel numerical approach for the simulation of viscous drop placed in an electric field in two and three spatial dimensions. Our method is constructed as a stable projection method on Quad/Octree grids. Using a modified pressure correction we were able to alleviate the standard time step restriction incurred by capillary forces. In weak electric fields, our results match remarkably well with the predictions from the Taylor-Melcher leaky dielectric model. In strong electric fields the so-called Quincke rotation is correctly reproduced.

8:26AM M16.00003 Quincke rotation of an ellipsoid¹, PETIA VLAHOVSKA, QUENTIN BROSSEAU, Brown University — The Quincke effect – spontaneous spinning of a sphere in a uniform DC electric field – has attracted considerable interest in recent year because of the intriguing dynamics exhibited by a Quincke-rotating drop and the emergent collective behavior of confined suspensions of Quincke-rotating spheres. Shape anisotropy, e.g., due to drop deformation or particle asphericity, is predicted to give rise to complex particle dynamics. Analysis of the dynamics of rigid prolate ellipsoid in a uniform DC electric field shows two possible stable states characterized by the orientation of the ellipsoid long axis relative to the applied electric field : spinless (parallel) and spinning (perpendicular). Here we report an experimental study testing the theoretical predictions. The phase diagram of ellipsoid behavior as a function of field strength and aspect ratio is in close agreement with theory. We also investigated the dynamics of the ellipsoidal Quincke roller: an ellipsoid near a planar surface with normal perpendicular to the field direction. We find novel behaviors such as swinging (long axis oscillating around the applied field direction) and tumbling due to the confinement.

¹Supported by NSF CBET awards 1437545 and 1544196

8:39AM M16.00004 Electrostatic drops in orbit¹, ISABEL J. RODRIGUEZ, ERIN SCHMIDT, MARK M. WEISLOGEL, Portland State Univ, DONALD PETTIT, NASA Johnson Space Center — We present what we think are the first intentional electrostatic orbits in the near-weightless environment of a drop tower. Classical physics problems involving Coulombic forces in orbital mechanics have traditionally been confined to thought experiments due to practical terrestrial experimental limitations, namely, the preponderance of gravity. However, the use of a drop tower as an experimental platform can overcome this challenge for brief periods. We demonstrate methanol-water droplets in orbit around a variety of charged objects- some of which can be used to validate special cases of N-body systems. Footage collected via a high-speed camera is analyzed and orbital trajectories are compared with existing theoretical predictions. Droplets of diameters 0.5 to 2mm in a variety of orbits are observed. Due to the repeatability of drop tower initial conditions and effective low-g environment, such experiments may be used to construct empirical analogues and confirm analyses toward the benefit of other fields including space and planetary science.

¹NASA Cooperative Agreement NNX12A047A, Portland State LSAMP, Robert E. McNair Scholars Program

8:52AM M16.00005 Spontaneous Droplet Jump with Electro-Bouncing¹ , ERIN SCHMIDT,

MARK WEISLOGEL, Portland State University — We investigate the dynamics of water droplet jumps from superhydrophobic surfaces in the presence of an electric field during a step reduction in gravity level. In the brief free-fall environment of a drop tower, when a strong non-homogeneous electric field (with a measured strength between 0.39 and 2.36 kV/cm) is imposed, body forces acting on the jumped droplets are primarily supplied by polarization stress and Coulombic attraction instead of gravity. The droplet charge, measured to be on the order of $2.3 \cdot (10^{-11})$ C, originates by electro-osmosis of charged species at the (PTFE coated) hydrophobic surface interface. This electric body force leads to a droplet bouncing behavior similar to well-known phenomena in 1-g, though occurring for larger drops ~ 0.1 mL for a given range of impact Weber numbers, $We < 20$. In 1-g, for $We > 0.4$, impact recoil behavior on a super-hydrophobic surface is normally dominated by damping from contact line hysteresis and by air-layer interactions. However, in the strong electric field, the droplet bounce dynamics additionally include electrohydrodynamic effects on wettability and Cassie-Wenzel transition. This is qualitatively discussed in terms of coefficients of restitution and trends in contact time.

¹This work was supported primarily by NASA Cooperative Agreement NNX12A047A.

9:05AM M16.00006 Electrodynamic behavior and interface instability of double emulsion droplets under high electric field , MUHAMMAD SALMAN ABBASI, RYUNGEUN SONG, JAEHOON KIM, JINKEE LEE, Sungkyunkwan University

— In this paper, numerical solution of electro-dynamic behavior and interface instability of double emulsion droplet is presented. Level set method and leaky dielectric model coupled with Navier-Stokes equation are used to solve the electrodynamic problem. The method is validated against the theoretical analysis and the simulation results of the other researchers. Double emulsion droplet with inner droplet (core) and outer droplet (shell) phases immersed in continuous phase is subjected to high electric field. Shell/continuous and core/shell interfaces of the droplet undergo prolate-oblate or oblate-prolate deformation depending on the extent of the penetration of electric potential and sense of charge distribution at the interfaces. The deformation of the shell deviates from theory at larger volume fraction of core for oblate-prolate case whereas it follows theory for prolate-oblate case. The interfaces showing oblate-prolate deformation split away at the poles whereas, for prolate-oblate, they split at the equator. The re-union of the two split parts under high electric field results with production of daughter droplet at the core. The large decrease in critical electric field for oblate-prolate case shows their less interface stability at larger volume fraction of core. When the core is eccentric, the electric field drives it towards the shell center or to the shell/continuous interface depending on electrical parameters.

9:18AM M16.00007 The Electrostatic Bell , GUILLAUME MARTROU, MARC LEONETTI, Aix Marseille Univ, CNRS, Centrale Marseille, IRPHE, Marseille, France

— An initially static fluid-fluid interface is known to become unstable under a strong electric field leading to jet instability, surface pattern and spout formation. Applying an electric field to an initial dripping mode accelerates the dripping rate and leads to a continuous jet mode. We show that those two different configurations, when applied to dielectric liquids, can lead to the same instability, the formation of an unexpected macroscopic fluid bell-shape of typical size few times the capillary length even if the needle is as small as $200 \mu\text{m}$. The instability results from the competition between the dielectric and the gravity forces, reminiscent of the Taylor-Melcher mechanism. The study is performed on several fluids of various densities, permittivity and surface tension on a large range of electric field. We show that the transition is an imperfect subcritical bifurcation with its characteristic bottleneck effect (lag time). Finally, in the case of flow rate, we established a shape diagram with four domains corresponding to dripping, jetting, bridge and electrostatic bell.

9:31AM M16.00008 Self-Similar Apical Sharpening of an Ideal Perfecting Conducting Fluid Subject to Maxwell Stresses , CHENGZHE ZHOU , SANDRA M. TROIAN, California Institute of Technology, 1200 E. California Blvd. MC 128-95, Pasadena, CA 91125

— We examine the apical behavior of an ideal, perfectly conducting incompressible fluid surrounded by vacuum in circumstances where the capillary, Maxwell and inertial forces contribute to formation of a liquid cone. A previous model based on potential flow [1] describes a family of self-similar solutions with conic cusps whose interior angles approach the Taylor cone angle. These solutions were obtained by matching powers of the leading order terms in the velocity and electric field potential to the asymptotic form dictated by a stationary cone shape. In re-examining this earlier work, we have found a more important, neglected leading order term in the velocity and field potentials, which satisfies the governing, interfacial and far-field conditions as well. This term allows for the development of additional self-similar, sharpening apical shapes, including time reversed solutions for conic tip recoil after fluid ejection. We outline the boundary-element technique [2] for solving the exact similarity solutions, which have parametric dependence on the far-field conditions, and discuss consequences of our findings.

[1] N. M. Zubarev, JETP **73**, 544-548 (2001)

[2] R. M. S. M. Schulkes, JFM **261**, 223-252 (1994)

9:44AM M16.00009 Electric field enhanced dropwise condensation on hydrophobic surfaces¹ , DAVOOD BARATIAN, HARMEN HOEK, DIRK VAN DEN ENDE, FRIEDER MUGELE, University of Twente, PHYSICS OF COMPLEX FLUIDS TEAM

— Dropwise condensation occurs when vapor condenses on a low surface energy surface, and the substrate is just partially wetted by the condensate. Dropwise condensation has attracted significant attention due to its reported superior heat transfer performance compared to filmwise condensation. Extensive research efforts are focused on how to promote, and enhance dropwise condensation by considering both physical and chemical factors. We have studied electrowetting-actuated condensation on hydrophobic surfaces, aiming for enhancement of heat transfer in dropwise condensation. The idea is to use suitably structured patterns of micro-electrodes that generate a heterogeneous electric field at the interface and thereby promote both the condensation itself and the shedding of condensed drops. Comforting the shedding of droplets on electrowetting-functionalized surfaces allows more condensing surface area for re-nucleation of small droplets, leading to higher condensation rates. Possible applications of this innovative concept include heat pipes for (micro) coolers in electronics as well as in more efficient heat exchangers.

¹We acknowledge financial support by the Dutch Technology Foundation STW, which is part of the Netherlands Organization for Scientific Research (NWO), within the VICI program.

9:57AM M16.00010 Streaming instability at the equator of an oblately deformed drop , QUENTIN BROSSEAU, PETIA VLAHOVSKA, Brown University

— Electrohydrodynamic streaming and jet formation from the conical tips formed at the poles of a highly conducting drop in strong electric field is a well known phenomenon. Here we report a novel streaming-like instability occurring with drops less conducting than the suspending medium. In a uniform DC electric field, the drop deforms into a flattened oblate (pancake-like) shape with cusped rim around the equator. The rim emits concentric threads which subsequently break up into tiny droplets forming a Saturn-like rings of droplets around the mother drop. The rate of droplet production is much larger than the classical tip-streaming and suggest a potential new route for "electroemulsification".

Tuesday, November 22, 2016 8:00AM - 10:10AM –

Session M17 Reacting Flows: Ignition and Extinction D131 - Jesse Capecelatro, University of Michigan

8:00AM M17.00001 Premixed autoignition in compressible turbulence¹, ADITYA KONDURI, HEMANTH KOLLA, ALEXANDER KRISMAN, JACQUELINE CHEN, Combustion Research Facility, Sandia National Laboratories — Prediction of chemical ignition delay in an autoignition process is critical in combustion systems like compression ignition engines and gas turbines. Often, ignition delay times measured in simple homogeneous experiments or homogeneous calculations are not representative of actual autoignition processes in complex turbulent flows. This is due to the presence of turbulent mixing which results in fluctuations in thermodynamic properties as well as chemical composition. In the present study the effect of fluctuations of thermodynamic variables on the ignition delay is quantified with direct numerical simulations of compressible isotropic turbulence. A premixed syngas-air mixture is used to remove the effects of inhomogeneity in the chemical composition. Preliminary results show a significant spatial variation in the ignition delay time. We analyze the topology of autoignition kernels and identify the influence of extreme events resulting from compressibility and intermittency. The dependence of ignition delay time on Reynolds and turbulent Mach numbers is also quantified.

¹Supported by Basic Energy Sciences, Dept of Energy, United States

8:13AM M17.00002 Dynamics of autoignitive DME/air coflow flames in oscillating flows, SILI DENG, Princeton University, PENG ZHAO, Oakland University, MICHAEL MUELLER, CHUNG LAW, Princeton University — The structure and dynamics of laminar nonpremixed dimethyl ether (DME)/air coflow flames were investigated at elevated temperatures and pressures, conditions at which autoignition times become competitive with flame times. Computations with detailed chemistry were performed for DME and heated coflow air at 30 atm with uniform but sinusoidally oscillating inlet velocities. These unsteady cases were compared with steady flames to elucidate the effect of oscillation frequency on the flame dynamics. In the oscillating reacting flow, periodic but hysteretic transition occurs between a multibrachial autoignition front that locates downstream at high inlet velocity and a tribrachial flame that locates upstream at low inlet velocity. The finite induction time for autoignition results in this hysteretic behavior, which diminishes at lower oscillation frequency as there is more time for chemistry to respond to the hydrodynamic changes and consequently approach steady state.

8:26AM M17.00003 Fluid-Plasma-Combustion Coupling Effects on the Ignition of a Fuel Jet¹, LUCA MASSA, Virginia Tech, JONATHAN FREUND, University of Illinois — We analyze the effect of plasma-combustion coupling on the ignition and flame supported by a DBD interacting with a jet of H₂ in a air cross-flow. We propose that plasma-combustion coupling is due to the strong temperature-dependence of specific collisional energy loss as predicted by the Boltzmann equation, and that e⁻ transport can be modeled by assuming a form for the E-field pulse in microstreamers. We introduce a two-way coupling based on the Boltzmann equation and the charged species conservation. The addition of this mechanism to a hydrogen combustion scheme leads to an improvement of the ignition prediction and of the understanding of the effect of the plasma on the flow. The key points of the analysis are 1) explanation of the mechanism for the two-stage ignition and quenching observed experimentally, 2) explanation of the existence of a power threshold above which the plasma is beneficial to the ignition probability, 3) understanding of the increase in power absorbed by the plasma in burning conditions and the reduction in power absorbed with an increase in the cross velocity, 4) explanation of the non-symmetric emissions and the increase in luminescence at the rotovibrational H₂O band. The model is validated in part against air-H₂ flow experiments.

¹This material is based in part upon work supported by the Department of Energy, National Nuclear Security Administration, under Award Number DE-NA0002374.

8:39AM M17.00004 A Computational and Experimental Study of Ignition Behavior of Gasoline Surrogate Fuels Under Low-Temperature Combustion Conditions, J. HAN, D.C. HAWORTH, Penn State, V.B. KALASKAR, A.L. BOEHMAN, University of Michigan — One strategy for next-generation engines is low-temperature compression ignition of gasoline. Reaction pathways that are not relevant for high-temperature flame propagation are activated under these conditions, and the ignition behavior of these fuels under low-temperature conditions has not been widely explored. Here the ignition behavior of gasoline and two- and three-component surrogates has been studied experimentally and computationally over a range of operating conditions of interest for low-temperature engine combustion. Experiments were performed in a single-cylinder research engine. For each fuel blend, the critical compression ratio (lowest compression ratio at which the main ignition occurs) was determined over a range of operating conditions, by varying one parameter at a time with all other parameters held fixed. A simplified CFD model that considers detailed chemical kinetics was used to simulate the experiment. The focus of the study is to determine which surrogate fuel mixtures and chemical mechanisms are able to capture the ignition behavior of gasoline under these conditions. For example, different ignition behavior is found for different surrogate mixtures that all have the same Research Octane Number, and it is important to capture this behavior in CFD models.

8:52AM M17.00005 Spark ignition of aviation fuel in isotropic turbulence¹, ALEX KRISMAN, Sandia National Laboratories, Livermore, TIANFENG LU, Universitiy of Connecticut, GIULIO BORGHESI, JACQUELINE CHEN, Sandia National Laboratories, Livermore — Turbulent spark ignition occurs in combustion engines where the spark must establish a viable flame kernel that leads to stable combustion. A competition exists between kernel growth, due to flame propagation, and kernel attenuation, due to flame stretch and turbulence. This competition can be measured by the Karlovitz number, Ka , and kernel viability decreases rapidly for $Ka \gg 1$. In this study, the evolution of an initially spherical flame kernel in a turbulent field is investigated at two cases: Ka_- ($Ka = 25$) and Ka_+ ($Ka = 125$) using direct numerical simulation (DNS). A detailed chemical mechanism for jet fuel (Jet-A) is used, which is relevant for many practical conditions, and the mechanism includes a pyrolysis sub-model which is important for the ignition of large hydrocarbon fuels. An auxiliary non-reacting DNS generates the initial field of isotropic turbulence with a turbulent Reynolds number of 500 (Ka_-) and 1,500 (Ka_+). The kernel is then imposed at the center of the domain and the reacting DNS is performed. The Ka_- case survives and the Ka_+ case is extinguished. An analysis of the turbulence chemistry interactions is performed and the process of extinction is described.

¹Department of Energy - office of Basic Energy Science under Award No. DE-SC0001198.

9:05AM M17.00006 Optimal ignition placement using nonlinear adjoint looping

UBAID QADRI, PETER SCHMID, Imperial College London, LUCA MAGRI, MATTHIAS IHME, Stanford University — Spark ignition of a turbulent mixture of fuel and oxidizer is a highly sensitive process. Traditionally, a large number of parametric studies are used to determine the effects of different factors on ignition and this can be quite tedious. In contrast, we treat ignition as an initial value problem and seek to find the initial condition that maximizes a given cost function. We use direct numerical simulation of the low Mach number equations with finite rate one-step chemistry, and of the corresponding adjoint equations, to study an axisymmetric jet diffusion flame. We find the $L - 2$ norm of the temperature field integrated over a short time to be a suitable cost function. We find that the adjoint fields localize around the flame front, identifying the most sensitive region of the flow. The adjoint fields provide gradient information that we use as part of an optimization loop to converge to a local optimal ignition location. We find that the optimal locations correspond with the stoichiometric surface downstream of the jet inlet plane. The methods and results of this study can be easily applied to more complex flow geometries.

9:18AM M17.00007 Adjoint-based sensitivity of flames to ignition parameters in non-premixed shear-flow turbulence, JESSE CAPECELATRO, University of Michigan, DANIEL BODONY, JONATHAN FREUND, University of Illinois — The adjoint of the linearized and perturbed compressible flow equations for a mixture of chemically reacting ideal gases is used to assess the sensitivity of ignition in non-premixed shear-flow turbulence. Direct numerical simulations are used to provide an initial prediction, and the corresponding space-time discrete-exact adjoint is used to provide a sensitivity gradient for a specific quantity of interest (QoI). Owing to the ultimately binary outcome of ignition (i.e., it succeeds or fails after some period), a QoI is defined that both quantifies ignition success and varies smoothly near its threshold based on the heat release parameter in a short-time horizon during the ignition process. We use the resulting gradient to quantify the flow properties and model parameters that most affect the initiation of a sustained flame. A line-search algorithm is used to identify regions of high ignition probability and map the boundary between successful and failed ignition. The approach is demonstrated on a non-premixed turbulent shear layer and on a reacting jet-in-crossflow.

9:31AM M17.00008 Numerical solution of an edge flame boundary value problem¹

, BENJAMIN SHIELDS, JONATHAN FREUND, CARLOS PANTANO, University of Illinois at Urbana-Champaign — We study edge flames for modeling extinction, reignition, and flame lifting in turbulent non-premixed combustion. An adaptive resolution finite element method is developed for solving a strained laminar edge flame in the intrinsic moving frame of reference of a spatially evolving shear layer. The variable-density zero Mach Navier-Stokes equations are used to solve for both advancing and retreating edge flames. The eigenvalues of the system are determined simultaneously (implicitly) with the scalar fields using a Schur complement strategy. A homotopy transformation over density is used to transition from constant- to variable-density, and pseudo arc-length continuation is used for parametric tracing of solutions. Full details of the edge flames as a function of strain and Lewis numbers will be discussed.

¹This material is based upon work supported [in part] by the Department of Energy, National Nuclear Security Administration, under Award Number DE-NA0002374.

9:44AM M17.00009 Real fuel effects on flame extinction and re-ignition, XINYU ZHAO,

BIFEN WU, CHAO XU, TIANFENG LU, University of Connecticut, JACQUELINE H CHEN, Sandia National Laboratories — Flame-vortex interactions have significant implications in studying combustion in practical aeronautical engines, and can be used to facilitate the model development in capturing local extinction and re-ignition. To study the interactions between the complex fuel and the intense turbulence that are commonly encountered in engines, direct numerical simulations of the interactions between a flame and a vortex pair are carried out using a recently-developed 24-species reduced chemistry for n-dodecane. Both non-premixed and premixed flames with different initial and inlet thermochemical conditions are studied. Parametric studies of different vortex strengths and orientations are carried out to induce maximum local extinction and re-ignition. Chemical-explosive-mode-analysis based flame diagnostic tools are used to identify different modes of combustion, including auto-ignition and extinction. Results obtained from the reduced chemistry are compared with those obtained from one-step chemistry to quantify the effect of fuel pyrolysis on the extinction limit. Effects of flame curvature, heat loss and unsteadiness on flame extinction are also explored. Finally, the validity of current turbulent combustion models to capture the local extinction and re-ignition will be discussed.

9:57AM M17.00010 Localized flame extinction and re-ignition in turbulent jet ignition assisted combustion, ABDOULAHAD VALIDI, HAROLD SCHOCK, FARHAD JABERI, Michigan State University, COMPUTATIONAL FLUID DYNAMICS LABORATORY TEAM — Direct numerical simulations (DNS) of turbulent jet ignition (TJI)-assisted combustion of ultra-lean fuel-air is performed in a three-dimensional planar jet configuration. TJI is a novel ignition enhancement method which facilitates the combustion of lean and ultra-lean mixtures by rapidly exposing them to high temperature combustion products. Fully compressible gas dynamics and species equations are solved with high order finite difference methods. The hydrogen-air reaction is simulated with a detailed chemical kinetics mechanism consisting of 9 species and 38 elementary reactions. The interesting phenomena involved in TJI combustion including localized premixed flame extinction/re-ignition and simultaneous premixed/non-premixed flames are investigated by using the flame heat release, temperature, species concentrations, and a newly defined TJI progress variable.

Tuesday, November 22, 2016 8:00AM - 10:10AM –

Session M18 Flow Instability: Interfacial D135 - Demetrios Papageorgiou, Imperial College

8:00AM M18.00001 Buckling of thin viscous sheets with inhomogenous viscosity under extensional flows, SIDDARTH SRINIVASAN, Harvard University, ZHIYAN WEI, Stanford University, L MAHADEVAN,

Harvard University — We investigate the dynamics, shape and stability of a thin viscous sheet subjected to an extensional flow under an imposed non-uniform temperature field. Using finite element simulations, we first solve for the stretching flow to determine the pre-buckling sheet thickness and in-plane flow velocities. Next, we use this solution as the base state and solve the linearized partial differential equation governing the out-of-plane deformation of the mid-surface as a function of two dimensionless operating parameters: the normalized stretching ratio α and a dimensionless width of the heating zone β . We show the sheet can become unstable via a buckling instability driven by the development of localized compressive stresses, and determine the global shape and growth rates of the most unstable mode. The growth rate is shown to exhibit a transition from stationary to oscillatory modes in region upstream of the heating zone. Finally, we investigate the effect of surface tension and present an operating diagram that indicates regions of the parameter space that minimizes or entirely suppresses the instability while achieving desired outlet sheet thickness. Therefore, our work is directly relevant to various industrial processes including the glass redraw & float-glass method.

8:13AM M18.00002 Dynamics of oil film spreading and dewetting on aqueous substrates

JIE FENG, OREST SHARDT, HOWARD A. STONE, Department of Mechanical and Aerospace Engineering, Princeton University — The spreading and dewetting dynamics of liquids on substrates has been studied intensively in recent years because of their fundamental role in nature and fluid dynamics, as well as practical relevance to many technological processes, such as coating flows. However, little is known about the wetting dynamics in a state called pseudo-partial wetting, which can contribute to efficient interfacial emulsification by bubble bursting. Here we describe the dynamics of the rim that forms when an oil film dewets in a pseudo-partial wetting state on an aqueous substrate. We observe that the rim around the expanding hole displays an instability which leads to the rim break-up into a series of humps. By using confocal microscopy and systematically manipulating the parameters of the multi-phase system, we quantify the dynamics of the oil rim and the formation of humps. We further study the mechanism underlying the break-up of the retracting oil rim. In particular, we theoretically explain the critical conditions at which humps form and highlight the roles of competing time scales during hole expansion and the growth of oil humps. Our work not only contributes to the fundamental understanding of film dynamics in pseudo-partial wetting but also may help improve the understanding and utilization of liquid film flows in industrial processes.

8:26AM M18.00003 Faraday instability in two-fluid mechanically forced rectangular and annular geometries¹

KEVIN WARD, Univ. of Florida, Dept. of Chemical Engineering, Gainesville, FL, USA, FARZAM ZOUESTIAGH, Univ. of Lille 1, IEMN CNRS 8520, Lille, France, RANGA NARAYANAN, Univ. of Florida, Dept. of Chemical Engineering, Gainesville, FL, USA — In this work, we theoretically and experimentally investigate Faraday instability in immiscible two-fluid rectangular and annular systems. Within the examined frequency regime, the selected modes are discretized and experiments for comparison to theory are possible. A stress-free sidewall condition is adopted in the theoretical model, and is realized experimentally through careful selection of the testing fluids. Rectangular geometries offer ease of visualization and testing cell fabrication when compared to cylindrical geometries, but can give rise to discrepancies between ideal theory and experiments due to corner effects and wall damping. Theoretical and experimental results for a large square geometry are first presented to highlight the discrepancies due to corner effects. Next, multiple high aspect ratio rectangular geometries, where corner effects should be suppressed, are shown. Annular geometries of comparable dimension to these rectangular geometries are also presented to confirm the absence of corner effects. Agreement between the theoretical and experimental modes for a given frequency are obtained for all geometries. However, agreement between the predicted and observed threshold amplitude is shown to depend strongly on the cell size due to sidewall damping.

¹Supported by NSF 0968313, CASIS NNH11CD70A, NSF DGE-1315138, and a Chateaubriand Fellowship.

8:39AM M18.00004 Interface structure behind a moving contact line

MENGFEI HE, SIDNEY NAGEL, The University of Chicago — When a flat solid substrate straddles the boundary between two fluids (e.g., water and air), there is a contact line where the two fluids and the solid meet. When the substrate is forced to penetrate further in either direction, it distorts the fluid interface and carries along with it a wedge of the trailing fluid. Numerous studies have investigated the onset of the contact-line motion in a two-dimensional geometry where it was assumed that no flows occurred in the direction along the surface of the substrate transverse to its direction of motion. Contrary to this assumption, we discovered that in steady state the fluid interface develops dramatic three-dimensional structure; there are multiple thin and thick regions of the fluid film alternating in the transverse direction. Thus the dynamics behind the contact line is not invariant in the transverse direction suggesting the existence of a new instability. We use interference to map the relative shape of this wedge-shaped region and a new interference technique to identify the absolute thickness of the wedge. It is particularly noteworthy that the same structure appears both in dewetting (when a substrate is removed from a liquid into the air) and in wetting (when it is plunged into the liquid).

8:52AM M18.00005 Interfacial instability in vertical counter-current gas-liquid film flow: theory, direct numerical simulation and experiment

PATRICK SCHMIDT, The University of Edinburgh, ILJA AUSNER, Sulzer Chemtech, LENNON Ó NÁRAIGH, University College Dublin, MATHIEU LUCQUIAUD, PRASHANT VALLURI, The University of Edinburgh — The dynamics of vertical counter-current gas-liquid flows are largely determined by interfacial instability, which gives rise to a multitude of complex wave patterns and internal flows. To study the genesis and evolution of the instability in detail, we employ theoretical stability analysis, experiment and a newly developed level set method based in-house solver to carry out direct numerical simulations. Crucial results of these simulations, such as growth rate and phase velocity of interfacial waves, are rigorously compared against linear and weakly nonlinear theory; thereby showing remarkable agreement. The analysis also reveals the spatio-temporal character of the waves, depicting regimes of absolute and convective instability. Complementing the benchmark set by (non-)linear theory, we perform film thickness measurements of a real gas-liquid system (air-silicone oil) by means of a non-intrusive light-induced fluorescence technique to further validate the solver regarding its capability of capturing interfacial dynamics accurately. These measurements are in good agreement with the results of the nonlinear direct numerical simulations with respect to wavelength and wave shape of the most unstable mode.

9:05AM M18.00006 Transitional inertialess instabilities in driven multilayer channel flows¹

EVANGELOS PAPAEFTHYMIOU², DEMETRIOS PAPAGEORGIOU, Imperial College London — We study the nonlinear stability of viscous, immiscible multilayer flows in channels driven both by a pressure gradient and/or gravity in a slightly inclined channel. Three fluid phases are present with two internal interfaces. Novel weakly nonlinear models of coupled evolution equations are derived and we concentrate on inertialess flows with stably stratified fluids, with and without surface tension. These are 2×2 systems of second-order semilinear parabolic PDEs that can exhibit inertialess instabilities due to resonances between the interfaces - mathematically this is manifested by a transition from hyperbolic to elliptic behavior of the nonlinear flux functions. We consider flows that are linearly stable (i.e. the nonlinear fluxes are hyperbolic initially) and use the theory of nonlinear systems of conservation laws to obtain a criterion (which can be verified easily) that can predict nonlinear stability or instability (i.e. nonlinear fluxes encounter ellipticity as they evolve spatiotemporally) at large times. In the former case the solution decays asymptotically to its base state, and in the latter nonlinear traveling waves emerge.

¹EPSRC grant numbers EP/K041134 and EP/L020564

²Membership Pending

9:18AM M18.00007 Capturing nonlinear dynamics of two-fluid Couette flows with asymptotic models¹, DEMETRIOS PAPAGEORGIOU, RADU CIMPEANU, Imperial College London, ANNA KALOGIROU, University of Leeds, ERIC KEAVENY, Imperial College London — The nonlinear stability of two-fluid Couette flows is studied using a novel evolution equation whose dynamics are validated by direct numerical simulations (DNS). The evolution equation incorporates inertial effects at arbitrary Reynolds numbers through a nonlocal term arising from the coupling between the two fluid regions, and is valid when one of the layers is thin. The equation predicts asymmetric solutions and exhibits bistability as seen in experiments. Related low-inertia models have been used in qualitative predictions using *ad hoc* modifications rather than the direct comparisons carried out here. Comparisons between model solutions and DNS show excellent agreement at Reynolds numbers of $\mathcal{O}(10^3)$ found in experiments. Direct comparisons are also made with the available experimental results of Barthelet et al. (1995) when the thin layer occupies 1/5 of the channel height. Pointwise comparisons of the travelling wave shapes are carried out and once again the agreement is very good.

¹EPSRC grant numbers EP/K041134 and EP/L020564

9:31AM M18.00008 Singularities of the charge transport equation¹, OMAR MATAR, ALEX WRAY, DEMETRIOS PAPAGEORGIOU, Imperial College London, QIMING WANG, New Jersey Institute of Technology — It has long been known (since the work of Taylor in the 60s) that electrohydrodynamic interfacial flows can exhibit singularities exemplified by the so-called 'Taylor Cone', for instance. Despite the large attention devoted to such flows in the literature, achieving fundamental understanding of these singularities has proved elusive. This is also in spite of the observation that certain parameter regimes appear to demonstrate the unusual phenomenon of cusp-like touchdown (as reported by Wray et al., 2013, to be discussed in the talk). Via the use of mathematical analysis, low-order models, and direct numerical simulations, we classify these singularities and isolate their underlying causes. We also demonstrate where they deviate from experimental predictions, and investigate how such discrepancies may be resolved.

¹EPSRC UK platform grant MACIPh (EP/L020564/1) and programme Grant MEMPHIS (EP/K003976/1)

9:44AM M18.00009 Tear Film Dynamics: the roles of complex structure and rheology, MOHAR DEY, JAMES FENG, Univ of British Columbia, Canada, ATUL S. VIVEK, HARISH N. DIXIT, Indian Institute of Technology Hyderabad, India, ASHUTOSH RICHHARIYA, LV Prasad Eye Institute, Hyderabad, India. — Ocular surface infections such as microbial and fungal keratitis are among leading causes of blindness in the world. A thorough understanding of the pre-corneal tear film dynamics is essential to comprehend the role of various tear layer components in the escalation of such ocular infections. The pre-corneal tear film comprises of three layers of complex fluids, viz. the innermost mucin layer, a hydrophilic protective cover over the sensitive corneal epithelium, the intermediate aqueous layer that forms the bulk of the tear film and is often embedded with large number of bio-polymers either in the form of soluble mucins or pathogens, and finally the outermost lipid layer that stabilizes the film by decreasing the air/tear film interfacial tension. We have developed a comprehensive mathematical model to describe such a film by incorporating the effects of the non-uniform mucin distribution along with the complex rheology of the aqueous layer with/without pathogens, Marangoni effects from the lipid layer and the slip effects at the base of the tear film. A detailed linear stability analysis and a fully non-linear solution determine the break up time (BUT) of such a tear film. We also probe the role of the various components of the pre-corneal tear film in the dynamics of rupture.

9:57AM M18.00010 Marangoni-induced symmetry-breaking pattern selection on viscous fluids¹, LI SHEN, FABIAN DENNER, Imperial College London, NEAL MORGAN, Shell Global Solutions Ltd, BEREND VAN WACHEM, DANIELE DINI, Imperial College London — Symmetry breaking transitions on curved surfaces are found in a wide range of dissipative systems, ranging from asymmetric cell divisions to structure formation in thin films. Inherent within the nonlinearities are the associated curvilinear geometry, the elastic stretching, bending and the various fluid dynamical processes. We present a generalised Swift-Hohenberg pattern selection theory on a thin, curved and viscous films in the presence of non-trivial Marangoni effect. Testing the theory with experiments on soap bubbles, we observe the film pattern selection to mimic that of the elastic wrinkling morphology on a curved elastic bilayer in regions of slow viscous flow. By examining the local state of damping of surface capillary waves we attempt to establish an equivalence between the Marangoni fluid dynamics and the nonlinear elastic shell theory above the critical wavenumber of the instabilities and propose a possible explanation for the perceived elastic-fluidic duality.

¹The authors acknowledge the financial support of the Shell University Technology Centre for fuels and lubricants.

Tuesday, November 22, 2016 8:00AM - 9:57AM – Session M19 Bio: Swimming of Fish and Jellyfish D136 - John Dabiri, Stanford University

8:00AM M19.00001 Simultaneous measurements of jellyfish bell kinematics and flow fields using PTV and PIV¹, NICOLE XU, JOHN DABIRI, Stanford University — A better understanding of jellyfish swimming can potentially improve the energy efficiency of aquatic vehicles or create biomimetic robots for ocean monitoring. *Aurelia aurita* is a simple oblate invertebrate composed of a flexible bell and coronal muscle, which contracts to eject water from the subumbrellar volume. Jellyfish locomotion can be studied by obtaining body kinematics or by examining the resulting fluid velocity fields using particle image velocimetry (PIV). Typically, swim kinematics are obtained by semi-manually tracking points of interest (POI) along the bell in video post-processing; simultaneous measurements of kinematics and flows involve using this semi-manual tracking method on PIV videos. However, we show that both the kinematics and flow fields can be directly visualized in 3D space by embedding phosphorescent particles in animals free-swimming in seeded environments. Particle tracking velocimetry (PTV) can then be used to calculate bell kinematics, such as pulse frequency, bell deformation, swim trajectories, and propulsive efficiency. By simultaneously tracking POI within the bell and collecting PIV data, we can further study the jellyfish's natural locomotive control mechanisms in conjunction with flow measurements.

¹NSF GRFP

8:13AM M19.00002 To flap or not to flap: a discussion between a fish and a jellyfish¹

, NATHAN MARTIN, CHRIS ROH, Caltech, SUHAIR IDREES, University of Cambridge, MORTEZA GHARIB, Caltech — Fish and jellyfish are known to swim by flapping and by periodically contracting respectively, but which is the more effective propulsion mechanism? In an attempt to answer this question, an experimental comparison is made between simplified versions of these motions to determine which generates the greatest thrust for the least power. The flapping motion is approximated by pitching plates while periodic contractions are approximated by clapping plates. A machine is constructed to operate in either a flapping or a clapping mode between Reynolds numbers 1,880 and 11,260 based on the average plate tip velocity and span. The effect of the total sweep angle, total sweep time, plate flexibility, and duty cycle are investigated. The average thrust generated and power required per cycle are compared between the two modes when their total sweep angle and total sweep time are identical. In general, operating in the clapping mode required significantly more power to generate a similar thrust compared to the flapping mode. However, modifying the duty cycle for clapping caused the effectiveness to approach that of flapping with an unmodified duty cycle. These results suggest that flapping is the more effective propulsion mechanism within the range of Reynolds numbers tested.

¹This work was supported by the Charyk Bio-inspired Laboratory at the California Institute of Technology, the National Science Foundation Graduate Research Fellowship under Grant No. DGE-1144469, and the Summer Undergraduate Research Fellowships program.

8:26AM M19.00003 On the dynamics of jellyfish locomotion via 3D particle tracking velocimetry

, MATTHEW PIPER, JIN-TAE KIM, LEONARDO P. CHAMORRO, University of Illinois at Urbana-Champaign — The dynamics of jellyfish (*Aurelia aurita*) locomotion is experimentally studied via 3D particle tracking velocimetry. 3D locations of the bell tip are tracked over 1.5 cycles to describe the jellyfish path. Multiple positions of the jellyfish bell margin are initially tracked in 2D from four independent planes and individually projected in 3D based on the jellyfish path and geometrical properties of the setup. A cubic spline interpolation and the exponentially weighted moving average are used to estimate derived quantities, including velocity and acceleration of the jellyfish locomotion. We will discuss distinctive features of the jellyfish 3D motion at various swimming phases, and will provide insight on the 3D contraction and relaxation in terms of the locomotion, the steadiness of the bell margin eccentricity, and local Reynolds number based on the instantaneous mean diameter of the bell.

8:39AM M19.00004 Spatially constrained propulsion in jumping archer fish

, LEAH MENDELSON, ALEXANDRA TECHET, MIT — Archer fish jump multiple body lengths out of the water for prey capture with impressive accuracy. Their remarkable aim is facilitated by jumping from a stationary position directly below the free surface. As a result of this starting position, rapid acceleration to a velocity sufficient for reaching the target occurs with only a body length to travel before the fish leaves the water. Three-dimensional measurements of jumping kinematics and volumetric velocimetry using Synthetic Aperture PIV highlight multiple strategies for such spatially constrained acceleration. Archer fish rapidly extend fins at jump onset to increase added mass forces and modulate their swimming kinematics to minimize wasted energy when the body is partially out of the water. Volumetric measurements also enable assessment of efficiency during a jump, which is crucial to understanding jumping's role as an energetically viable hunting strategy for the fish.

8:52AM M19.00005 Increased Thrust through Passively Variable Tail Stiffness in Fast Starting Fish, TODD CURRIER, GANZHONG MA, YAHYA MODARRES-SADEGHI, Univ of Mass - Amherst — An experimental study is conducted in the effect of tail stiffness on increased acceleration in mechanisms designed to emulate fast-start fish maneuvers. The variable stiffness is characterized by the directionality of loading. As load is applied in one direction on the fin the structure is flexible, simulating the preparatory stage of the maneuver, and as load is applied in the opposing direction the fin rigidly maintains its shape during the propulsive stage. A 3D printed fin structure is used to achieve the directional stiffness and is tested dynamically. Thrust is measured at various rates of rotation studying the influence of timing on peak acceleration.

9:05AM M19.00006 Spatial organization and Synchronization in collective swimming of *Hemigrammus bleheri*¹, INTESAAF ASHRAF, THANH-TUNG HA, RAMIRO GODOY-DIANA, BENJAMIN THIRIA, PMMH, CNRS, PSL-ESPCI, Paris 06, Paris 07, JOSE HALLOY, BERTRAND COLLIGNON , LIED, CNRS, Paris 07, LABORATOIRE DE PHYSIQUE ET MCANIQUE DES MILIEUX HTROGNES (PMMH) TEAM, LABORATOIRE INTERDISCIPLINAIRE DES ENERGIES DE DEMAIN (LIED) TEAM — In this work, we study the collective swimming of *Hemigrammus bleheri* fish using experiments in a shallow swimming channel. We use high-speed video recordings to track the midline kinematics and the spatial organization of fish pairs and triads. Synchronizations are characterized by observance of "out of phase" and "in phase" configurations. We show that the synchronization state is highly correlated to swimming speed. The increase in synchronization led to efficient swimming based on Strouhal number. In case of fish pairs, the collective swimming is 2D and the spatial organization is characterized by two characteristic lengths: the lateral and longitudinal separation distances between fish pairs. For fish triads, different swimming patterns or configurations are observed having three dimensional structures. We performed 3D kinematic analysis by employing 3D reconstruction using the Direct Linear Transformation (DLT). We show that fish still keep their nearest neighbor distance (NND) constant irrespective of swimming speeds and configuration. We also point out characteristic angles between neighbors, hence imposing preferred patterns. At last we will give some perspectives on spatial organization for larger population.

¹Sorbonne Paris City College of Doctoral Schools. European Union Information and Communication Technologies project ASSISIbf, FP7-ICT-FET- 601074.

9:18AM M19.00007 Mechanisms of force production during linear accelerations in bluegill sunfish *Lepomis macrochirus*¹, ERIC D. TYTELL, TYLER N. WISE, ALEXANDRA L. BODEN, ERIN K. SANDERS, MARGOT A. B. SCHWALBE, Tufts Univ — In nature, fish rarely swim steadily. Although unsteady behaviors are common, we know little about how fish change their swimming kinematics for routine accelerations, and how these changes affect the fluid dynamic forces and the wake produced. To study force production during acceleration, particle image velocimetry was used to quantify the wake of bluegill sunfish *Lepomis macrochirus* and to estimate the pressure field during linear accelerations and steady swimming. We separated "steady" and "unsteady" trials and quantified the forward acceleration using inertial measurement units. Compared to steady sequences, unsteady sequences had larger accelerations and higher body amplitudes. The wake consisted of single vortices shed during each tail movement (a '2S' wake). The structure did not change during acceleration, but the circulation of the vortices increased, resulting in larger forces. A fish swimming unsteadily produced significantly more force than the same fish swimming steadily, even when the accelerations were the same. This increase is likely due to increased added mass during unsteady swimming, as a result of the larger body amplitude. Pressure estimates suggest that the increase in force is correlated with more low pressure regions on the anterior body.

¹This work was supported by ARO W911NF-14-1-0494 and NSF RCN-PLS 1062052

9:31AM M19.00008 Kinematics and flow fields in 3D around swimming lamprey using light field PIV, ANDREA M. LEHN, ALEXANDRA H. TECHET, MIT — The fully time-resolved 3D kinematics and flow field velocities around freely swimming sea lamprey are derived using 3D light field imaging PIV. Lighthill's Elongated Body Theory (EBT) predicts that swimmers with anguilliform kinematics likened to lamprey, and similarly eels, will exhibit relatively poor propulsive efficiency. However, previous experimental studies of eel locomotion utilizing 2D PIV suggest disagreement with EBT estimates of wake properties; although, the thrust force generated by such swimmers has yet to be fully resolved using 3D measurements. A light field imaging array of multiple high-speed cameras is used to perform 3D synthetic aperture PIV around ammocoete sea lamprey (*Petromyzon marinus*). Fluid mechanics equations are used to determine thrust force generation, leading experimental studies closer to underpinning the physical mechanisms that enable aquatic locomotion of long, slender undulatory swimmers.

9:44AM M19.00009 Experimental measurement of dolphin thrust generated during a tail stand using DPIV, TIMOTHY WEI, University of Nebraska - Lincoln, FRANK FISH, West Chester University, TERRIE WILLIAMS, University of California - Santa Cruz, VICKI WU, Boeing Corp, ERICA SHERMAN, MITCHEL MISFELDT, HUNTER RINGENBERG, DYLAN ROGERS, University of Nebraska - Lincoln — The thrust generated by dolphins doing tail stands was measured using DPIV. The technique entailed measuring vortex strength associated with the tail motion and correlating it to above water video sequences showing the amount of the dolphin's body that was being lifted out of the water. The underlying drivers for this research included: i) understanding the physiology, hydrodynamics and efficiency of dolphin locomotion, ii) developing non-invasive measurement techniques for studying marine swimming and iii) quantifying the actual propulsive capabilities of these animals. Two different bottlenose dolphins at the Long Marine Lab at UC-Santa Cruz were used as test subjects. Application of the Kutta-Joukowski Theorem on measured vortex circulations yielded thrust values that were well correlated with estimates of dolphin body weight being supported above water. This demonstrates that the tail motion can be interpreted as a flapping hydrofoil that can generate a sustained thrust roughly equal to the dolphin's weight. Videos of DPIV measurements overlaid with the dolphins will be presented along with thrust/weight data.

Tuesday, November 22, 2016 8:00AM - 10:10AM – Session M20 Bio: General Topics II D137-138 - Aloke Kumar, University of Alberta

8:00AM M20.00001 Failure of bacterial streamers in creeping flows, ISHITA BISWAS, Department of Mechanical Engineering, University of Alberta, Edmonton, T6G 2G8, RANAJAY GHOSH, Department of Mechanical and Aerospace Engineering, University of Central Florida, Orlando FL 32816, MOHTADA SADRZADEH, ALOKE KUMAR, Department of Mechanical Engineering, University of Alberta, Edmonton, T6G 2G8 — In the recent years, the dynamical response of filamentous bacterial aggregates called bacterial streamer in creeping flows has attracted attention. We report the observation of 'necking-type' instability leading to failure in bacterial (*Pseudomonas fluorescens*) streamers formed in creeping flows. Quantification of the failure process was made possible through the use of 200 nm red fluorescent polystyrene tracer particles embedded in the bacterial extracellular polymeric substances (EPS). The nonlinear failure behavior shows distinct phases of deformation with mutually different characteristic times, which end with a distinct localized failure of the streamer. We also develop a simplified analytical model to describe the experimental observations of the failure phenomena. The theoretical power law relationship between critical stretch ratio and the fluid velocity scale matches closely experimental observations.

8:13AM M20.00002 Coupled gel spreading and diffusive transport models describing microbicidal drug delivery, CLAIRE FUNKE, KELSEY MACMILLAN, Univ of California - Berkeley, ANTHONY S. HAM, ImQuest BioSciences, ANDREW J. SZERI, Univ of California - Berkeley, DAVID F. KATZ, Duke University — Gels are a drug delivery platform being evaluated for application of active pharmaceutical ingredients, termed microbicides, that act topically against infection by sexually transmitted HIV. Despite success in one Phase IIb trial of a vaginal gel delivering tenofovir, problems of user adherence to designed gel application regimen compromised results in two other trials. The microbicide field is responding to this issue by simultaneously analyzing behavioral determinants of adherence and pharmacological determinants of drug delivery. Central to both user adherence and mucosal drug delivery are gel properties (e.g. rheology) and applied volume. The specific problem to be solved here is to develop a model for how gel rheology and volume, interacting with loaded drug concentration, govern the transport of the microbicide drug tenofovir into the vaginal mucosa to its stromal layer. The analysis here builds upon our current understanding of vaginal gel deployment and drug delivery, incorporating key features of the gel's environment, fluid production and subsequent gel dilution, and vaginal wall elasticity. We consider the microbicide drug tenofovir as it is the most completely studied drug, in both *in vitro* and *in vivo* studies, for use in vaginal gel application. Our goal is to contribute to improved pharmacological understanding of gel functionality, providing a computational tool that can be used in future vaginal microbicide gel design.

8:26AM M20.00003 Biofluid dynamics of two phase stratified flow through flexible membranes, DINESH BHAGAVATULA NVSSR, PUSHPAVANAM S, None — Two phase stratified flows between flexible membranes arise in biological flows like lung airway reopening, blood flow in arteries and movement of spinal cord. It is important to understand the physics behind the interaction of flexible membranes and the fluid flow. In this work, a theoretical model is developed and different types of instabilities that arise due to the fluid flow are understood. The solid membrane is modeled as an incompressible linear viscoelastic solid. To simplify the analysis, inertia in the solid is neglected. Linear stability analysis is carried around the base state velocity of the fluid and displacement field of the solid. The flow is perturbed by a small disturbance and a normal mode analysis is carried out to study the growth rate of the disturbance. An eigenvalue problem is formulated using Chebyshev spectral method and is solved to obtain the growth rate of the disturbance. The effect of different parameters such as thickness of the flexible membrane, Reynolds number, viscosity ratio, density ratio, Capillary number and Weissenberg number on the stability characteristics of the flow is studied in detail. Dispersion curves are obtained which explain the stability of the flow. A detail energy analysis is carried out to determine different ways through which energy transfers from the base flow to the disturbed flow.

8:39AM M20.00004 Transition of torque pattern in undulatory locomotion due to wave number variation in resistive force dominated media¹, YANG DING, TINGYU MING, Beijing Computational Science Research Center — In undulatory locomotion, torque (bending moment) is required along the body to overcome the external forces from environments and bend the body. Previous observations on animals using less than two wavelengths on the body showed such torque has a single traveling wave pattern. Using resistive force theory model and considering the torque generated by external force in a resistive force dominated media, we found that as the wave number (number of wavelengths on the locomotor's body) increases from 0.5 to 1.8, the speed of the traveling wave of torque decreases. When the wave number increases to 2 and greater, the torque pattern transits from a single traveling wave to a two traveling waves and then a complex pattern that consists two wave-like patterns. By analyzing the force distribution and its contribution to the torque, we explain the speed decrease of the torque wave and the pattern transition.

¹This research is partially supported by the Recruitment Program of Global Young Experts (China)

8:52AM M20.00005 Mechanics of kinetochore microtubules and their interactions with chromosomes during cell division, EHSSAN NAZOCKDAST, SEBASTIAN FRTHAUER, Simons Foundation, STEPHANIE REDEMANN, Technische Universität Dresden, JOHANNES BAUMGART, Max Planck Institute for the Physics of Complex Systems, NORBERT LINDOW, ANDREA KRATZ, STEFFEN PROHASKA, Zuse Institute, Berlin, THOMAS MLLER-REICHERT, Technische Universität Dresden, MICHAEL SHELLEY, Simons Foundation, Courant Institute (NYU) — The accurate segregation of chromosomes, and subsequent cell division, in Eukaryotic cells is achieved by the interactions of an assembly of microtubules (MTs) and motor-proteins, known as the mitotic spindle. We use a combination of our computational platform for simulating cytoskeletal assemblies and our structural data from high-resolution electron tomography of the mitotic spindle, to study the kinetics and mechanics of MTs in the spindle, and their interactions with chromosomes during chromosome segregation in the first cell division in *C.elegans* embryo. We focus on kinetochore MTs, or KMTs, which have one end attached to a chromosome. KMTs are thought to be a key mechanical component in chromosome segregation. Using exploratory simulations of MT growth, bending, hydrodynamic interactions, and attachment to chromosomes, we propose a mechanical model for KMT-chromosome interactions that reproduces observed KMT length and shape distributions from electron tomography. We find that including detailed hydrodynamic interactions between KMTs is essential for agreement with the experimental observations.

9:05AM M20.00006 Coarse-grained Simulations of Conformational Changes in Multidrug Resistance Transporters¹, S M YEAD JEWEL, PRASHANTA DUTTA, JIN LIU, Washington State University —

The overexpression of multidrug resistance (MDR) systems on the gram negative bacteria causes serious problems for treatment of bacterial infectious diseases. The system effectively pumps the antibiotic drugs out of the bacterial cells. During the pumping process one of the MDR components, AcrB undergoes a series of large-scale conformational changes which are responsible for drug recognition, binding and expelling. All-atom simulations are unable to capture those conformational changes because of computational cost. Here, we implement a hybrid coarse-grained force field that couples the united-atom protein models with the coarse-grained MARTINI water/lipid, to investigate the proton-dependent conformational changes of AcrB. The simulation results in early stage (~100 ns) of proton-dependent conformational changes agree with all-atom simulations, validating the coarse-grained model. The coarse-grained force field allows us to explore the process in microsecond simulations. Starting from the crystal structures of Access(A)/Binding(B)/Extrusion(E) monomers in AcrB, we find that deprotonation of Asp407 and Asp408 in monomer E causes a series of large-scale conformational changes from ABE to AAA in absence of drug molecules, which is consistent with experimental findings.

¹This work is supported by NIH grant: 1R01GM122081-01

9:18AM M20.00007 Investigation of polymeric scaffold degradation for drug delivery and neovascularization applications¹, KARTIK V. BULUSU, MITRA ALIBOUZAR, NATHAN J. CASTRO, LIJIE G. ZHANG, KAUSIK SARKAR, MICHAEL W. PLESNIAK, George Washington University —

Degradable polymer-based prosthetics for the treatment of osseous tissue defects, maxillo-/cranio-facial trauma and brain injury face two common clinical obstacles impeding efficient tissue engraftment i.e., controlled material release and neovascularization. Ascertaining the time scales of polymer degradation for controlled delivery of drugs and nutrients is critical to treatment efficacy and strategy. We incorporated multiple experimental methodologies to understand the driving forces of transport mechanisms in polyvinyl alcohol-based (PVA) 3D-printed scaffolds of different porosity. Scaffold degradation was monitored various pulsatile flow conditions using MEMS-based pressure catheters and an ultrasonic flow rate sensor. Ultrasonic properties (bulk attenuation and sound velocity) were measured to monitor the degradation process in a static, alkaline medium. Viscosity and the absorption spectra variations with PVA-solute concentrations were measured using a rheometer and a spectrophotometer, respectively. A simple mathematical model based on Ficks law of diffusion provides the fundamental description of solute transport from the scaffold matrices. However, macroscopic material release could become anomalous or non-Fickian in complex polymeric scaffold matrices.

¹Supported by the GW Center for Biomimetics and Bioinspired Engineering and NIH Directors New Innovator Award 1DP2EB020549-01.

9:31AM M20.00008 Simulations of heart valves by thin shells with non-linear material properties.¹, IMAN BORAZJANI, HAFEZ ASGHARZADEH, MOHAMMADALI HEDAYAT, The State University of New York at Buffalo —

The primary function of a heart valve is to allow blood to flow in only one direction through the heart. Triangular thin-shell finite element formulation is implemented, which considers only translational degrees of freedom, in three-dimensional domain to simulate heart valves undergoing large deformations. The formulation is based on the nonlinear Kirchhoff thin-shell theory. The developed method is intensively validated against numerical and analytical benchmarks. This method is added to previously developed membrane method to obtain more realistic results since ignoring bending forces can result in unrealistic wrinkling of heart valves. A nonlinear Fung-type constitutive relation, based on experimentally measured biaxial loading tests, is used to model the material properties for response of the in-plane motion in heart valves. Furthermore, the experimentally measured liner constitutive relation is used to model the material properties to capture the flexural motion of heart valves. The Fluid structure interaction solver adopts a strongly coupled partitioned approach that is stabilized with under-relaxation and the Aitken acceleration technique.

¹This work was supported by American Heart Association (AHA) grant 13SDG17220022 and the Center of Computational Research (CCR) of University at Buffalo.

9:44AM M20.00009 Modes of thrust generation in flying animals¹, HAOXIANG LUO, Vanderbilt University, JIALEI SONG, Beijing Computational Science Research Center, BRETT TOBALSKE, University of Montana, LUO TEAM, TOBALSKE TEAM —

For flying animals in forward flight, thrust is usually much smaller as compared with weight support and has not been given the same amount of attention. Several modes of thrust generation are discussed in this presentation. For insects performing slow flight that is characterized by low advance ratios (i.e., the ratio between flight speed and wing speed), thrust is usually generated by a "backward flick" mode, in which the wings move upward and backward at a faster speed than the flight speed. Paddling mode is another mode used by some insects like fruit flies who row their wings backward during upstroke like paddles (Ristroph et al, PRL, 2011). Birds wings have high advance ratios and produce thrust during downstroke by directing aerodynamic lift forward. At intermediate advance ratios around one (e.g., hummingbirds and bats), the animal wings generate thrust during both downstroke and upstroke, and thrust generation during upstroke may come at cost of negative weight support. These conclusions are supported by previous experiment studies of insects, birds, and bats, as well as our recent computational modeling of hummingbirds.

¹Supported by the NSF.

9:57AM M20.00010 Non-invasive quantification of hemodynamics in human choriocapillaries., HUIDAN (WHITNEY) YU¹, ROU CHEN, SENYOU AN², Indiana University-Purdue University Indianapolis (IUPUI), JAMES McDONOUGH, University of Kentucky, BRADLEY GELFAND, School of Medicine, University of Virginia, JUN YAO, China University of Petroleum (East China) — The development of retinal disease is inextricably linked to defects in the choroidal blood supply. However, to date a description of the hemodynamics in the human choroidal circulation is lacking. Through high resolution choroidal vascular network mapped from immunofluorescent labeling and confocal microscopy of human cadaver donor eyes. We noninvasively quantify hemodynamics including velocity, pressure, and wall-shear stress (WSS) in choriocapillaries through mesoscale modeling and GPU-accelerated fast computation. This is the first-ever map of hemodynamic parameters (WSS, pressure, and velocity) in anatomically accurate human choroidal vasculature in health and disease. The pore scale simulation results are used to evaluate porous media models with the same porosity and boundary conditions.

¹School of Medicine, Indiana University

²China University of Petroleum (East China)

Tuesday, November 22, 2016 8:00AM - 9:57AM –
Session M21 Bio: Dynamic Mechanisms of Insect Flight D139-140 - Hamid Vejdani, Brown University

8:00AM M21.00001 Toward understanding the mechanics of hovering in insects, hummingbirds and bats, HAMID VEJDANI, School of Engineering, Brown University, Providence, RI, DAVID BOERMA , SHARON SWARTZ , Department of Ecology and Evolutionary Biology, Brown University, Providence, RI, KENNETH BREUER , School of Engineering, Brown University, Providence, RI — We present results on the dynamical characteristics of two different mechanisms of hovering, corresponding to the behavior of hummingbirds and bats. Using a Lagrangian formulation, we have developed a dynamical model of a body (trunk) and two rectangular wings. The trunk has 3 degrees of freedom (x, z and pitch angle) and each wing has 3 modes of actuation: flapping, pronation/supination, and wingspan extension/flexion (only present for bats). Wings can be effectively massless (hummingbird and insect wings) or relatively massive (important in the case of bats). The aerodynamic drag and lift forces are calculated using a quasi-steady blade-element model. The regions of state space in which hovering is possible are computed by over an exhaustive range of parameters. The effect of wing mass is to shrink the phase space available for viable hovering and, in general, to require higher wingbeat frequency. Moreover, by exploring hovering energy requirements, we find that the pronation angle of the wings also plays a critical role. For bats, who have relatively heavy wings, we show wing extension and flexion is critical in order to maintain a plausible hovering posture with reasonable power requirements. Comparisons with biological data show good agreement with our model predictions.

8:13AM M21.00002 Effects of altitude on the climbing performance of Monarch butterflies¹, CHANG-KWON KANG, MADHU SRIDHAR, DAVID LANDRUM, Univ of Alabama - Huntsville, HIKARU AONO, Tokyo University of Science — Millions of Monarchs annually travel up to 4,000km, the longest migration distance among insects. They fly and overwinter at high altitudes. However, the aerodynamic mechanism enabling the long-range flight of Monarch butterflies is unknown. To study the effects of altitude on the aerodynamic performance of Monarch butterflies, a unique combination of a motion tracking system and a variable pressure chamber that allows controlling the density is used. The condition inside the chamber is systematically varied to simulate high altitude conditions up to 3,000 m. An optical tracking technique is used to characterize the climbing trajectories of freely flying Monarch butterflies. Customized reflective markers are designed to minimize the effects of marker addition. Flapping amplitude and frequency as well as climbing trajectories are measured. Lift acting on the butterfly is also determined by considering the force balance. Results show that the average flight speed and the Reynolds number, in general, decreased with the altitude, whereas, interestingly, the lift coefficient increased with the altitude. More detailed measurements and analyses will be performed in the future to explain the lift enhancement by flying at higher altitudes.

¹This work is partly supported by NSF grant CBET-1335572 and in part by CK's startup fund provided by UAH.

8:26AM M21.00003 Bats dynamically change wingspan to enhance lift and efficiency, SHIZHAO WANG, XING ZHANG, GUOWEI HE, Institute of Mechanics, Chinese Academy of Sciences, TIANSHU LIU, Western Michigan University, TURBULENCE TEAM — Bats can dynamically change the wingspan by controlling the joints on the wings. This work focuses on the effect of dynamically changing wingspan on the lift and efficiency in slow-flying bats. The geometry and kinematics of the bat model is constructed based on the experimental measurements of Wolf et al. (J. Exp. Biol. 213, 2142–2153). The Navier-Stokes equations for incompressible flows are solved numerically to investigate the 3D unsteady flows around the bat model. It is found that the dynamically changing wingspan can significantly enhance the lift and efficiency. The lift enhancement is contributed by both lifting surface area extended during the downstroke and the vortex force associated with the leading-edge vortices intensified by the dynamically changing wingspan. The nonlinear interaction between the dynamically changing wing and the vortex structures plays an important role in the lift enhancement of a slow-flying bat in addition to the geometrical effect of changing the lifting-surface area in a flapping cycle.

8:39AM M21.00004 Investigation of Body-involved Lift Enhancement in Bio-inspired Flapping Flight¹, JUNSHI WANG, GENG LIU, YAN REN, HAIBO DONG, University of Virginia — Previous studies found that insects and birds are capable of using many unsteady aerodynamic mechanisms to augment the lift production. These include leading edge vortices, delayed stall, wake capture, clap-and-fling, etc. Yet the body-involved lift augmentation has not been paid enough attention. In this work, the aerodynamic effects of the wing-body interaction on the lift production in cicada and hummingbird forward flight are computationally investigated. 3D wing-body systems and wing flapping kinematics are reconstructed from the high-speed videos or literatures to keep their complexity. Vortex structures and associated aerodynamic performance are numerically studied by an in-house immersed-boundary-method-based flow solver. The results show that the wing-body interaction enhances the overall lift production by about 20% in the cicada flight and about 28% in the hummingbird flight, respectively. Further investigation on the vortex dynamics has shown that this enhancement is attributed to the interactions between the body-generated vortices and the flapping wings. The output from this work has revealed a new lift enhancement mechanism in the flapping flight.

¹This work is supported by NSF CBET-1313217 and AFOSR FA9550-12-1-0071

8:52AM M21.00005 Role of passive deformation on propulsion through a lumped torsional flexibility model¹, NIPUN ARORA, AMIT GUPTA, Indian Institute of Technology Delhi — Scientists and biologists have been affianced in a deeper examination of insect flight to develop an improved understanding of the role of flexibility on aerodynamic performance. Here, we mimic a flapping wing through a fluid-structure interaction framework based upon a lumped torsional flexibility model. The developed fluid and structural solvers together determine the aerodynamic forces and wing deformation, respectively. An analytical solution to the simplified single-spring structural dynamics equation is established to substantiate simulations. It is revealed that the dynamics of structural deformation is governed by the balance between inertia, stiffness and aerodynamics, where the former two oscillate at the plunging frequency and the latter oscillates at twice the plunging frequency. We demonstrate that an induced phase difference between plunging and passive pitching is responsible for a higher thrust coefficient. This phase difference is also shown to be dependent on aerodynamics to inertia and natural to plunging frequency ratios. For inertia dominated flows, pitching and plunging always remain in phase. As the aerodynamics dominates, a large phase difference is induced which is accountable for a large passive deformation and higher thrust.

¹ Authors acknowledge the financial support received from the Aeronautics Research and Development Board (ARDB) under SIGMA project No. 1705 and thank the IIT Delhi HPC facility for computational resources.

9:05AM M21.00006 The role of resonance in propulsion of an elastic pitching wing with or without inertia¹, YANG ZHANG, Vanderbilt University, CHUNHUA ZHOU, Nanjing University of Aeronautics and Astronautics, HAOXIANG LUO, Vanderbilt University, LUO TEAM, ZHOU TEAM — Flapping wings of insects and undulating fins of fish both experience significant elastic deformations during propulsion, and it has been shown that in both cases, the deformations are beneficial to force enhancement and power efficiency. In fish swimming, the inertia of the fin structure is negligible and the hydrodynamic force is solely responsible for the deformation. However, in insect flight, both the wing inertia and aerodynamic force can be important factors leading to wing deformation. This difference raises the question about the role of the system (fluid-structure) resonance in the performance of propulsion. In this study, we use a 2D pitching foil as a model wing and vary its bending rigidity, pitching frequency, and mass ratio to investigate the fluid-structure interaction near resonance. The results show that at low mass ratios, i.e., a scenario of swimming, the system resonance greatly enhances thrust production and power efficiency, which is consistent with previous experimental results. However, at high mass ratios, i.e., a scenario of flying, the system resonance leads to overly large deformation that actually does not bring benefit any more. This conclusion thus suggests that resonance plays different roles in flying and in swimming.

¹Supported by the NNSF of China and the NSF of US.

9:18AM M21.00007 Bristles reduce force required to fling wings apart in small insects., SHANNON JONES, YOUNG YUN, TYSON HEDRICK, BOYCE GRIFFITH, LAURA MILLER, University of North Carolina at Chapel Hill — The smallest flying insects commonly possess wings with long bristles. Little quantitative information is available on the morphology of these bristles, and the functional importance of these bristles remains a mystery. In this study, we used the immersed boundary method to determine via numerical simulation if bristled wings reduced the force required to fling the wings apart during "clap and fling". The challenge of studying the fluid dynamics of bristles was in resolving the fluid flow between the bristles. The effects of Reynolds number, angle of attack, bristle spacing, and wing-wing interactions were investigated. We found that a bristled wing experiences less force than a solid wing, however bristled wings may act more like solid wings at lower angles of attack than they do at higher angles of attack. In wing-wing interactions, bristled wings significantly decrease the drag required to fling two wings apart compared with solid wings, especially at lower Reynolds numbers. These results support the idea that bristles may offer an aerodynamic benefit during clap and fling by reducing the force required to fling the wings apart in tiny insects.

9:31AM M21.00008 Flapping flight using bristled wings: effects of varying gap to diameter ratios, VISHWA TEJA KASOJU, ARVIND SANTHANAKRISHNAN, Oklahoma State Univ — The smallest flying insects with body lengths under 1 mm, such as thrips, show a preferential adaptation for fringed or bristled wings. In addition, these tiny insects have been observed to use wing-wing interaction via the clap and fling mechanism. We have previously shown that the use of bristled wings can lower forces required to clap the wings together and fling them apart. Tremendous variation is observed in bristled wing design among tiny insects. In this study, we examine the role of ratio of bristle gap to diameter (G/D) on force generation and flow structures at Reynolds numbers on the order of 10. A dynamically scaled robotic model was developed for this study, in which physical models of bristled wings were programmed to execute a 2D clap and fling kinematics. Bristled wing models with G/D ranging from 5 through 17 were examined. Lift and drag forces were measured using strain gages and phase-locked particle image velocimetry was used to visualize flow structures generated from the flapping motion. The results showed reductions in the size of the leading edge vortex and drag force with increasing G/D. The effects of increasing G/D on leakiness through the bristles will be presented.

9:44AM M21.00009 Hovering and targeting flight simulations of a dragonfly-like flapping wing-body model by IB-LBM¹, TAKAJI INAMURO, KENSUKE HIROHASHI, Dept. Aeronautics and Astronautics, Kyoto University — Hovering and targeting flights of the dragonfly-like flapping wing-body model are numerically investigated by using the immersed boundary-lattice Boltzmann method (IB-LBM). The governing parameters of the problem are the Reynolds number Re , the Froude number Fr , and the non-dimensional mass m . We set the parameters at $Re = 200$, $Fr = 15$, and $m = 51$. First, we simulate free flights of the model for various values of the phase difference angle ϕ between the forewing and the hindwing motions and for various values of the stroke angle β between the stroke plane and the horizontal plane. We find that the vertical motion of the model depends on the phase difference angle ϕ , and the horizontal motion of the model depends on the stroke angle β . Secondly, using the above results we try to simulate the hovering flight by dynamically changing the phase difference angle ϕ and the stroke angle β . The hovering flight can be successfully simulated by a simple proportional controller of the phase difference angle and the stroke angle. Finally, we simulate targeting flight by dynamically changing the stroke angle β .

¹The authors acknowledge the HPCI System Research Project (hp140025 and hp150087) and the Grants-in-Aid Scientific Research (No. 26420108) from JSPS.

Tuesday, November 22, 2016 8:00AM - 10:10AM –
Session M22 Microscale Flows: Interfaces and Wetting E141/142 - Ying Sun, Drexel University

8:00AM M22.00001 Effect of Microstructure and Wettability on Liquid Delivery in Structured Surfaces, DIYUAN ZONG, Tsinghua University, ARIF ROKONI, Drexel University, YUANYUAN DUAN, Tsinghua University, YING SUN, Drexel University — Liquid delivery rate in structured surfaces contributes greatly to the enhancement in critical heat flux during boiling and thin film evaporation. In the present study, the effect of microstructure and wettability on the liquid delivery rate has been investigated using molecular dynamics (MD) simulations for droplet spreading on nanostructured surfaces and liquid infiltration in nanopores. The wicked volume flux, a measure of the liquid delivery rate during droplet spreading, is found to increase with Wenzel roughness ratio regardless of the microstructure and this finding is supported by existing experiments. In addition, the effects of long-range intermolecular interactions and external electric field on liquid delivery rate are investigated. Long-range intermolecular interaction has been found to have negligible effect on liquid delivery rate. Both the wicking length in droplet spreading and the infiltration length of liquid infiltration in nanopores scale with $t^{(1/2)}$, consistent with the classic theory. The effects of the nanostructure depth, width-to-spacing ratio, wettability, and external electric field on the wicking coefficient have been studied and the results can be used to guide structured surface designs for enhanced liquid delivery.

8:13AM M22.00002 The failure of a superhydrophobic surface under external flow¹, YING LIU, MATTHEW FU, MARCUS HULTMARK, ALEXANDER SMITS, HOWARD STONE, Princeton University, MURI SLIPS TEAM — The advantages of superhydrophobic surfaces (SHS), such as ultra water-repellency, drag reduction and enhanced heat transfer, rely on the existence of the air trapped inside the surface geometries. Thus, it is important to study the failure of SHS, i.e., how the air-filled cavities are filled with water. Most of the previous work on this topic focuses on static pressure-driven failure. Here, we study experimentally the dynamic failure of SHS under an external flow. Conditions leading to failure are identified. The effects of both the pressure and the shear from the external flow on the failure of SHS are discussed.

¹This work is supported under Office of Naval Research (ONR) Multidisciplinary University Research Initiative (MURI) Grants N00014-12-1-0875 and N00014-12-1-0962 (Program Manager Dr. Ki-Han Kim).

8:26AM M22.00003 Absence of molecular slip on ultraclean and SAM-coated surfaces, JUSTIN PYE, CLAY WOOD, JUSTIN BURTON, Emory University — The liquid/solid boundary condition is a complex problem that is becoming increasingly important for the development of nanoscale fluidic devices. Many groups have now measured slip near an interface at nanoscale dimensions using a variety of experimental techniques. In simple systems, large slip lengths are generally measured for non-wetting liquid/solid combinations, but many conflicting measurements and interpretations remain. We have developed a novel pseudo-differential technique using a quartz crystal microbalance (QCM) to measure slip lengths on various surfaces. A drop of one liquid is grown on the QCM in the presence of a second, ambient liquid. We have isolated any anomalous boundary effects such as interfacial slip by choosing two liquids which have identical bulk effects on the QCM frequency and dissipation in the presence of no-slip. Slip lengths are –less than 2 nm– for water (relative to undecane) on all surfaces measured, including plasma cleaned gold, SiO₂, and two different self assembled monolayers (SAMs), regardless of contact angle. We also find that surface cleanliness is crucial to accurately measure slip lengths. Additionally, clean glass substrates appear to have a significant adsorbed water layer and SAM surfaces show excess dissipation, possibly associated with contact line motion. In addition to investigating other liquid pairs, future work will include extending this technique to surfaces with independently controllable chemistry and roughness, both of which are known to strongly affect interfacial hydrodynamics.

8:39AM M22.00004 Apparent and Actual Dynamic Contact Angles in Confined Two-Phase Flows¹, TAKESHI OMORI, TAKEO KAJISHIMA, Osaka Univ — To accurately predict the fluid flow with moving contact lines, it has a crucial importance to use a model for the dynamic contact angle which gives contact angles on the length scale corresponding to the spacial resolution of the fluid solver. The angle which a moving fluid interface forms to a solid surface deviates from an actual (macroscopic) dynamic contact angle depending on the distance from the contact line and should be called an apparent (macroscopic) dynamic contact angle. They were, however, often undistinguished especially in the experimental works, on which a number of empirical correlations between a contact angle and a contact line velocity have been proposed. The present study is the first attempt to measure both apparent and actual contact angles from the identical data sets to discuss the difference and the relationship between these two contact angles of difference length scales. The study is conducted by means of numerical simulation, solving the Navier-Stokes equation and the Cahn-Hilliard equation under the generalized Navier boundary condition for the immiscible two-phase flow in channels. The present study also illustrates how the system size and the physical properties of the adjoining fluid affect the apparent and the actual dynamic contact angles.

¹JSPS KAKENHI Grant No. 15K17974

8:52AM M22.00005 From catastrophic acceleration to deceleration of liquid plugs in prewetted capillary tubes.¹, JUAN MAGNIEZ, MICHAEL BAUDOIN, FARZAM ZOUESHTIAGH, IEMN, International Laboratory, UMR CNRS 8520, Universit de Lille, LEMAC/LICS TEAM — Liquid/gas flows in capillaries are involved in a multitude of systems including flow in porous media, petroleum extraction, imbibition of paper or flows in pulmonary airways in pathological conditions. Liquid plugs, which compose the biphasic flows, can have a dramatic impact on patients with pulmonary obstructive diseases, since they considerably alter the circulation of air in the airways and thus can lead to severe breathing difficulties. Here, the dynamics of liquid plugs in prewetted capillary tube is investigated experimentally and theoretically, with a particular emphasis on the role of the prewetting films and of the driving condition (constant flow rate, constant pressure). For both driving conditions, the plugs can either experience a continuous increase or decrease of their size. While this phenomenon is regular in the case of imposed flow rate, a constant pressure head can lead to a catastrophic acceleration of the plug and eventually its rupture or a dramatic increase of the plug size. A theoretical model is proposed to explain the transition between these two regimes. These results give a new insight on the critical pressure required for airways obstruction and reopening.

¹IEMN, International Laboratory LEMAC/LICS, UMR CNRS 8520, University of Lille

9:05AM M22.00006 Liquid/liquid/solid contact angles, MARINE BOROCCO, PMMH, ESPCI, Paris, and LadHyX, Ecole Polytechnique, Palaiseau, France, CHARLOTTE PELLET, JEAN-REN AUTHELIN, Sanofi Pharmaceutical Engineering, Vitry-sur-Seine, France, CHRISTOPHE CLANET, DAVID QUR, PMMH, ESPCI, Paris, and LadHyX, Ecole Polytechnique, Palaiseau, France, COMPAGNIE DES INTERFACES TEAM — Many studies have investigated solid/liquid/air interfaces and their corresponding wetting properties. We discuss what happens in less-studied liquid/liquid/solid systems, and focus on questions of dynamical wetting in a tube, having in mind applications in detergency. We use a capillary tube filled with water and containing a slug of silicone oil (or vice-versa), and present a series of experiments to determine static and dynamic wetting properties corresponding to this situation. We also discuss interfacial aging of such systems.

9:18AM M22.00007 Dynamic wetting of a liquid film in a vertical hydrophobic tube

, FRANCK PIGEONNEAU, Surface du Verre et Interface - UMR 125 CNRS/saint-Gobain, PASCALINE HAYOUN, ETIENNE BARTHEL, FRANCOIS LEQUEUX, EMILIE VERNEUIL, ESPCI - Physico-chimie des Polymeres et Milieux Disperses, ALBAN LETAILLER, JEREMIE TEISSEIRE, Saint-Gobain Recherche, SAINT-GOBAIN RECHERCHE COLLABORATION, ESPCI - PHYSICO-CHIMIE DES POLYMERES ET MILIEUX DISPERSES COLLABORATION, SURFACE DU VERRE ET INTERFACES COLLABORATION — The drop of a liquid plug through a tube occurs for instance in vending machine. In such a system, the fouling is linked to the creation of the liquid film at the rear of the liquid plug. Consequently, the conditions leading to the film creation are important to know. We study numerically the dynamic wetting transition of a liquid plug undergoing gravity on hydrophobic surface in a vertical tube. Using a lubrication theory, the liquid film thickness obeys the mass conservation equation with a volume flow rate depending on the relative motion of the tube, capillary and gravity forces. An ad hoc friction at the triple line is used to take into account the wetting dynamics. The lubrication equation is solved using a finite difference technique in space and a time integrator for stiff system with an adaptive time step. The numerical results are compared to experimental data. The complex film morphology due to the transients and the critical slowing down at the dynamic transition are reproduced. However, several experimental features are not predicted numerically especially the width of the transition. Our preliminary calculations suggest that the dispersion relation of the liquid film mode can explain the discrepancy.

9:31AM M22.00008 ABSTRACT WITHDRAWN —**9:44AM M22.00009 Comparison of the Cahn-Hilliard-Navier-Stokes and Molecular Dynamics Approaches for the Simulation of Droplet Coalescence and Wetting Phenomena**

, URBAIN VAES, Department of Mathematics, Imperial College London, BENJAMIN AYMARD, SRIKANTH RAVIPATI, PETR YATSYSHIN, Complex Multiscale Systems Group, Department of Chemical Engineering, Imperial College London, AMPARO GALINDO, Centre for Process Systems Engineering, Department of Chemical Engineering, Imperial College London, SERAFIM KALLIADASIS, Complex Multiscale Systems Group, Department of Chemical Engineering, Imperial College London — Diffuse-interface/Cahn-Hilliard equations, coupled to Navier-Stokes (CHNS), have been used extensively over the last few years in fluid dynamics including interfacial phenomena in multiphase systems. Applications range from turbulent two-phase flows to rheological systems and microfluidic devices. But despite the considerable attention CHNS have received, little work has been undertaken to investigate the extent to which they agree with "first-principles" physical models such as those provided by molecular dynamics (MD). Here we compare MD simulations with solutions of the CHNS system obtained numerically using an efficient and systematic finite-element methodology we have developed recently. For this purpose, we consider two paradigmatic model systems: droplet coalescence and droplet motion on a substrate with varying wettability.

9:57AM M22.00010 Rayleigh-Plateau instability of slipping viscous filaments in v-shaped grooves

, MARTIN BRINKMANN, TAK SHING CHAN, RALF SEEMANN, Experimental Physics, Saarland University — Since the seminal works of Rayleigh and Plateau on the break-up of free-standing liquid jets, a large number of studies have addressed capillary instability of cylindrical interfaces in various settings. Here, we report the numerical results of a linear stability analysis of cylindrical liquid filament wetting v-shaped grooves employing a boundary element formalism. It is found that slip affects the wavelength λ^{\max} of the fastest growing mode whenever the transverse dimension W of the filaments is comparable, or smaller than the Navier slip-length B . The corresponding timescale of the decay, τ^{\max} , grows logarithmically with increasing B/W . In the opposite limit $B/W \ll 1$, however, λ^{\max} grows unboundedly with increasing B/W while τ^{\max} saturates to a finite lower bound, similar to the situation observed for free-standing viscous liquid cylinders in the absence of inertial effects. Long wavelength approximations of the flows for $B/W \ll W$ and $B/W \gg 1$ are in good agreement with the numerical results only for contact angles $0 < \theta - \psi \ll 1$ where the neutrally stable wavelength $\lambda^* < \lambda^{\max}$ is large compared to the transverse filament dimension W .

Tuesday, November 22, 2016 8:00AM - 9:57AM —

Session M25 Microscale Flows: Emulsions and Mixing E145 - Shelley Anna, Carnegie Mellon University

8:00AM M25.00001 The Role of Colloidal Interactions on the Formation of Particle Stabilized Capsules¹

, SHELLEY ANNA, CHARLES SHARKEY, ANTHONY KOTULA, Carnegie Mellon University — Nanoparticles can adsorb to fluid-fluid interfaces to make stable foams and emulsions. Surfactants adsorbed to the nanoparticle surface modulate both particle wettability and interparticle interactions, altering the nanoparticle adsorption. We have shown that bubbles generated in a nanoparticle-surfactant mixture collect particles as they travel through a long microchannel. The particle stabilized region of the bubble grows in a manner consistent with convection and diffusion of particles in the fluid surrounding the bubble. If the bubble residence time is long enough compared with the adsorption timescales, a stable, non-spherical, gas-filled capsule emerges from the microchannel and retains its shape for tens of hours. We find that the nanoparticle-surfactant mixture composition can be used to tune the degree of capsule stabilization. Greater stabilization occurs with larger surfactant concentrations for a fixed nanoparticle volume fraction. These observations can be rationalized in terms of the particle wettability and electrostatic interactions as well as interfacial elasticity and bulk nanoparticle transport and adsorption.

¹National Science Foundation Grant No. 1511016

8:13AM M25.00002 Compact and controlled microfluidic mixing and biological particle capture¹

, MATTHEW BALLARD, DREW OWEN, ZACHARY GRANT MILLS, PETER J. HESKETH, ALEXANDER ALEXEEV, Georgia Institute of Technology — We use three-dimensional simulations and experiments to develop a multifunctional microfluidic device that performs rapid and controllable microfluidic mixing and specific particle capture. Our device uses a compact microfluidic channel decorated with magnetic features. A rotating magnetic field precisely controls individual magnetic microbeads orbiting around the features, enabling effective continuous-flow mixing of fluid streams over a compact mixing region. We use computer simulations to elucidate the underlying physical mechanisms that lead to effective mixing and compare them with experimental mixing results. We study the effect of various system parameters on microfluidic mixing to design an efficient micromixer. We also experimentally and numerically demonstrate that orbiting microbeads can effectively capture particles transported by the fluid, which has major implications in pre-concentration and detection of biological particles including various cells and bacteria, with applications in areas such as point-of-care diagnostics, biohazard detection, and food safety.

¹Support from NSF and USDA is gratefully acknowledged.

8:26AM M25.00003 Investigating droplet internal flow in concentrated emulsion when flowing in microchannel using micro-PIV, CHIA MIN LEONG, Mechanical Engineering, Rensselaer Polytechnic Institute, YA GAI, Aeronautics and Astronautics, Stanford University, SINDY K. Y. TANG, Mechanical Engineering, Stanford University — Droplet microfluidics has enabled a wide variety of high throughput applications through the use of monodisperse droplets. Previous fluid studies of droplet microfluidics have focused on single drops or emulsions at low volume fractions. The study of concentrated emulsions at high volume fractions is important for increasing the throughput, but the fluid dynamics of such emulsions in confined channels is not well understood. Here we describe two-dimensional, mid-height measurements of the flow inside individual drops within a concentrated emulsion using micro-PIV. The emulsion has 85% volume fraction and flows as a monolayer in a straight microfluidic channel. The effects of confinement and viscosity ratio on the internal flow patterns inside the drops were studied. The results show rotational structures inside the drops always exist, and are independent of viscosity ratio for the conditions tested. The structures depend on droplet mobility which in turn, depends on the confinement of the emulsion and the location of the drops in the channel. To our best knowledge, no work has probed the flow field inside droplets of concentrated emulsions at high volume fractions in confined channels. Current work is in progress to measure the three-dimensional flow field in such system.

8:39AM M25.00004 Periodic dislocation dynamics in two-dimensional concentrated emulsion flowing in a tapered microchannel, YA GAI, Aeronautics and Astronautics, Stanford University, CHIA MIN LEONG, Mechanical Aerospace and Nuclear Engineering, Rensselaer Polytechnic Institute, WEI CAI, SINDY K. Y. TANG, Mechanical Engineering, Stanford University — Here we report a surprising order in concentrated emulsion when flowing as a monolayer in a tapered microfluidic channel. The flow of droplets in micro-channels can be non-trivial, and may lead to unexpected phenomena such as long-period oscillations and chaos. Previously, there have been studies on concentrated emulsions in straight channels and channels with bends. The dynamics of how drops flow and rearrange in a tapered geometry has not yet been characterized. At sufficiently slow flow rates, the drops arrange into a hexagonal lattice. At a given x-position, the time-averaged droplet velocities are uniform. The instantaneous drop velocities, however, reveal a different, wave-like pattern. Within the rearrangement zone where the number of rows of drops decreases from N to N-1, there is always a drop moved faster than the others. Close examination reveals the anomalous velocity profile arises from a series of dislocations that are both spatial and temporal periodic. To our knowledge, such reproducible dislocation motion has not been reported before. Our results are useful in novel flow control and mixing strategies in droplet microfluidics as well as modeling crystal plasticity in low-dimensional nanomaterials.

8:52AM M25.00005 Phonons in active microfluidic crystals, ALAN CHENG HOU TSANG, Stanford University, EVA KANSO, University of Southern California — One-dimensional crystals of driven particles confined in quasi two-dimensional microfluidic channels have been shown to exhibit propagating sound waves in the form of ‘phonons’, including both transverse and longitudinal normal modes. Here, we focus on one-dimensional crystals of motile particles in uniform external flows. We study the propagation of phonons in the context of an idealized model that accounts for hydrodynamic interactions among the motile particles. We obtain a closed-form analytical expression for the dispersion relation of the phonons. In the moving frame of reference of the crystals, the traveling directions of the phonons depend on the intensity of the external flow, and are exactly opposite for the transverse and longitudinal modes. We further investigate the stability of the phonons and show that the longitudinal mode is linearly stable, whereas the transverse mode is subject to an instability arising from the activity and orientation dynamics of the motile particles. These findings are important for understanding the propagation of disturbances and instabilities in confined motile particles, and could generate practical insights into the transport of motile cells in microfluidic devices.

9:05AM M25.00006 Microfluidic step-emulsification in a cylindrical geometry¹, INDRA-JIT CHAKRABORTY, ALEXANDER M LESHANSKY, Department of Chemical Engineering, Technion-IIT — The model microfluidic device for high-throughput droplet generation in a confined cylindrical geometry is investigated numerically. The device comprises of core-annular pressure-driven flow of two immiscible viscous liquids through a cylindrical capillary connected co-axially to a tube of a larger diameter through a sudden expansion, mimicking the microfluidic step-emulsifier (1). To study this problem, the numerical simulations of axisymmetric Navier-Stokes equations have been carried out using an interface capturing procedure based on coupled level set and volume-of-fluid (CLSVOF) methods. The accuracy of the numerical method was favorably tested vs. the predictions of the linear stability analysis of core-annular two-phase flow in a cylindrical capillary. Three distinct flow regimes can be identified: the dripping (D) instability near the entrance to the capillary, the step-(S) and the balloon- (B) emulsification at the step-like expansion. Based on the simulation results we present the phase diagram quantifying transitions between various regimes in plane of the capillary number and the flow-rate ratio. (1) Z. Li et al., Lab on a Chip 15, 1023 (2015).

¹MICROFLUSA EU H2020 project

9:18AM M25.00007 A Novel Miniaturized Mixer Based on a Wankel Geometry, PANKAJ KUMAR, STEPHEN WAN, IHPC, A-Star — Mixing in microfluidic systems is a challenge since the flow regime encountered in these systems is typically very low Reynolds number laminar flow, in which viscous forces dominate inertial forces, which precludes efficient turbulence-based mixing. Mixing based purely on diffusion is also not a practical alternative due to the long times required to achieve a sufficient level of mixing. The present study presents a pump based on Wankel geometry as a mixer for efficient mixing in a microfluidic system. Then, a novel modification to the internal geometry of the Wankel-pump-mixer is analyzed and is shown to enable robust mixing without the introduction of an additional system component and hence without the expense of undesirable dead volume. The Lagrangian Coherent Structures (LCS) calculated from the Finite-Time Lyapunov Exponent (FTLE) field with a mixing measure is used to quantify the mixing.

9:31AM M25.00008 Design and optimization of anode flow field of a large proton exchange membrane fuel cell for high hydrogen utilization¹, SERHAT YESILYURT, OMID RIZWANDI, Sabanci University, Istanbul — We developed a CFD model of the anode flow field of a large proton exchange membrane fuel cell that operates under the ultra-low stoichiometric (ULS) flow conditions which intend to improve the disadvantages of the dead-ended operation such as severe voltage transient and carbon corrosion. Very small exit velocity must be high enough to remove accumulated nitrogen, and must be low enough to retain hydrogen in the active area. Stokes equations are used to model the flow distribution in the flow field, Maxwell-Stefan equations are used to model the transport of the species, and a voltage model is developed to model the reactions kinetics. Uniformity of the distribution of hydrogen concentration is quantified as the normalized area of the region in which the hydrogen mole fraction remains above a certain level, such as 0.9. Geometry of the anode flow field is modified to obtain optimal configuration; the number of baffles at the inlet, width of the gaps between baffles, width of the side gaps, and length of the central baffle are used as design variables. In the final design, the hydrogen-depleted region is less than 0.2% and the hydrogen utilization is above 99%.

¹This work was supported by The Scientific and Technological Research Council of Turkey, TUBITAK-213M023

9:44AM M25.00009 Parametric study on phase separation of binary mixtures in a lid driven cavity: A DPD study, HARINADHA GIDITURI, VIJAY ANAND, MAHESH PANCHAGNULA, SRIKANTH VEDANTAM, Indian Inst of Tech-Madras — We investigate the phase separation behavior of binary mixtures in two dimensional periodic and lid driven cavity domains using dissipative particle dynamics (DPD). The effect of DPD parameters like repulsion coefficient, dissipative coefficient, cut-off radius, and weight function exponent on domain size growth has been studied. The phase separation is delayed for low values of repulsion coefficient. Under these conditions, a few clusters of the dispersed phase are distributed in a continuous phase. This is because of weak inter-particle repulsion. As we increase the repulsion coefficient value, this behavior disappears. The domain growth rate is also observed to increase with an increase in the value of the dissipation coefficient as well as cut-off radius. Finally, the dynamics of phase separation in the lid driven cavity problem are significantly different when compared to that in the periodic domain, due to the formation of a stable vortex in the cavity. The vortex results in a dynamic equilibrium between clustering and separation. The distribution of cluster sizes is studied as a function of the driven cavity parameters.

Tuesday, November 22, 2016 8:00AM - 10:10AM – Session M26 Suspensions: General E146 - Enrique Rame, USRA

8:00AM M26.00001 Electric field driven mesoscale phase transition in polarized colloids.¹, BORIS KHUSID, EZINWA ELELE, QIAN LEI, New Jersey Inst of Technology — A mesoscale phase transition in a polarized suspension was reported by Kumar, Khusid, Acrivos, PRL95, 2005 and Agarwal, Yethiraj, PRL102, 2009. Following the application of a strong AC field, particles aggregated head-to-tail into chains that bridged the interelectrode gap and then formed a cellular pattern, in which large particle-free domains were enclosed by particle-rich thin walls. Cellular structures were not observed in numerous simulations of field induced phase transitions in a polarized suspension. A requirement for matching the particle and fluid densities to avoid particle settling limits terrestrial experiments to negatively polarized particles. We present data on the phase diagram and kinetics of the phase transition in a neutrally buoyant, negatively polarized suspension subjected to a combination of AC and DC. Surprisingly, a weak DC component drastically speeds up the formation of a cellular pattern but does not affect its key characteristic. However, the application of a strong DC field destroys the cellular pattern, but it restores as the DC field strength is reduced. We also discuss the design of experiments to study phase transitions in a suspension of positively polarized, non-buoyancy-matched particles in the International Space Station.

¹Supported by NASA's Physical Science Research Program, NNX13AQ53G.

8:13AM M26.00002 A hydrodynamic linear instability in a system of confined colloidal rollers¹, BLAISE DELMOTTE, ALEKSANDAR DONEV, Courant Institute of Mathematical Sciences, New York University, MICHELLE DRISCOLL, PAUL CHAIKIN, Department of Physics, New York University — In a typical flow instability, the fastest growing wavelength is selected by two or more competing stresses. In this talk I will discuss a very different kind of instability, controlled by a single geometric parameter. We study theoretically a new instability which has been observed experimentally and numerically: the fingering of a front of suspended microrollers near a floor. Our continuum model shows that this instability is linear and that the size scale selection arises only from hydrodynamic interactions between the particles and the wall, independently of the driving forces and viscosity. We believe that this instability mechanism is quite generic and selects the instability length scale in a number of suspension/colloid systems near a wall.

¹This work was supported primarily by the Materials Research Science and Engineering Center (MRSEC) program of the National Science Foundation under Award Number DMR-1420073.

8:26AM M26.00003 Stability and transition to chaos of regular capsule trains¹, SPENCER BRYNGELSON, JONATHAN FREUND, University of Illinois at Urbana-Champaign — Elastic capsules flowing in sufficiently narrow confines, such as red blood cells in capillaries, are well-known to line up in a single-file train. The stability of such a train in less confined environments, where this organization is not observed, is investigated in a model system that includes full coupling between the viscous flow and suspended elastic capsules. A rich set of linearly amplifying disturbances, including short- and long-time perturbations (non-modal and spectral, respectively) are identified and analyzed. Finite-amplitude transiently amplifying perturbations are shown to provide a mechanism that can bypass slower asymptotic modal linear growth and precipitate the onset of nonlinear dynamics. Direct numerical simulations are used to verify the linear analysis and track the subsequent transition of the regular capsule trains into an apparently chaotic flow.

¹This work was supported in part by the National Science Foundation under Grant No. CBET 13-36972.

8:39AM M26.00004 Testing the paradigms of the glass transition in colloids via dynamic simulation, JIALUN WANG, Cornell University, XIAOGUANG PENG, QI LI, GREGORY MCKENNA, Texas Tech University, ROSEANNA ZIA, Cornell University — Upon cooling, molecular glass-formers undergo a glass transition during which viscosity appears to diverge, and the material transitions from a liquid to an amorphous solid. However, the new state is not an equilibrium phase: material properties such as enthalpy continue to evolve in time. Rather, the material evolves toward an intransient state, as measured by the Kovacs signature experiments, e.g. the intrinsic isotherm, which reveals a paradoxical dependence of transition time on quench depth, and suggests that whether the glass transition occurs at the beginning or end of this transition is an open question. Colloidal glass formers provide a natural way to model such behavior, owing to the disparity in time scales that allow tracking of particle dynamics. We interrogate these ideas via dynamic simulation of a hard-sphere colloidal glassy state induced by jumps in volume fraction. We explore three methods to model the jump: evaporation, aspiration, and particle-size jumps. During and following each jump, the positions, velocities, and particle-phase stress are tracked and utilized to characterize relaxation time scales and structural changes. Analogs for the intrinsic isotherms are developed. The results provide insight into the existence of an ideal glass transition.

8:52AM M26.00005 Dynamics of two balls in bounded shear flow of Oldroyd-B fluids¹, SHANG-HUAN CHIU, TSORNG-WHAY PAN, ROLAND GLOWINSKI, Department of Mathematics, University of Houston, Houston, TX — The motion of dilute sphere suspensions in bounded shear flow of Oldroyd-B fluids has been studied at zero Reynolds number. Up to the initial sphere displacement, binary encounters of spheres in bounded shear flow of Newtonian fluid are known to have either swapping or non-swapping trajectories at zero Reynolds number (Zurita-gotor et al., J. Fluid Mech. 592 (2007) 447-469). We have simulated the interaction of two spherical particles in Newtonian fluid and Oldroyd-B fluid, respectively, and compared the resulting motions of particles. The motions of two spheres in Newtonian fluid are consistent with those in literature. In Oldroyd-B fluid, swapping trajectories can be obtained for the lower values of the relaxation time. For the non-swapping cases, two spheres do not return to their original transversed position once the encounter terminates, but being closer to the mid-plane between two walls, due to the effect of the elastic force. Two spheres may also attract each other first and then form rotating dipole in bounded shear flow, depending on the value of the relaxation time and initial sphere displacement.

¹NSF

9:05AM M26.00006 Anomalous low friction coefficient in shear thickening suspensions¹, CECILE CLAAUD, ANTOINE BERUT, BLOEN METZGER, YOEL FORTERRE, IUSTI CNRS Marseille France, GEP TEAM — We study the frictional behavior of classical and shear thickening suspensions under low confining pressure, by measuring the pile slope-angle in rotating drum flow experiments. We show that, at low rotation rates, the pile angle of the shear thickening suspension is about 8.5°, which is much lower than pile angles observed with classical suspensions in the same conditions ($\sim 25^\circ$). We then study the frictional behavior of silica powders in water, and show that we can switch from low to high pile angle by changing the salinity of the suspension. These results support a recent scenario for the shear-thickening transition in such non-Brownian systems, where inter particle repulsive forces suppress friction at low confining pressure.

¹ERC PlantMove

9:18AM M26.00007 Direct numerical simulation of particle alignment in viscoelastic fluids, MARTIEN HULSEN, NICK JAENSSON, PATRICK ANDERSON, Department of Mechanical Engineering, Eindhoven University of Technology, Eindhoven, The Netherlands — Rigid particles suspended in viscoelastic fluids under shear can align in string-like structures in flow direction. To unravel this phenomenon, we present 3D direct numerical simulations of the alignment of two and three rigid, non-Brownian particles in a shear flow of a viscoelastic fluid. The equations are solved on moving, boundary-fitted meshes, which are locally refined to accurately describe the polymer stresses around and in between the particles. A small minimal gap size between the particles is introduced. The Giesekus model is used and the effect of the Weissenberg number, shear thinning and solvent viscosity is investigated. Alignment of two and three particles is observed. Morphology plots have been created for various combinations of fluid parameters. Alignment is mainly governed by the value of the elasticity parameter S , defined as half of the ratio between the first normal stress difference and shear stress of the suspending fluid. Alignment appears to occur above a critical value of S , which decreases with increasing shear thinning. This result, together with simulations of a shear-thinning Carreau fluid, leads us to the conclusion that normal stress differences are essential for particle alignment to occur, but it is also strongly promoted by shear thinning.

9:31AM M26.00008 ABSTRACT WITHDRAWN —

9:44AM M26.00009 Evolution of Clouds of Migrating Micron-particles with Hydrodynamic and Electrostatic Interactions., SHUIQING LI, SHENG CHEN, Key Laboratory for Thermal Science and Power Engineering of Ministry of Education, Department of Thermal Engineering, Tsinghua University — The evolution of dilute clouds of charged micron-sized particles during the migration in an external electric field is numerically investigated. The hydrodynamic interaction is modeled employing the Oseen dynamics in the limit of small-but-finite particle Reynolds number. The effects of external field and inter-particle Coulomb repulsion are accounted by a pairwise summation. As a result, with a dominant external electrostatic force, the cloud is seen to flatten into a planar configuration with particle leakage in the tail and eventually breaks up into two small clouds. Decreasing the external force or increasing the pairwise Coulomb repulsion has a similar effect on the dynamics of the cloud, i.e., decreases the scaled migrating velocity of the cloud and makes the cloud steady in its spherical shape. While this behavior bears some similarity with the transition from the Stokes regime to the micro-scale inertia dominant regime, the underlying physical mechanisms differ. Finally, the variation of the typical aspect ratio of the cloud, as a function of a scaled radial velocity of particles, is used to quantify the effect of Coulomb repulsion on the stability of the shape of the cloud.

9:57AM M26.00010 Detecting plastic events in emulsions simulations¹, MATTEO LULLI, Univ of Rome Tor Vergata, MATTEO LULLI, MASSIMO BERNASCHI, MAURO SBRAGAGLIA TEAM — Emulsions are complex systems which are formed by a number of non-coalescing droplets dispersed in a solvent leading to non-trivial effects in the overall flowing dynamics. Such systems possess a yield stress below which an elastic response to an external forcing occurs, while above the yield stress the system flows as a non-Newtonian fluid, i.e. the stress is not proportional to the shear. In the solid-like regime the network of the droplets interfaces stores the energy coming from the work exerted by an external forcing, which can be used to move the droplets in a non-reversible way, i.e. causing plastic events. The Kinetic-Elasto-Plastic (KEP) theory is an effective theory describing some features of the flowing regime relating the rate of plastic events to a scalar field called fluidity $f = \frac{\dot{\gamma}}{\sigma}$, i.e. the inverse of an effective viscosity. Boundary conditions have a non-trivial role not captured by the KEP description. In this contribution we will compare numerical results against experiments concerning the Poiseuille flow of emulsions in microchannels with complex boundary geometries. Using an efficient computational tool we can show non-trivial results on plastic events for different realizations of the rough boundaries.

¹The research leading to these results has received funding from the European Research Council under the European Communities Seventh Framework Programme (FP7/2007- 2013)/ERC Grant Agreement no. [279004]

Tuesday, November 22, 2016 8:00AM - 10:10AM —
Session M27 CFD: Immersed Boundary Methods E147-148 - Amneet Pal Singh Bhalla, University of North Carolina, Chapel Hill

8:00AM M27.00001 Immersed boundary peridynamics (IB/PD) method to simulate aortic dissection¹, AMNEET PAL SINGH BHALLA, BOYCE GRIFFITH, University of North Carolina at Chapel Hill — Aortic dissection occurs when an intimal tear in the aortic wall propagates into the media to form a false lumen within the vessel wall. Rupture of the false lumen and collapse of the true lumen both carry a high risk of morbidity and mortality. Surgical treatment consists of either replacement of a portion of the aorta, or stent implantation to cover the affected segment. Both approaches carry significant risks: open surgical intervention is highly invasive, whereas stents can be challenging to implant and offer unclear long-term patient outcomes. It is also difficult to time optimally the intervention to ensure that the benefits of treatment outweigh its risks. In this work we develop innovative fluid-structure interaction (FSI) model combining elements from immersed boundary (IB) and peridynamics (PD) methods to simulate tears in membranes. The new approach is termed as IB/PD method. We use non-ordinary state based PD to represent material hyperelasticity. Several test problems are taken to validate peridynamics approach to model structural dynamics, with and without accounting for failure in the structures. FSI simulations using IB/PD method are compared with immersed finite element method (IB/FE) to validate the new hybrid approach.

¹NIH Award R01HL117163 NSF Award ACI 1450327

8:13AM M27.00002 A stencil penalty method for improving accuracy of constraint immersed boundary method, RAHUL BALE, AICS, RIKEN, NICLAS JANSSON, Department of High Performance Computing and Visualization, KTH , KEIJI ONISHI, MAKOTO TSUBOKURA, AICS, RIKEN, NEELESH PATANKAR, Department of Mechanical Engineering, Northwestern University — The constraint based immersed boundary (cIB) method is known to be accurate for low and moderate Reynolds number (Re) flows. At high Re, we found that cIB is not able produce accurate results. High Re flows typically result in large pressure gradient across fluid-IB interface. This is especially pronounced when the IB is an interface with "zero-thickness." There is also a jump in pressure which leads to incorrect evaluation of pressure gradients near the fluid-IB interface. This error leads to inaccuracies in the boundary layer around the IB and can also lead to leakage of flow across the interface. We propose a novel IB formulation with a modified pressure gradient operator that calculates one-sided gradients on either side of the interface. This removes spurious gradients in pressure across the interface. The pressure gradient operator is modified using a WENO based stencil penalization scheme.

8:26AM M27.00003 A high-order Immersed Boundary method for the simulation of polymeric flow., DAVID STEIN, Simons Foundation, BECCA THOMASES, ROBERT GUY, University of California, Davis — We present a robust, flexible, and high-order Immersed Boundary method for simulating fluid flow, including the Incompressible Navier-Stokes equations and certain models of viscoelastic flow, e.g. the Stokes-Oldroyd-B equations. 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. Low and zero Reynolds number problems are handled efficiently and accurately. We demonstrate pointwise convergence of the polymeric stress for flows in complex domains, in contrast to the standard Immersed Boundary method, which generates large errors in the polymeric stress near to the boundaries.

8:39AM M27.00004 An efficient strongly coupled immersed boundary method for deforming bodies¹, ANDRES GOZA, TIM COLONIUS, California Institute of Technology — Immersed boundary methods treat the fluid and immersed solid with separate domains. As a result, a nonlinear interface constraint must be satisfied when these methods are applied to flow-structure interaction problems. This typically results in a large nonlinear system of equations that is difficult to solve efficiently. Often, this system is solved with a block Gauss-Seidel procedure, which is easy to implement but can require many iterations to converge for small solid-to-fluid mass ratios. Alternatively, a Newton-Raphson procedure can be used to solve the nonlinear system. This typically leads to convergence in a small number of iterations for arbitrary mass ratios, but involves the use of large Jacobian matrices. We present an immersed boundary formulation that, like the Newton-Raphson approach, uses a linearization of the system to perform iterations. It therefore inherits the same favorable convergence behavior. However, we avoid large Jacobian matrices by using a block LU factorization of the linearized system. We derive our method for general deforming surfaces and perform verification on 2D test problems of flow past beams. These test problems involve large amplitude flapping and a wide range of mass ratios.

¹This work was partially supported by the Jet Propulsion Laboratory and Air Force Office of Scientific Research

8:52AM M27.00005 The parallelizationn of the immersed interface method for flow around moving rigid objects¹, SHENG XU, Southern Methodist University — To simulate flow around moving objects, the immersed interface method treats the objects as the fluid and recovers their effect on the surrounding flow by incorporating jump conditions across the surfaces of the objects into numerical schemes. In this talk, I will present some recent enhancement of the method toward its parallelization for flow around a large number of rigid objects of complex geometries in 3d. I will give an overview of the method, derive necessary jump conditions for objects represented by triangular meshes, and then discuss how to parallelize the method. Numerical examples will be shown to test the accuracy, efficiency and robustness of the method.

¹NSF DMS1320317

9:05AM M27.00006 Simulations of Compressible Viscous Flows and Wave Scattering Using the Immersed Boundary Method¹, WALTER ARIAS-RAMIREZ, UNICAMP-Univ de Campinas, BRITTON J. OLSON, Lawrence Livermore National Laboratory, LLNL, WILLIAM R. WOLF, UNICAMP-Univ de Campinas, UNIVERSITY OF CAMPINAS TEAM, LAWRENCE LIVERMORE NATIONAL LABORATORY TEAM — The immersed boundary method (IBM) in combination with a high-order finite difference compact formulation is used to study canonical test cases in fluid mechanics and acoustics, including viscous compressible flows, acoustic wave reflection and diffraction, and shock-wave reflections. In this study, two IB formulations are implemented: the continuous forcing and the discrete forcing approaches. Results obtained for the two methodologies are presented for 1-D problems involving acoustic and shock wave reflection, plane wave acoustic scattering along a cylinder and the viscous flow past a solid cylinder. Additionally, a grid convergence study is carried out for the simulations showing first-order convergence for the current implementation of the continuous forcing approach and second-order convergence for the discrete forcing approach.

¹The authors acknowledge the financial support received from So Paulo Research Foundation, under grants No. 2016/00904-5, No. 2014/10166-6, No. 2014/24043-3, No.2013/03413-4, and the computational resources provided by the Lawrence Livermore National Lab.

9:18AM M27.00007 An immersed boundary method for aeroacoustic flow using a high-order finite difference method, BRITTON OLSON, Lawrence Livermore Natl Lab — An immersed boundary method that achieves second order accuracy in space on acoustic reflection problems is introduced and tested on a number of aero-acoustic related problems. The method follows a continuous forcing approach and uses existing solver operators to smoothly extend the flow solution through the immersed boundary. Both no-slip and free-slip boundary conditions are demonstrated on complex geometries using a high-order finite difference code on a Cartesian grid. High Mach number test problems are also shown, demonstrating the method's robustness in the presence of shock waves.

9:31AM M27.00008 An immersed boundary method for non-uniform Cartesian grids, JUWON JANG, CHANGHOON LEE, Yonsei University — Many kinds of immersed boundary method have been developed, but most of them have been used in uniform grids with discrete Dirac delta functions. Therefore, the distribution of Lagrangian points over the immersed surface is usually made uniformly. However, when any immersed boundary method is to be applied to non-uniform grids, uniform distribution might not be optimum for good performance. Recently, Akiki and Balachandar (2016) proposed a method to distribute the Lagrangian points nonuniformly over the surface of a sphere near the wall, but it cannot not be extended to more general shape of immersed surface. We propose a method that is capable for properly distributing the Lagrangian points over any kind of surface by considering the size of nearby Eulerian grids. Present method first finds intersection points between immersed surface and nonuniform Cartesian grids. Then, the centroid of the intersection points is projected on the immersed surface to be designated by Lagrangian point. This procedure guarantees one Lagrangian point per the Eulerian grid cell. This method is validated for various problems such as flows around a settling sphere, a moving sphere in the near-wall region and a tilted ellipsoid near the wall.

9:44AM M27.00009 Efficient ghost cell reconstruction for embedded boundary methods¹, NARSIMHA RAPAKA, MOHAMAD AL-MAROUF, RAVI SAMTANEY, King Abdullah University of Science and Technology — A non-iterative linear reconstruction procedure for Cartesian grid embedded boundary methods is introduced. The method exploits the inherent geometrical advantage of the Cartesian grid and employs batch sorting of the ghost cells to eliminate the need for an iterative solution procedure. This reduces the computational cost of the reconstruction procedure significantly, especially for large scale problems in a parallel environment that have significant communication overhead, e.g., patch based adaptive mesh refinement (AMR) methods. In this approach, prior computation and storage of the weightage coefficients for the neighbour cells is not required which is particularly attractive for moving boundary problems and memory intensive stationary boundary problems. The method utilizes a compact and unique interpolation stencil but also provides second order spatial accuracy. It provides a single step/direct reconstruction for the ghost cells that enforces the boundary conditions on the embedded boundary. The method is extendable to higher order interpolations as well. Examples that demonstrate the advantages of the present approach are presented.

¹Supported by the KAUST Office of Competitive Research Funds under Award No. URF/1/1394-01.

9:57AM M27.00010 ABSTRACT WITHDRAWN —

Tuesday, November 22, 2016 8:00AM - 10:10AM —
Session M28 Particle-laden Flows: Particle-Turbulence Interaction I F149 - Evan Variano, University of California, Berkeley

8:00AM M28.00001 Rotations of long, inertialess rods in turbulence, NIMISH PUJARA, EVAN VARIANO, Univ of California - Berkeley, GREG VOTH, Wesleyan University — We present results on rotation of rods with lengths varying through the inertial range in turbulence. Rod motion is computed using one-way coupling in the Johns Hopkins University Turbulence Database of homogenous isotropic turbulence at $Re_\lambda = 433$. We consider zero-volume, zero-inertia rods, whose motion we model by advecting two tracer particles with constant distance between them. By making the tumbling rate dimensionless with a timescale that corresponds to turbulent motions of the same size as the rod, we show that such motions are responsible for the most of the variance in rod tumbling. Flatness factors for tumbling show that large deviations from the mean tumbling rate become less frequent as rod length increases, suggesting that longer rods are less responsive to intermittent events in turbulence. Finally, to investigate how such rods respond to turbulent flow forcing at their scale, we calculate the coarse-grained velocity gradient tensor by fitting to the velocity field sampled at discrete points within the sphere that circumscribes the rod. Results of instantaneous rod alignment with the vorticity and strain-rate eigenvectors of this tensor enable us to understand the preferential orientation of rods with respect to the flow field they are experiencing.

8:13AM M28.00002 A direct comparison of fully resolved and point-particle models in particle-laden turbulent flow¹, JEREMY HORWITZ, Stanford University, MOHAMMAD MEHRABADI, University of Illinois at Urbana-Champaign, SHANKAR SUBRAMANIAM, Iowa State University, ALI MANI, Stanford University — Point-particle methods have become a popular methodology to simulate viscous fluids laden with dispersed solid elements. Such methods may be contrasted with particle-resolved methods, whereby the boundary conditions between particles and fluid are treated exactly, while point-particle methods do not capture the boundary conditions exactly and couple the continuous and dispersed phase via point-forces. This allows point-particle methods to simulate particle-turbulence interaction at considerably lower resolution and computational cost than particle-resolved methods. However, lack of validation of point-particle methods begs the question of the predictive power of point-particle methods. In other words, can point-particle methods recover particle and fluid statistics compared with particle-resolved simulation of dynamically equivalent non-dimensional problems? We address this question in this work by examining decaying homogeneous isotropic turbulence laden with particles. For the same nominal conditions, we compare statistics predicted by a particle resolved method to those predicted by a point-particle method. We also examine the effect of the undisturbed velocity in the point-particle drag law by studying the same problem with a correction scheme.

¹Supported by DOE and NSF

8:26AM M28.00003 The effect of wall geometry in particle-laden turbulent flow, HOORA ABDEHKAKHA, GIANLUCA IACCARINO, Stanford University — Particle-laden turbulent flow plays a significant role in various industrial applications, as turbulence alters the exchange of momentum and energy between particles and fluid flow. In wall-bounded flows, inhomogeneity in turbulent properties is the primary cause of turbophoresis that leads the particles toward the walls. Conversely, shear-induced lift force on the particles can become important if large scale vortical structures are present. The objective of this study is to understand the effects of geometry on fluid flows and consequently on particles transport and concentration. Direct numerical simulations combined with point particle Lagrangian tracking are performed for several geometries such as a pipe, channel, square duct, and squiricle (rounded-corners duct). In non-circular ducts, anisotropic and inhomogeneous Reynolds stresses are the most influential phenomena that produce the secondary flows. It has been shown that these motions can have a significant impact on transporting momentum, vorticity, and energy from the core of the duct to the corners. The main focus of the present study is to explore the effects of near the wall structures and secondary flows on turbophoresis, lift, and particle concentration.

8:39AM M28.00004 Stochastic modeling of fluid-particle flows in homogeneous cluster-induced turbulence, ALESSIO INNOCENTI, SERGIO CHIBBARO, Univ Pierre et Marie Curie, RODNEY FOX, Iowa State University, MARIA VITTORIA SALVETTI, University of Pisa — Inertial particles in turbulent flows are characterized by preferential concentration and segregation and, at sufficient mass loading, dense clusters may spontaneously generate due to momentum coupling between the phases. These clusters in turn can generate and sustain turbulence in the fluid phase, which we refer to as cluster-induced turbulence (CIT). In the present work, we tackle the problem of homogeneous gravity driven CIT in the framework of a stochastic model, based on a Lagrangian formalism which includes naturally the Eulerian one. A rigorous formalism has been put forward focusing in particular on the terms responsible of the two-way coupling in the carrier phase, which is the key mechanism in this type of flow. Moreover, the decomposition of the particle-phase velocity into the spatially correlated and uncorrelated components has been used allowing to identify the contributions to the correlated fluctuating energy and to the granular temperature. Tests have been performed taking into account also the effects of collisions between particles. Results are compared against DNS, and they show a good accuracy in predicting first and second order moments of particle velocity and fluid velocity seen by particles.

8:52AM M28.00005 Modeling particle-laden turbulent flows with two-way coupling using a high-order kernel density function method¹, TIMOTHY SMITH, XIAOYI LU, University of Illinois at Urbana-Champaign, REETESH RANJAN, Georgia Institute of Technology, CARLOS PANTANO, University of Illinois at Urbana-Champaign — We describe a two-way coupled turbulent dispersed flow computational model using a high-order kernel density function (KDF) method. The carrier-phase solution is obtained using a high-order spatial and temporal incompressible Navier-Stokes solver while the KDF dispersed-phase solver uses the high-order Legendre WENO method. The computational approach is used to model carrier-phase turbulence modulation by the dispersed phase, and particle dispersion by turbulence as a function of momentum coupling strength (particle loading) and number of KDF basis functions. The use of several KDF's allows the model to capture statistical effects of particle trajectory crossing to high degree. Details of the numerical implementation and the coupling between the incompressible flow and dispersed-phase solvers will be discussed, and results at a range of Reynolds numbers will be presented.

¹This work was supported by the National Science Foundation under grant DMS-1318161

9:05AM M28.00006 Particle dynamics during the transition from isotropic to anisotropic turbulence, CHUNG-MIN LEE, California State University Long Beach, ARMANN GYLFASSON, Reykjavik University, FEDERICO TOSCHI, Eindhoven University of Technology — Turbulent fluctuations play an important role on the dynamics of particles in turbulence, enhancing their dispersion and mixing. In recent years the statistical properties of particles in several statistically stationary turbulent flows have been the subject of many numerical and experimental studies. In many natural and industrial environments, however, one deals with turbulence in a transient state. As a prototype system, we investigate the transition from an isotropic to an anisotropic flow, namely looking at the influence of a developing mean flow on the dynamics of particles. We simulate, via direct numerical simulation, stationary homogeneous and isotropic turbulence and then suddenly impose a mean shear or strain. This allows us to quantify the effects of the mean flow on particle dynamics in these transient periods. Preliminary results on single particle properties, such as velocities and accelerations will be reported.

9:18AM M28.00007 Modification of particle-laden horizontal channel turbulence, JUNGHOOON LEE, CHANGHOON LEE, Yonsei University — Modification of channel turbulence by small, heavy particles that settle towards the bottom wall under the influence of gravity, interacting with the turbulence is investigated using direct numerical simulations coupled with Lagrangian particle tracking. Particles' momentum transfer to the fluid is implemented via a point-force approximation, and particle-particle interactions are not taken into account in the simulation assuming a dilute suspension. Once a particle reaches the bottom wall, it is removed, and then a new particle is injected at a random location in the very vicinity of the top wall with the vertical terminal velocity and horizontal fluid velocities at the new position, with a focus on particles-turbulence interaction before their deposition. We compare our simulation with the available experimental data to validate the simulation condition used. We discuss modifications of turbulence statistics and coherent structures for various Stokes numbers and identify the role of gravity in the particles-turbulence interaction. Plausible physical mechanisms responsible for the modification behavior are also provided.

9:31AM M28.00008 Direct numerical simulation of particles in a turbulent channel flow, ANKIT TYAGI, VISHWANATHAN KUMARAN, Indian Institute of Science — Goswami and Kumaran(2009a,b,2011a) studied the effect of fluid turbulence on particle phase in DNS. However, their studies were restricted to one way coupling where the effect of particles on fluid turbulence was not incorporated. We have extended their work by formulating a reverse force treatment through multipole expansion for the particle disturbance to the fluid turbulence. Here, the fluid velocity, strain rate and rotation rate at the particle position are used, as a far field, to calculate the disturbance caused by the particle and relaxing the point particle approximation. The simulations are done at high Stokes number where the fluid velocity fluctuations are uncorrelated over time scales of the particle dynamics. The results indicate that the particle mean velocity and stress are reduced when reverse force is incorporated. Level of reduction increases with mass loading and Stokes number. The variance of particle distribution function is reduced due to reduction in the fluid turbulent intensities. The particle velocity, angular velocity distribution function and stresses are compared for simulations where only the reverse force is incorporated, and where the dipoles are also incorporated, to examine the effect of force dipoles on the fluid turbulence and the particle distributions.

9:44AM M28.00009 DNS-DEM of Suspended Sediment Particles in an Open Channel

Flow¹, PEDRAM PAKSERESHT, SOURABH APTE, Oregon State University, Corvallis, OR, JUSTIN FINN, NETL, Albany, OR — DNS with point-particle based discrete element model (DEM) is used to study particle-turbulence interactions in an open channel flow at Re_τ of 710, corresponding to the experimental observations of Righetti & Romano (JFM, 2004). Large particles of diameter 200 microns (10 in wall units) with volume loading on the order of 10^{-3} are simulated using four-way coupling with closure models for drag, added mass, lift, pressure, and inter-particle collision forces. The point-particle model is able to accurately capture the effect of particles on the fluid flow in the outer layer. However, the particle is significantly larger than the wall-normal grid in the near-wall region, but slightly smaller than the axial and longitudinal grid resolutions. The point-particle model fails to capture the interactions in the near-wall region. In order to improve the near-wall predictions, particles are represented by Lagrangian material points which are used to perform interpolations from the grid to the Lagrangian points and to distribute the two-way coupling force to the Eulerian grid. Predictions using this approach is compared with the experimental data to evaluate its effectiveness.

¹ NSF project #1133363, Sediment-Bed-Turbulence Coupling in Oscillatory Flows.

9:57AM M28.00010 Particle-turbulence-acoustic interactions in high-speed free-shear flows, GREGORY SHALLCROSS, Univ of Michigan - Ann Arbor, DAVID BUCHTA, Univ of Illinois - Urbana-Champaign, JESSE CAPECELATRO, Univ of Michigan - Ann Arbor — Experimental studies have shown that the injection of micro-water droplets in turbulent flows can be used to reduce the intensity of near-field pressure fluctuations. In this study, direct numerical simulation (DNS) is used to evaluate the effects of particle-turbulence-acoustic coupling for the first time. Simulations of temporally developing mixing layers are conducted for a range of Mach numbers and mass loadings. Once the turbulence reaches a self-similar state, the air-density shear layer is seeded with a random distribution of mono disperse water-density droplets. For $M=0.9$ to $M=1.75$, preliminary results show reductions in the near-field pressure fluctuations for moderate mass loadings, consistent with experimental studies under similar conditions. At high speed, the principle reduction of the normal velocity fluctuations, which increases with particle mass loading, appears to correlate to the reduction of the near-field radiated pressure fluctuations. These findings demonstrate that the DNS reproduces the observed particle-turbulence-acoustic phenomenology, and its complete space-time database can be used to further understand their interactions.

Tuesday, November 22, 2016 8:00AM - 10:10AM –

Session M29 CFD: Uncertainty Quantification and Error Estimation F150 - Gianluca Iaccarino, Stanford University

8:00AM M29.00001 Structural Uncertainties in RANS Models: Reynolds Stress Transport contra Eddy Viscosity Frameworks¹, AASHWIN MISHRA, WOUTER EDELING, GIANLUCA IACCARINO, Center for Turbulence Research, Stanford University — A vast majority of turbulent flow studies, both in academia and industry, utilize Reynolds Averaged Navier Stokes based models. There are different RANS modeling frameworks to select from, depending on their complexity and computational requirements, such as eddy viscosity based models, second moment closures, etc. While the relative strengths and weaknesses of each modeling paradigm (vis-a-vis their predictive fidelity, realizability, etc) are roughly established for disparate flows, there are no extant comparative estimates on the relative uncertainty in their predictions. In this investigation, we estimate the structural uncertainty inherent to different RANS modeling approaches for select internal flows. This involves comparisons between models conforming to the same framework, and, across different modeling frameworks. We establish, compare, analyze and explicate the model inadequacy for flows such as in parallel, curved, converging and diverging channels for different models. One of the novel facets of this study involves the estimation of the structural uncertainties of established Reynolds Stress Transport models, and, contrasting these against simpler eddy viscosity models.

¹This work was supported under the DARPA EQUiPS project(Technical Monitor: Fariba Fahroo).

8:13AM M29.00002 Error estimation and adaptivity for transport problems with uncertain parameters¹, ONKAR SAHNI, JASON LI, ASSAD OBERAI, MANE, RPI — Stochastic partial differential equations (PDEs) with uncertain parameters and source terms arise in many transport problems. In this study, we develop and apply an adaptive approach based on the variational multiscale (VMS) formulation for discretizing stochastic PDEs. In this approach we employ finite elements in the physical domain and generalize polynomial chaos based spectral basis in the stochastic domain. We demonstrate our approach on non-trivial transport problems where the uncertain parameters are such that the advective and diffusive regimes are spanned in the stochastic domain. We show that the proposed method is effective as a local error estimator in quantifying the element-wise error and in driving adaptivity in the physical and stochastic domains. We will also indicate how this approach may be extended to the Navier-Stokes equations.

¹NSF Award 1350454 (CAREER)

8:26AM M29.00003 The Contribution of Statistical Errors in DNS Data Quantified with RANS-DNS Simulations¹, SVETLANA V. POROSEVA, University of New Mexico, ELBERT JEYAPAL, None, SCOTT M. MURMAN, NASA Ames Research Center, JUAN D. COLMENARES F., University of New Mexico — In RANS-DNS simulations, the Reynolds-averaged Navier-Stokes (RANS) equations are solved, with all terms but molecular diffusion being represented by the data from direct numerical simulations (DNS). No turbulence modeling is involved in such simulations. Recently, we demonstrated the use of RANS-DNS simulations as a framework for uncertainty quantification in statistical data collected from DNS. In the current study, contribution of the statistical error in the DNS data uncertainty is investigated using RANS-DNS simulations. Simulations of the Reynolds stress transport were conducted in a planar fully-developed turbulent channel flow at $Re = 392$ (based on the friction velocity) using DNS data collected at seven averaging times. The open-source CFD software OpenFOAM was used in RANS simulations. Budgets for the Reynolds stresses were obtained from DNS performed using a pseudo-spectral (Fourier/Chebyshev-tau) method.

¹The material is in part based upon work supported by NASA under award NNX12AJ61A

8:39AM M29.00004 Uncertainty Quantification for atmospheric flows: natural terrain and urban area applications, CLARA GARCÍA-SÁNCHEZ, von Karman Institute for Fluid Dynamics / Antwerp University / Columbia University, CATHERINE GORLÉ, Civil & Environmental Engineering, Stanford University — Modeling Atmospheric Boundary Layer (ABL) flows is an important concern for a wide range of applications, including the assessment of air quality and wind energy resources. The complexity of these ABL flows, whether in urban areas or over natural terrain, still poses a challenge for Reynolds-averaged Navier-Stokes models. In the present research, the effect of uncertainties in the inflow boundary conditions on the prediction of the flow patterns is investigated, considering two test cases for which field measurements are available: the Askervein Hill experiment (natural terrain) and the Joint Urban 2003 campaign (urban environment). The uncertainty in the inflow boundary conditions is represented by three uncertain parameters, and a non-intrusive polynomial chaos method is used to propagate these uncertainties to the quantities of interest, namely the prediction of the velocity at the locations of the different measurement stations. The results highlight some differences between ABL flows over natural terrain and those in an urban environment, in particular regarding the influence of the different uncertain parameters on the prediction of the velocity field. The implications for evaluating the effect of inflow uncertainties in these different types of ABL flows will be discussed.

8:52AM M29.00005 Stochastic optimization algorithm for inverse modeling of air pollution, KYONGMIN YEO, YOUNGDEOK HWANG, XIAO LIU, JAYANT KALAGNANAM, IBM Research — A stochastic optimization algorithm to estimate a smooth source function from a limited number of observations is proposed in the context of air pollution, where the source-receptor relation is given by an advection-diffusion equation. First, a smooth source function is approximated by a set of Gaussian kernels on a rectangular mesh system. Then, the generalized polynomial chaos (gPC) expansion is used to represent the model uncertainty due to the choice of the mesh system. It is shown that the convolution of gPC basis and the Gaussian kernel provides hierarchical basis functions for a spectral function estimation. The spectral inverse model is formulated as a stochastic optimization problem. We propose a regularization strategy based on the hierarchical nature of the basis polynomials. It is shown that the spectral inverse model is capable of providing a good estimate of the source function even when the number of unknown parameters (m) is much larger than the number of data (n), $m/n \geq 50$.

9:05AM M29.00006 Assessment of accuracy of CFD simulations through quantification of a numerical dissipation rate¹, J.A. DOMARADZKI, G. SUN, X. XIANG, K.K. CHEN, University of Southern California — The accuracy of CFD simulations is typically assessed through a time consuming process of multiple runs and comparisons with available benchmark data. We propose that the accuracy can be assessed in the course of actual runs using a simpler method based on a numerical dissipation rate which is computed at each time step for arbitrary sub-domains using only information provided by the code in question (Schramm et al., 2015; Castiglioni and Domaradzki, 2015). Here, the method has been applied to analyze numerical simulation results obtained using OpenFOAM software for a flow around a sphere at Reynolds number of 1000. Different mesh resolutions were used in the simulations. For the coarsest mesh the ratio of the numerical dissipation to the viscous dissipation downstream of the sphere varies from 4.5% immediately behind the sphere to 22% further away. For the finest mesh this ratio varies from 0.4% behind the sphere to 6% further away. The large numerical dissipation in the former case is a direct indicator that the simulation results are inaccurate, e.g., the predicted Strouhal number is 16% lower than the benchmark. Low numerical dissipation in the latter case is an indicator of an acceptable accuracy, with the Strouhal number in the simulations matching the benchmark.

¹supported by NSF

9:18AM M29.00007 Predicting night-time natural ventilation in Stanford's Y2E2 building using an integral model in combination with a CFD model, GIACOMO LAMBERTI, CATHERINE GORLÉ, Stanford University — Natural ventilation can significantly reduce energy consumption in buildings, but the presence of uncertainty makes robust design a challenging task. We will discuss the prediction of the natural ventilation performance during a 4 hour night-flush in Stanfords Y2E2 building using a combination of two models with different levels of fidelity: an integral model that solves for the average air and thermal mass temperature and a CFD model, used to calculate discharge and heat transfer coefficients to update the integral model. Uncertainties are propagated using polynomial chaos expansion to compute the mean and 95% confidence intervals of the quantities of interest. Comparison with building measurements shows that, despite a slightly to fast cooling rate, the measured air temperature is inside the 95% confidence interval predicted by the integral model. The use of information from the CFD model in the integral model reduces the maximum standard deviation of the volume-averaged air temperature by 20% when compared to using literature-based estimates for these quantities. The heat transfer coefficient resulting from the CFD model was found to be within the literature-based interval initially assumed for the integral model, but the discharge coefficients were found to be different.

9:31AM M29.00008 Quantifying the Discrepancy in RANS Modeling of Reynolds Stress Eigenvectors System, JINLONG WU, Virginia Tech, RONEY THOMPSON, Federal University of Rio de Janeiro, JIANXUN WANG, Virginia Tech, LUIZ SAMPAIO, Stanford University, HENG XIAO, Virginia Tech — Reynolds-Averaged Navier-Stokes (RANS) equations are the dominant tool for engineering design and analysis applications involving wall bounded turbulent flows. However, the modeled Reynolds stress tensor is known to be a main source of uncertainty, comparing to other sources like geometry, boundary conditions, etc. Recently, several works have been conducted with the aim to quantify the uncertainty of RANS simulation by studying the discrepancy of anisotropy and turbulence kinetic energy of the Reynolds stress tensor with respect to a reference database obtained from DNS. On the other hand, the eigenvectors system of Reynolds stress tensor is less investigated. In this work, a general metric is proposed to visualize the discrepancy between two eigenvectors systems. More detailed metrics based on the Euler angle and the direction cosine are also proposed to quantify the discrepancy of eigenvectors systems. The results show that even a small discrepancy of the eigenvectors of the Reynolds stress can lead to a drastically different mean velocity field, demonstrating the importance of quantifying this kind of uncertainty/error. Furthermore, the Euler angle and the direction cosine are compared for the purpose of uncertainty quantification and machine learning, respectively.

9:44AM M29.00009 Modeling of stochastic dynamics of time-dependent flows under high-dimensional random forcing¹, HESSAM BABAEE, MIT, GEORGE KARNIADAKIS, Brown University — In this numerical study the effect of high-dimensional stochastic forcing in time-dependent flows is investigated. To efficiently quantify the evolution of stochasticity in such a system, the dynamically orthogonal method is used. In this methodology, the solution is approximated by a *generalized* Karhunen-Loeve (KL) expansion in the form of $\mathbf{u}(\mathbf{x}, t; \omega) = \bar{\mathbf{u}}(\mathbf{x}, t) + \sum_{i=1}^N \mathbf{y}_i(t; \omega) \mathbf{u}_i(\mathbf{x}, t)$, in which $\bar{\mathbf{u}}(\mathbf{x}, t)$ is the stochastic mean, the set of $\mathbf{u}_i(\mathbf{x}, t)$'s is a deterministic orthogonal basis and $\mathbf{y}_i(t; \omega)$'s are the stochastic coefficients. Explicit evolution equations for $\bar{\mathbf{u}}$, \mathbf{u}_i and \mathbf{y}_i are formulated. The elements of the basis $\mathbf{u}_i(\mathbf{x}, t)$'s remain orthogonal for all times and they evolve according to the system dynamics to capture the energetically dominant stochastic subspace. We consider two classical fluid dynamics problems: (1) flow over a cylinder, and (2) flow over an airfoil under up to one-hundred dimensional random forcing. We explore the interaction of intrinsic with extrinsic stochasticity in these flows.

¹DARPA N66001-15-2-4055, Office of Naval Research N00014-14-1-0166

9:57AM M29.00010 Parameter Estimation for a Turbulent Buoyant Jet Using Approximate Bayesian Computation, JASON D. CHRISTOPHER, NICHOLAS T. WIMER, TORREY R. S. HAYDEN, CAELAN LAPOINTE, IAN GROOMS, GREGORY B. RIEKER, PETER E. HAMLINGTON, University of Colorado, Boulder — Approximate Bayesian Computation (ABC) is a powerful tool that allows sparse experimental or other “truth” data to be used for the prediction of unknown model parameters in numerical simulations of real-world engineering systems. In this presentation, we introduce the ABC approach and then use ABC to predict unknown inflow conditions in simulations of a two-dimensional (2D) turbulent, high-temperature buoyant jet. For this test case, truth data are obtained from a simulation with known boundary conditions and problem parameters. Using spatially-sparse temperature statistics from the 2D buoyant jet truth simulation, we show that the ABC method provides accurate predictions of the true jet inflow temperature. The success of the ABC approach in the present test suggests that ABC is a useful and versatile tool for engineering fluid dynamics research.

Tuesday, November 22, 2016 8:00AM - 10:10AM – Session M30 Flow Instability: Rayleigh-Taylor F151 - Daniel Attinger, Iowa State University

8:00AM M30.00001 Self-similarity of a Rayleigh-Taylor mixing layer at low Atwood number with a multimode initial perturbation¹, BRANDON MORGAN, BRITTON OLSON, Lawrence Livermore National Laboratory, JUSTIN WHITE, JACOB MCFARLAND, Department of Mechanical and Aerospace Engineering, University of Missouri — High-fidelity large eddy simulation (LES) of a low-Atwood number ($A = 0.05$) Rayleigh-Taylor mixing layer is performed using the tenth-order compact difference code Miranda. An initial multimode perturbation spectrum is specified in Fourier space as a function of mesh resolution such that a database of results is obtained in which each successive level of increased grid resolution corresponds approximately to one additional doubling of the mixing layer width, or *generation*. The database is then analyzed to determine approximate requirements for self-similarity, and a new metric is proposed to quantify how far a given simulation is from the limit of self-similarity. It is determined that the present database reaches a high degree of self-similarity after approximately 4.5 generations. Finally, self-similar turbulence profiles from the LES database are compared with one-dimensional simulations using the $k-L-a$ and BHR-2 Reynolds-averaged Navier-Stokes (RANS) models. The $k-L-a$ model, which is calibrated to reproduce a quadratic turbulence kinetic energy profile for a self-similar mixing layer, is found to be in better agreement with the LES than BHR-2 results.

¹This work was preformed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract No. DE-AC52-07NA27344.

8:13AM M30.00002 Viscous effects on the Rayleigh-Taylor instability with background temperature gradient, DANIEL LIVESCU, SERGIY GERASHCHENKO, Los Alamos National Laboratory — The growth rate of the compressible Rayleigh-Taylor instability is studied in the presence of a background temperature gradient, Θ , using a normal mode analysis. The effect of Θ variation is examined for three interface types corresponding to combinations of the viscous properties of the fluids (inviscid-inviscid, viscous-viscous and viscous-inviscid) at different Atwood numbers, At, and, when at least one of the fluids' viscosity is non-zero, as a function of the Grashof number. Compared to the $\Theta = 0$ case, the role of $\Theta < 0$ (hotter light fluid) is destabilizing and becomes stabilizing when $\Theta > 0$ (colder light fluid). The most pronounced effect of $\Theta \neq 0$ is found at low At and/or at large perturbation wavelengths relative to the domain size for all interface types. The results are applied to two practical examples, using sets of parameters relevant to Inertial Confinement Fusion coasting stage and solar corona plumes. The role of viscosity on the growth rate reduction is discussed together with highlighting the range of wavenumbers most affected by viscosity. The viscous effects further increase in the presence of a background temperature gradient, when the viscosity is temperature dependent.

8:26AM M30.00003 Predictions and Measurements of Blood Backspatter from a Gunshot in Bloodstain Pattern Analysis¹, PATRICK COMISKEY, ALEXANDER YARIN, Univ of Illinois - Chicago, SUNGU KIM, DANIEL ATTINGER, Iowa State University — A theoretical model for predicting and interpreting blood spatter patterns resulting from a gunshot wound is proposed. The physical process generating a backward spatter of blood is linked to the Rayleigh-Taylor instability of blood accelerated toward the surrounding air allowing the determination of initial distribution of drop sizes and velocities. Then, the motion of many drops in air is considered with governing equations accounting for gravity and air drag. The model predicts the atomization process, the trajectories of the back spatter drops of blood from the wound to the ground, the impact angle and the impact Weber number on the ground, as well as the number of, distribution, and location of blood stains and their shapes and sizes. The drop cloud originating from a wound entrains a significant mass of air due to the action of viscous forces. As a result of this collective effect, air drag acting on individual drops in the cloud is significantly reduced and fully accounted for in the model. The results of the model are compared to experimental data on back spatter generated by a gunshot impacting a blood-impregnated sponge. The model proposed in this work is in reasonable agreement with the results from the experimental data.

¹Support of this work by the US National Institute of Justice (award NIJ 2014-DN-BX-K036) is greatly appreciated.

8:39AM M30.00004 Effect of noise on Rayleigh-Taylor mixing with space-dependent acceleration¹, ARUN PANDIAN, SNEZHANA ABARZHI, Carnegie Mellon University — We analyze, for the first time by our knowledge, the effect of noise on Rayleigh-Taylor (RT) mixing with space-dependent acceleration by applying the stochastic model. In these conditions, the RT mixing is a statistically unsteady process where the mean values of the flow quantities vary in space and time, and there are also the space and time dependent fluctuations around these mean values. The stochastic model is derived from the momentum model and is represented by a set of nonlinear differential equations with multiplicative noise. The models equations are solved theoretically and numerically. Investigating a broad range of values of acceleration, self-similar asymptotic solutions are found in the mixing regime. There are two types of mixing sub-regimes (acceleration-driven and dissipation-driven respectively), each of which has its own types of solutions and characteristic values with the latter saturating to a value on the order of one. It is also observed that the representation of the dynamics in an implicit form is noisier as compared to the case of an explicit time-dependent form.

¹The work is supported by the US National Science Foundation

8:52AM M30.00005 Highly symmetric interfacial coherent structures in Rayleigh Taylor instability with time-dependent acceleration¹, AKLANT K. BHOWMICK, SNEZHANA ABARZHI, Carnegie Mellon University — Rayleigh Taylor instability in a power-law time dependent acceleration field is investigated theoretically for a flow with the symmetry group p6mm (hexagon) in the plane normal to acceleration. In the nonlinear regime, regular asymptotic solutions form a one-parameter family. The physically significant solution is identified with the one having the fastest growth and being stable (bubble tip velocity). Two distinct regimes are identified depending on the acceleration exponent. Particularly, the RM-type regime, where the dynamics is identical to conventional RM instability and is dominated by initial conditions, and the RT-type regime where the dynamics is dominated by the acceleration term. For the latter, the time dependence has profound effects on the dynamics. In the RT non-linear regime, the time dependence has no consequence on the morphology of the bubbles; the growth rate (bubble tip velocity) evolves as power law with the exponent set by the acceleration. The solutions for a one-parameter family, and are convergent with exponential decay of Fourier amplitudes. The solutions are stable at maximum tip velocity, whereas flat bubbles are unstable, and the growth/decay of perturbations is no longer purely exponential and depends on the acceleration exponent.

¹The work is supported by the US National Science Foundation

9:05AM M30.00006 Low-symmetric coherent structures and dimensional crossover in Rayleigh Taylor flows driven by time dependent accelerations¹, AKLANT BHOWMICK, SNEZHANA ABARZHI, Carnegie Mellon University — We investigate the nature of the dimensional crossover i.e. transition between the nearly isotropic 3D periodic flows with group p4mm (square) to highly anisotropic 2D periodic flows with group p2m1 in Rayleigh Taylor (RT) instability. Power law time dependence of the acceleration is considered with the emphasis on sub-regime, where the behavior is the RT type. We consider flow with group p2mm (rectangle) and obtain the 3D square and 2D limits with leading order rectangular corrections. Regular asymptotic solutions evolve as power law and form a two parameter family parametrized by the principal curvatures of the bubble. The bubbles with near circular contour separate the 2-dimensional solution space into two sub-regimes having distinct properties under the dimensional crossover. In one sub-regime, the elongated bubbles transform to 2D solutions, whereas in the other they flatten. 3D square bubbles are universally stable whereas 2D bubbles are unstable with respect to 3D modulations, implying that the dimensional crossover is discontinuous. We find that the time dependence affects the growth/decay of perturbations and has no consequence on the overall stability properties of the solution.

¹The work is supported by the US National Science Foundation

9:18AM M30.00007 Rayleigh-Taylor mixing with space-dependent acceleration¹, SNEZHANA ABARZHI, Carnegie Mellon University — We extend the momentum model to describe Rayleigh-Taylor (RT) mixing driven by a space-dependent acceleration. The acceleration is a power-law function of space coordinate, 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 sub-regimes of self-similar RT mixing: the acceleration-driven Rayleigh-Taylor-type mixing and dissipation-driven Richtmyer-Meshkov-type mixing 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 space-dependent acceleration from canonical cases of homogeneous turbulence as well as blast waves with first and second kind self-similarity.

¹The work is supported by the US National Science Foundation

9:31AM M30.00008 Long-wave analysis and control of the viscous Rayleigh-Taylor instability with electric fields, RADU CIMPEANU, Imperial College London, THOMAS ANDERSON, California Institute of Technology, PETER PETROPOULOS, New Jersey Institute of Technology, DEMETRIOS PAPAGEORGIOU, Imperial College London — We investigate the electrostatic stabilization of a viscous thin film wetting the underside of a solid surface in the presence of a horizontally acting electric field. The competition between gravity, surface tension and the nonlocal effect of the applied electric field is captured analytically in the form of a nonlinear evolution equation. A semi-spectral solution strategy is employed to resolve the dynamics of the resulting partial differential equation. Furthermore, we conduct direct numerical simulations (DNS) of the Navier-Stokes equations and assess the accuracy of the obtained solutions when varying the electric field strength from zero up to the point when complete stabilization at the target finite wavelengths occurs. We employ DNS to examine the limitations of the asymptotically derived behavior in the context of increasing liquid film heights, with agreement found to be excellent even beyond the target lengthscales. Regimes in which the thin film assumption is no longer valid and droplet pinch-off occurs are then analyzed. Finally, the asymptotic and computational approaches are used in conjunction to identify efficient active control mechanisms allowing the manipulation of the fluid interface in light of engineering applications at small scales, such as mixing.

9:44AM M30.00009 Experimental investigation of late time Rayleigh—Taylor mixing at high Atwood number, PRASOON SUCHANDRA, MARK MIKHAEL, DEVESH RANJAN, Georgia Inst of Tech — Dynamics of late time, high Reynolds number ($Re > 20000$) Rayleigh—Taylor (RT) mixing is studied using statistically steady experiments performed in a multi-layer gas tunnel. The density ratio of air and air-Helium mixture used in the present experiment results in an Atwood number ~ 0.73 . Three types of diagnostics — back-lit visualization, hot-wire anemometry and stereo particle image velocimetry (S-PIV) — are employed to obtain mixing width, velocity and density fields, with S-PIV employed for the first time for such experimental conditions. Velocity and density statistics, and their correlations ($u', v', w', \rho', \rho'v'$) are presented. Calculations of probability density functions (p.d.f.s) and energy spectra are made to provide further insight into the flow physics. Energy budget of the flow is also discussed. Reference: AKULA, B. & RANJAN, D. 2016 Dynamics of buoyancy-driven flows at moderately high Atwood numbers. *Journal of Fluid Mechanics* 795, 313–355.

9:57AM M30.00010 Effects of acceleration rate on Rayleigh-Taylor instability in elastic-plastic materials.¹, ARINDAM BANERJEE, RINOSH POLAVARAPU, Lehigh University — The effect of acceleration rate in the elastic-plastic transition stage of Rayleigh-Taylor instability in an accelerated non-Newtonian material is investigated experimentally using a rotating wheel experiment. A non-Newtonian material (mayonnaise) was accelerated at different rates by varying the angular acceleration of a rotating wheel and growth patterns of single mode perturbations with different combinations of amplitude and wavelength were analyzed. Experiments were run at two different acceleration rates to compare with experiments presented in prior years at APS DFD meetings and the peak amplitude responses are captured using a high-speed camera. Similar to the instability acceleration, the elastic-plastic transition acceleration is found to be increasing with increase in acceleration rate for a given amplitude and wavelength. The experimental results will be compared to various analytical strength models and prior experimental studies using Newtonian fluids.

¹ Authors acknowledge funding support from Los Alamos National Lab subcontract(370333) and DOE-SSAA Grant (DE-NA0001975).

Tuesday, November 22, 2016 8:00AM - 10:10AM –

Session M31 Experimental Techniques - General F152 - Shahram Pouya, Michigan State University

8:00AM M31.00001 Vorticity Measurement using LG Laser Beams with Orbital Angular Momentum¹, MANOOCHEHR KOOOCHEFAHANI, SHAHRAM POUYA, ALIREZA SAFARIPOUR, ANTON RYABTSEV, MARCOS DANTUS, Michigan State University — We present direct measurement of vorticity in a fluid flow based on angular velocity measurement of microparticles contained in the fluid. The method uses Laguerre-Gaussian (LG) laser beams that possess orbital angular momentum (OAM), a spatial (azimuthal) modulation of the beam phase front, and takes advantage of the rotational Doppler shift from microparticles intersecting the beam focus. Results are shown for the flow field of solid body rotation, where the flow vorticity is known precisely.

¹This work was supported by AFOSR award number FA9550-14-1-0312.

8:13AM M31.00002 Viscous Drag Measurements using Three Diagnostics¹, JONATHAN NAUGHTON, ERIC DEMILLARD, University of Wyoming, JAMES CRAFTON, JESSICA WEBB, ISSI — The measurement of viscous drag on surfaces is difficult due to the small forces involved compared to the pressure force. In addition, the different diagnostics used for measuring viscous drag often work only under a limited number of conditions. To address this issue, three different approaches for measuring viscous drag were evaluated: oil film interferometry, an integral momentum approach based on velocity profile measurements, and a novel drag balance. Oil film interferometry has been widely used for wall shear stress measurements, but only works on smooth surfaces with the appropriate optical properties. Integral momentum approaches should work on all surfaces, but are limited to two-dimensional flows and require the measurement of detailed velocity profiles. Force balances can also provide measurement on any surface, but are subject to misalignment and pressure gradient errors and provide no information about the flow over the surface. In this study, the three diagnostics were tested over a small region of a plate on which a two-dimensional turbulent boundary layer developed. Measurements were made on both smooth and rough surfaces. The strengths and weaknesses of each of the approaches and the benefits of the combined information they provide are discussed.

¹This work is supported by both the Air Force and NASA

8:26AM M31.00003 Inverse method for the instantaneous measure of wall shear rate magnitude and direction using electrodiffusion probes, MARC-ETIENNE LAMARCHE-GAGNON, JEROME VETEL, Polytechnique Montreal — Several methods can be used when one needs to measure wall shear stress in a fluid flow. Yet, it is known that a precise shear measurement is seldom met, mostly when both time and space resolutions are required. The electrodiffusion method lies on the mass transfer between a redox couple contained in an electrolyte and an electrode flush mounted to a wall. Similarly to the heat transfer measured by a hot wire anemometer, the mass transfer can be related to the fluid's wall shear rate. When coupled with a numerical post-treatment by the so-called *inverse method*, precise instantaneous wall shear rate measurements can be obtained. With further improvements, it has the potential to be effective in highly fluctuating three-dimensional flows.

We present developments of the inverse method to two-component shear rate measurements, that is shear magnitude and direction. This is achieved with the use of a three-segment electrodiffusion probe. Validation tests of the inverse method are performed in an oscillating plane Poiseuille flow at moderate pulse frequencies, which also includes reverse flow phases, and in the vicinity of a separation point where the wall shear stress experiences local inversion in a controlled separated flow.

8:39AM M31.00004 Using Magnetic Resonance Imaging (MRI) to Investigate Scalar Contaminant Dispersion in an Urban Environment, JOSEPH CYMERMANN, Stanford University — Research in the past few decades on the dispersion of a scalar contaminant through an urban environment, with testing occurring in multiple cities worldwide, has mostly relied on point measurement systems. These models, which are strongly affected by the orientation of the buildings and environmental conditions, obtain relatively few data points and fail to achieve a robust understanding of the complex flow fields from an experimental perspective. A time-averaged MRI-based experimental measurement of the complete three-dimensional flow field has analyzed an array of buildings at two orientations in fully turbulent flow. The Reynolds number and upstream development match a series of tests conducted by both the ARL Atmospheric Sciences Branch and the Los Alamos National Laboratory for comparison, with several million measurements of each of the three components of the mean velocity across the full field. The water-based experimental measurements allow for the assessment of the geometric dispersion of the streamlines and a comparison with several computational models as a means of validation.

8:52AM M31.00005 Characterization of a custom-built RF coil for a high-resolution phase-contrast magnetic resonance velocimeter¹, BYUNGKUEN YANG, Hanyang University, JEE-HYUN CHO, Korea Advanced Institute of Science and Technology, SIMON SONG, Hanyang University — For the use of clinical purpose magnetic resonance velocimeter (MRV) is a versatile flow visualization technique in that it allows opaque flow, complex geometry, no use of tracer particles and facile fast non-invasive measurements of 3 dimensional and 3 component velocity vectors. However, the spatial resolution of a commercial MR machine is lower than optics-based techniques like PIV. On the other hand, the use of MRV for clinical purposes like cardiovascular flow visualization requires accurate measurements or estimations on wall shear stress (WSS) with a high spatial resolution. We developed a custom-built solenoid RF coil for phase-contrast (PC) MRV to improve its resolution. We compared signal-to-noise ratio, WSS estimations, partial volume effects near wall between the custom RF coil and a commercial coil. Also, a Hagen-Poiseuille flow was analyzed with the custom RF coil.

¹This work was supported by the National Research Foundation of Korea(NRF) grant funded by the Korea government(MSIP) (No. 2016R1A2B3009541).

9:05AM M31.00006 Visualization of vacuum cleaner-induced flow in a carpet by using magnetic resonance velocimetry¹, JEESOO LEE, SIMON SONG, Hanyang Univ — Understanding characteristics of in-carpet flow induced by a vacuum cleaner nozzle is important to improve the design and performance of the cleaner nozzle. However, optical visualization techniques like PIV are limited to uncover the flow details because a carpet is opaque porous media. We have visualized a mean flow field in a cut-pile type carpet by magnetic resonance velocimetry. The flow was generated by a static vacuum cleaner nozzle, and the working fluid is a copper sulfate aqueous solution. Three dimensional, three component velocity vectors were obtained in a measurement domain of $336 \times 128 \times 14$ mm³ covering the entire nozzle span and a 7-mm thick carpet below the nozzle. The voxel size was $1 \times 1 \times 0.5$ (depthwise) mm³. Based on the visualization data, the permeability, the Forchheimer coefficient and pressure distribution were calculated for the carpet.

¹This work was supported by the National Research Foundation of Korea(NRF) grant funded by the Korea government(MSIP) (No. 2016R1A2B3009541).

9:18AM M31.00007 An experimental validation of the influence of flow profiles and stratified two-phase flow to Lorentz force velocimetry for weakly conducting fluids¹, ANDREAS WIEDERHOLD, RESCHAD EBERT, CHRISTIAN RESAGK, TU Ilmenau, RESEARCH TRAINING GROUP: "LORENTZ FORCE VELOCIMETRY AND LORENTZ FORCE EDDY CURRENT TESTING" TEAM — We report about the feasibility of Lorentz force velocimetry (LFV) for various flow profiles. LFV is a contactless non-invasive technique to measure flow velocity and has been developed in the last years in our institute. This method is advantageous if the fluid is hot, aggressive or opaque like glass melts or liquid metal flows. The conducted experiments shall prove an increased versatility for industrial applications of this method. For the force measurement we use an electromagnetic force compensation balance. As electrolyte salty water is used with an electrical conductivity in the range of 0.035 which corresponds to tap water up to 20 Sm^{-1} . Because the conductivity is six orders less than that of liquid metals, here the challenging bottleneck is the resolution of the measurement system. The results show only a slight influence in the force signal at symmetric and strongly asymmetric flow profiles. Furthermore we report about the application of LFV to stratified two-phase flows. We show that it is possible to detect interface instabilities, which is important for the dimensioning of liquid metal batteries.

¹Deutsche Forschungsgemeinschaft DFG

9:31AM M31.00008 Rheoscopic Fluids in a Post-Kalliroscope World, DANIEL BORRERO-ECHEVERRY, Department of Physics, Willamette University, 900 State St., Salem OR 97301, CHRISTOPHER J. CROWLEY, Center for Nonlinear Science and School of Physics, Georgia Institute of Technology, 837 State St., Atlanta, GA 30332 — In rheoscopic flow visualization the working fluid is seeded with small plate-shaped particles, which preferentially align in the flow due to their anisotropy. This leads to preferential light scattering, which highlights qualitatively different regions of the flow. For the past four decades, the gold standard in rheoscopic flow visualization has been Kalliroscope, a commercial product consisting of crystalline guanine particles. Guanine is a shiny compound extracted from fish scales and has traditionally been used in cosmetics to provide a pearlescent effect. It stands out among other options for rheoscopic flow visualization (e.g., aluminum flakes or coated mica particles) due to its relatively good density match with water. Guanine extraction, however, is an expensive process and as the cosmetics industry has adopted less expensive alternatives, commercial guanine production has dropped, leading to the closure of the Kalliroscope Corporation in 2014. In this talk, we discuss our recent discovery of a rheoscopic fluid based on stearic acid crystals, which has an overall performance similar to, and in some cases superior to, Kalliroscope. This rheoscopic fluid can be extracted from household items making it very inexpensive and readily accessible to researchers around the world.

9:44AM M31.00009 Modeling Shock Train Leading Edge Detection in Dual-Mode Scramjets, FOLUSO LADEINDE, ZHIPENG LOU, State Univ of NY- Stony Brook, WENHAI LI, TTC Technologies, Inc. Centereach, NY — The objective of this study is to accurately model the detection of shock train leading edge (STLE) in dual-mode scramjet (DMJS) engines intended for hypersonic flight in air-breathing propulsion systems. The associated vehicles have applications in military warfare and intelligence, and there is commercial interest as well. Shock trains are of interest because they play a significant role in the inability of a DMJS engine to develop the required propulsive force. The experimental approach to STLE detection has received some attention; as have numerical calculations. However, virtually all of the numerical work focus on mechanically- (i.e., pressure-) generated shock trains, which are much easier to model relative to the phenomenon in the real system where the shock trains are generated by combustion. A focus on combustion, as in the present studies, enables the investigation of the effects of equivalence ratio, which, together with the Mach number, constitutes an important parameter determining mode transition. The various numerical approaches implemented in our work will be reported, with result comparisons to experimental data. The development of an STLE detection procedure in an a priori manner will also be discussed.

9:57AM M31.00010 High speed velocimetry and concentration measurements in a microfluidic mixer using fluorescence confocal microscopy¹, VENKATESH INGUVA, BLAIR PEROT, University of Massachusetts Amherst, SAGAR KATHURIA, University of Massachusetts Medical School, JONATHAN ROTHSTEIN, University of Massachusetts Amherst, OSMAN BILSEL, University of Massachusetts Medical School — This work experimentally examines the performance of a quasi-turbulent micro-mixer that was designed to produce rapid mixing for protein-folding experiments. The original design of the mixer was performed using Direct Numerical Simulation (DNS) of the flow field and LES of the high Sc number scalar field representing the protein. The experimental work is designed to validate the DNS results. Both the velocity field and the protein concentration require validation. Different experiments were carried out to measure these two quantities. Concentration measurements are performed using a 488nm continuous wave laser coupled with a confocal microscope to measure fluorescence intensity during mixing. This is calibrated using the case where no mixing occurs. The velocity measurements use a novel high speed velocimetry technique capable of measuring speeds on the order of 10 m/s in a micro channel. The technique involves creating a pulsed confocal volume from a Ti-Sapphire laser with a pulse width of 260ns and observing the decay of fluorescence due to the fluid motion. Results from both experiments will be presented along with a comparison to the DNS results.

¹The work is supported by NSF IDBR Award No. 1353942

Tuesday, November 22, 2016 8:00AM - 10:10AM –
Session M32 Turbulent Boundary Layers: Structures II Oregon Ballroom 201 - Julio Soria,
Monash University

8:00AM M32.00001 Wall parallel cross-correlations of volumetric PTV measurements in a perturbed turbulent boundary layer , YAN MING TAN, ELLEN LONGMIRE, University of Minnesota

— A canonical turbulent boundary layer ($Re_\tau = 2500$) was perturbed by a narrowly spaced (0.2δ) array of cylinders extending normal to the wall. Two array heights were considered, $H=0.2\delta$ and $H=\delta$. Volumetric PTV measurements were acquired to understand 3-D variations in large scale structures within the log region of the unperturbed and perturbed flow. The recovery in the streamwise velocity coherence across the depth of the log region was analyzed using cross correlations between wall parallel planes. Conditional cross correlations are analyzed to examine the recovery in coherence specific to low momentum regions (LMRs), which can be signatures of vortex packets. The measurement volume was 0.70δ (streamwise, x), 0.90δ (spanwise, y), 0.12δ (wall-normal, z). In the unperturbed flow, LMRs frequently extended through the entire depth ($155 \leq z^+ \leq 465$). The cross correlations between planes at $z^+ = 155$ and $z^+ = 465$ exhibited strong skewness indicative of forward leaning structures. By comparison, downstream of the $H=\delta$ array, the wall normal extent of individual LMRs was frequently limited to the lower part of the measurement volume. The cross correlation magnitude and skewness remained suppressed relative to unperturbed flow up to 4.7δ downstream. These observations suggest reduced coherence of LMRs and high momentum regions across the log region. This result was consistent with previous planar PIV measurements at $z^+ = 500$ that showed hardly any long LMRs over distances up to 7δ downstream of the $H=\delta$ array.

8:13AM M32.00002 Analysis of coherent dynamical processes through computer vision , M. J. PHILIPP HACK, Center for Turbulence Research, Stanford University — Visualizations of turbulent boundary layers show an abundance of characteristic arc-shaped structures whose apparent similarity suggests a common origin in a coherent dynamical process. While the structures have been likened to the hairpin vortices observed in the late stages of transitional flow, a consistent description of the underlying mechanism has remained elusive. Detailed studies are complicated by the chaotic nature of turbulence which modulates each manifestation of the process and which renders the isolation of individual structures a challenging task. The present study applies methods from the field of computer vision to capture the time evolution of turbulent flow features and explore the associated physical mechanisms. The algorithm uses morphological operations to condense the structure of the turbulent flow field into a graph described by nodes and links. The low-dimensional geometric information is stored in a database and allows the identification and analysis of equivalent dynamical processes across multiple scales. The framework is not limited to turbulent boundary layers and can also be applied to different types of flows as well as problems from other fields of science.

8:26AM M32.00003 Evolution of vortex-surface fields in transitional boundary layers¹ , YUE YANG, YAOMIN ZHAO, SHIYING XIONG, Peking Univ — We apply the vortex-surface field (VSF), a Lagrangian-based structure-identification method, to the DNS database of transitional boundary layers (Sayadi et al., J. Fluid Mech., 724, 2013). The VSFs are constructed from the vorticity fields within a sliding window at different times and locations using a recently developed boundary-constraint method. The isosurfaces of VSF, representing vortex surfaces consisting of vortex lines with different wall distances in the laminar stage, show different evolutionary geometries in transition. We observe that the vortex surfaces with significant deformation evolve from wall-parallel planar sheets through hairpin-like structures and packets into a turbulent spot with regeneration of small-scale hairpins. From quantitative analysis, we show that a small number of representative or influential vortex surfaces can contribute significantly to the increase of the drag coefficient in transition, which implies a reduced-order model based on VSF.

¹This work has been supported in part by the National Natural Science Foundation of China (Grant Nos. 11472015, 11522215 and 11521091), and the Thousand Young Talents Program of China.

8:39AM M32.00004 3D critical layers in fully-developed turbulent flows¹ , THERESA SAXTON-FOX, BEVERLEY MCKEON, Caltech — Recent work has shown that 3D critical layers drive self-sustaining behavior of exact coherent solutions of the Navier-Stokes equations (Wang et al 2007; Hall and Sherwin 2010; Park and Graham 2015). This study investigates the role of 3D critical layers in fully-developed turbulent flows. 3D critical layer effects are identified in instantaneous snapshots of turbulent boundary layers in both experimental and DNS data (Wu et al 2014). Additionally, a 3D critical layer effect is demonstrated to appear using only a few resolvent response modes from the resolvent analysis of McKeon and Sharma 2010, with phase relationships appropriately chosen. Connections are sought to the thin shear layers observed in turbulent boundary layers (Klewicki and Hirschi 2004; Eisma et al 2015) and to amplitude modulation observations (Mathis et al 2009; Duvvuri and McKeon 2014).

The support of the Center for Turbulence Research (CTR) summer program at Stanford is gratefully acknowledged.

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.

8:52AM M32.00005 Coherent structures of a self-similar adverse pressure gradient turbulent boundary layer¹ , ATSUSHI SEKIMOTO, VASSILI KITSIOS, CALLUM ATKINSON, Monash Univ, JAVIER JIMÉNEZ, U. Politécnica Madrid, JULIO SORIA, Monash Univ — The turbulence statistics and structures are studied in direct numerical simulation (DNS) of a self-similar adverse pressure gradient turbulent boundary layer (APG-TBL). The self-similar APG-TBL at the verge of separation is achieved by a modification of the far-field boundary condition to produce the desired pressure gradient. The turbulence statistics in the self-similar region collapse by using the scaling of the external velocity and the displacement thickness. The coherent structures of the APG-TBL are investigated and compared to those of zero-pressure gradient case and homogeneous shear flow.

¹The support of the ARC, NCI and Pawsey SCC funded by the Australian and Western Australian governments as well as the support of PRACE funded by the European Union are gratefully acknowledged.

9:05AM M32.00006 Turbulent Spots Inside the Turbulent Boundary Layer , JINHIE SKARDA, Center for Turbulence Research, Stanford University, XIAOHUA WU, Royal Military College of Canada, PARVIZ MOIN, ADRIAN LOZANO-DURAN, Center for Turbulence Research, Stanford University, JAMES WALLACE, University of Maryland, JEAN-PIERRE HICKEY, University of Waterloo — We present evidence that the buffer region of the canonical turbulent boundary layer is populated by locally generated turbulent spots, which cause strong indentations on the near-wall low-momentum streaks. This evidence is obtained from a spatially-developing direct numerical simulation carrying the inlet Blasius boundary layer through a bypass transition to the turbulent boundary layer state over a moderate Reynolds number range. The turbulent spots are structurally analogous to their transitional counter-parts but without any direct causality connection. High-pass filtered time-history records are used to calculate the period of turbulent spot detection and this period is compared to the boundary layer bursting period reported in hot-wire experiments. The sensitivity of the results to parameters such as the high pass filter frequency and the amplitude discriminator level is examined. The characteristics of these turbulent spots are also quantified using a spatial connectivity based conditional sampling technique. This evidence seems to be at odds with the notion that the buffer region is dominated by quasi-streamwise vortices, and contributes to the potential unification of the studies on near-wall turbulent boundary layer dynamics.

9:18AM M32.00007 Velocity-vorticity correlation structures in compressible turbulent boundary layer¹, JUN CHEN, SHI-YAO LI, ZHEN-SU SHE, Peking Univ. — A velocity-vorticity correlation structure (VVCS) analysis is applied to analyze data of 3-dimensional (3-D) direct numerical simulations (DNS), to investigate the quantitative properties of the most correlated vortex structures in compressible turbulent boundary layer (CTBL) at Mach numbers, $Ma = 2.25$ and 6.0 . It is found that the geometry variation of the VVCS closely reflects the streamwise development of CTBL. In laminar region, the VVCS captures the instability wave number of the boundary layer. The transition region displays a distinct scaling change of the dimensions of VVCS. The developed turbulence region is characterized by a constant spatial extension of the VVCS. For various Mach numbers, the maximum correlation coefficient of the VVCS presents a clear multi-layer structure with the same scaling laws as a recent symmetry analysis proposed to quantifying the sublayer, the log-layer, and the wake flow. A surprising discovery is that the wall friction coefficient, C_f , holds a “-1”-power law of the wall normal distance of the VVCS, y_s . This validates the speculation that the wall friction is determined by the near-wall coherent structure, which clarifies the correlation between statistical structures and the near-wall dynamics.

¹Project 11452002 and 11172006 supported by National Natural Science Foundation of China

9:31AM M32.00008 Evolution and formation of shear layers in a developing turbulent boundary layer¹, JUNGHOON LEE, JASON MONTY, NICHOLAS HUTCHINS, The University of Melbourne — The evolution and formation mechanism of shear layers in the outer region of a turbulent boundary layer are investigated using time-resolved PIV datasets of a developing turbulent boundary layer from inception at the trip up to $Re_\tau = 3000$. An analysis of a sequence of instantaneous streamwise velocity fluctuation fields reveals that strong streamwise velocity gradients are prevalent along interfaces where low- and high-speed regions interact. To provide an insight on how such regions are associated with the formation of shear layers in the outer regions, we compute conditional averages of streamwise velocity fluctuations based on a strong shear layer. Our results reveal that one possible mechanism for the generation of shear layers in the outer region is due to the mismatch in the convection velocities between low- and high-speed regions. The results also indicate that the angle of the inclined shear layer is developing in time. In addition, the conditionally averaged velocity fluctuations exhibit a local instability along these shear layers, leading to a shear layer roll-up event as the layers evolve in time. Based on these findings, we propose a conceptual model which describes dynamic interactions of shear layers and their associated large-scale coherent motions.

¹The authors wish to acknowledge the financial support of the Australian Research Council

9:44AM M32.00009 Traveling wave solutions of large-scale structures in turbulent channel flow at $Re_\tau = 1000$.¹, YONGYUN HWANG, Department of Aeronautics, Imperial College London, ASHLEY WILLIS, School of Mathematics and Statistics, University of Sheffield, CARLO COSSU, Institut de Mcanique des Fluides de Toulouse — Recently, a set of stationary invariant solutions for the large-scale structures in turbulent Couette flow was computed at $Re_\tau \simeq 128$ using an over-damped LES with the Smagorinsky model which accounts the effect of the surrounding small-scale motions (Rawat et al., 2015, J. Fluid Mech., 782:515). In this talk, we show that this approach can be extended to $Re_\tau \simeq 1000$ in turbulent channel flow, towards the regime where the large-scale structures in the form of very-large-scale motions (long streaky motions) and large-scale motions (short vortical structures) energetically emerge. We demonstrate that a set of invariant solutions in the form of a traveling wave can be computed from simulations of the self-sustaining large-scale structures in the minimal unit with midplane reflection symmetry. By approximating the surrounding small scales with an artificially elevated Smagorinsky constant, a set of equilibrium states are found, labelled upper- and lower-branch according to their related wall shear stress. In particular, we will show that the upper-branch equilibrium state is a reasonable proxy for the spatial structure and the turbulent statistics of the self-sustaining large-scale structures.

¹Engineering and Physical Sciences Research Council, UK (EP/N019342/1)

9:57AM M32.00010 Time-evolution of uniform momentum zones in a turbulent boundary layer, ANGELIKI LASKARI, R. JASON HEARST, ROELAND DE KAT, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton — Time-resolved planar particle image velocimetry (PIV) is used to analyse the organisation and evolution of uniform momentum zones (UMZs) in a turbulent boundary layer. Experiments were performed in a recirculating water tunnel on a streamwise-wall-normal plane extending approximately $0.5\delta \times 1.8\delta$, in x and y , respectively. In total 400,000 images were captured and for each of the resulting velocity fields, local peaks in the probability density distribution of the streamwise velocity were detected, indicating the instantaneous presence of UMZs throughout the boundary layer. The main characteristics of these zones are outlined and more specifically their velocity range and wall-normal extent. The variation of these characteristics with wall normal distance and total number of zones are also discussed. Exploiting the time information available, time-scales of zones that have a substantial coherence in time are analysed and results show that the zones’ lifetime is dependent on both their momentum deficit level and the total number of zones present. Conditional averaging of the flow statistics seems to further indicate that a large number of zones is the result of a wall-dominant mechanism, while the opposite implies an outer-layer dominance.

Tuesday, November 22, 2016 8:00AM - 10:10AM – Session M33 Turbulence: Flow Thru Pipes Oregon Ballroom 202 - Pinaki Chakraborty, Okinawa Institute of Science and Technology

8:00AM M33.00001 Turbulence in Reynolds’ flashes¹, RORY CERBUS, CHIEN-CHIA LIU, GUSTAVO GIOIA, PINAKI CHAKRABORTY, Okinawa Institute of Science and Technology — Osborne Reynolds’ seminal work from 1883 revealed that the transition from quiescent, laminar flow to a turbulent pipe filled with roiling eddies is mediated by localized flashes of fluctuations. Later work has unveiled many features of these flashes: they proliferate or fade away, maintain their shape or continually expand. The nature of the fluctuations in the flashes, however, has remained mysterious. Here, using measures traditionally attributed to high Reynolds number (Re) flows, we present experimental results on the fluctuations of the flashes. Our results suggest that the transition to turbulence is the low Re limit of the high Re , fully developed flow.

¹Okinawa Institute of Science and Technology

8:13AM M33.00002 Self-similarity of the large-scale motions in turbulent pipe flow¹

, LEO HELLSTRÖM, Princeton University, IVAN MARUSIC, University of Melbourne, ALEXANDER SMITS, Princeton University; Monash University — Townsend's attached eddy hypothesis assumes the existence of a set of energetic and geometrically self-similar eddies in the logarithmic layer in wall-bounded turbulent flows. These eddies can be completely scaled with the distance from their center to the wall. We performed stereo PIV measurements together with a proper orthogonal decomposition (POD) analysis, to address the self-similarity of the energetic motions, or eddies, in fully-developed turbulent pipe flow. The resulting modes/eddies, extracted at $Re_\tau = 2460$, show a self-similar behavior for eddies with wall-normal length scales spanning a decade. This single length scale provides a complete description of the cross-sectional shape of the self-similar eddies.

¹ONR Grant N00014-15-1-2402 and the Australian Research Council

8:26AM M33.00003 High Reynolds number examination for fully developed pipe flow - Mean velocity profile and friction factor¹, NORIYUKI FURUICHI, AIST, NMJJ, YUKI WADA, YOSHIYUKI TSUJI, Nagoya Univ., YOSHIYA TERAO, AIST, NMJJ — The pipe flow examinations at high Reynolds numbers up to $Re=18,000,000$ are performed using the high Reynolds number actual flow facility Hi-Reff at AIST, NMJJ. The precise measurements of the friction factor and velocity profile are achieved by the highly accurate measurement of the flow rate. The friction factor data is obviously different from the Prandtl equation and the experimental results from the Superpipe at Princeton University. The deviation from the Superpipe is -6% at $Re=10,000,000$. The velocity profile is measured by LDV. The consistency between the mean velocity profile and the friction factor measured is investigated. The velocity profile data is fitted to a velocity profile form based on the log law, and an equation for the friction factor is derived by integration. The derived equation for the friction factor accurately represents the friction factor data. The deviation from the friction factor data is less than 1%. Based on the equations for the friction factor derived using the mean velocity profile, the best-fitting constants for the friction factor data are also proposed. For the high Reynolds number region, the Karman constant given by the velocity profile and the friction factor is completely consistent and it is 0.383.

8:39AM M33.00004 Statistical growths of turbulent structures in a pipe flow¹, JUNSUN AHN, HYUNG JIN SUNG, KAIST — The streamwise and spanwise (or azimuthal) growths of turbulent coherent structures in a turbulent pipe flow ($Re_\tau = 3008$) are explored. Two-point correlation and 1-D pre-multiplied energy spectra of the streamwise velocity fluctuations are obtained to analyze the statistical growths of the streamwise and spanwise structures. The streamwise and spanwise length scales linearly grow along the wall-normal distance and the relationship between both length scales is shown to be linear, which support the attached eddy hypothesis. Furthermore, the statistical scalings of the coherent structures are demonstrated and compared to 2-D pre-multiplied energy spectra of the streamwise velocity fluctuations. Finally, the relationship between the streamwise and spanwise structures is analyzed by using the POD based on the translational invariance method (Duggleby et al. 2009). Several representative energetic modes are observed. The combinations of the energetic modes are used to examine the behaviors of the large- and very-large-scale motions.

¹This work was supported by the Creative Research Initiatives (No. 2016-004749) program of the National Research Foundation of Korea (MSIP) and partially supported by KISTI under the Strategic Supercomputing Support Program.

8:52AM M33.00005 Five layers in a turbulent pipe flow¹, JINYOUNG LEE², JUNSUN AHN, HYUNG JIN SUNG, KAIST — The scaling laws governing the five layers of the mean velocity distribution of a turbulent pipe flow were characterized using the available DNS data ($Re_\tau = 544, 934, 3008$). Excluding the very near-wall and core regions, the buffer, meso- and log layers were identified by examining the streamwise mean momentum equation and the net force spectra. The (outer) log layer was located in the overlap region where the viscous force was negligible. Another (inner) log layer was observed in the buffer layer, in which the viscous force was directly counterbalanced by the turbulent inertia. A meso-layer between the buffer and outer log layers was found to feature viscous effects. The acceleration force of the large-scale motions (LSMs) penetrated the outer log layer at higher Reynolds numbers, as observed in the net force spectra. The acceleration force of the LSMs became strong and was counterbalanced by the deceleration force of the small-scale motions (SSMs), indicating that the inner and outer length scales contributed equally to the meso-layer. The outer log layer was established by forming an extended connection link between the meso- and outer layers.

¹This work was supported by the Creative Research Initiatives (No. 2016-004749) program of the National Research Foundation of Korea (MSIP) and partially supported by KISTI under the Strategic Supercomputing Support Program.

²The first two authors contributed equally to this work.

9:05AM M33.00006 Effect of Reynolds number on flow and mass transfer characteristics of a 90 degree elbow¹, NOBUYUKI FUJISAWA, YUYA IKARASHI, TAKAYUKI YAMAGATA, SYOICHI TAGUCHI, Niigata University — The flow and mass transfer characteristics of a 90 degree elbow was studied experimentally by using the mass transfer measurement by plaster dissolution method, the surface flow visualization by oil film method and stereo PIV measurement. The experiments are carried out in a water tunnel of a circular pipe of 56mm in diameter with a working fluid of water. The Reynolds number was varied from 30000 to 200000. The experimental result indicated the change of the mass transfer coefficient distribution in the elbow with increasing the Reynolds number. This phenomenon is further examined by the surface flow visualization and measurement of secondary flow pattern in the elbow, and the results showed the suggested change of the secondary flow pattern in the elbow with increasing the Reynolds numbers.

9:18AM M33.00007 Turbulent pipe flows subjected to temporal decelerations¹, WONG-WAN JEONG, HYUNGJAE LIM, JAE HWA LEE, Ulsan Natl Inst of Sci & Tech — Direct numerical simulations of temporally decelerating turbulent pipe flows were performed to examine effects of temporal decelerations on turbulence. The simulations were started with a fully developed turbulent pipe flow at a Reynolds number, $Re_D=24380$, based on the pipe radius (R) and the laminar centerline velocity (U_{c0}). Three different temporal decelerations were imposed to the initial flow with $f=|dU_b/dt|=-0.00127, 0.00625$ and 0.025 , where U_b is the bulk mean velocity. Comparison of Reynolds stresses and turbulent production terms with those for steady flow at a similar Reynolds number showed that turbulence is highly intensified with increasing f due to delay effects. Furthermore, inspection of the Reynolds shear stress profiles showed that strong second- and fourth-quadrant Reynolds shear stresses are greatly increased, while first- and third-quadrant components are also increased. Decomposition of streamwise Reynolds normal stress with streamwise cutoff wavelength (λ_x) $1R$ revealed that the turbulence delay is dominantly originated from delay of strong large-scale turbulent structures in the outer layer, although small-scale motions throughout the wall layer adjusted more rapidly to the temporal decelerations.

¹This research was supported by Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Education (NRF-2014R1A1A2057031).

9:31AM M33.00008 Transition in Pulsatile Pipe Flow , PAVLOS VLACHOS, MELISSA BRINDISE,

Purdue Univ — Transition has been observed to occur in the aorta, and stenotic vessels, where pulsatile flow exists. However, few studies have investigated the characteristics and effects of transition in oscillating or pulsatile flow and none have utilized a physiological waveform. In this work, we explore transition in pipe flow using three pulsatile waveforms which all maintain the same mean and maximum flow rates and range to zero flow, as is physiologically typical. Velocity fields were obtained using planar particle image velocimetry for each pulsatile waveform at six mean Reynolds numbers ranging between 500 and 4000. Turbulent statistics including turbulent kinetic energy (TKE) and Reynolds stresses were computed. Quadrant analysis was used to identify characteristics of the production and dissipation of turbulence. Coherent structures were identified using the λ_{ci} method. We developed a wavelet-Hilbert time-frequency analysis method to identify high frequency structures and compared these to the coherent structures. The results of this study demonstrate that the different pulsatile waveforms induce different levels of TKE and high frequency structures, suggesting that the rates of acceleration and deceleration influence the onset and development of transition.

9:44AM M33.00009 Numerical simulation of pulsatile flow in rough pipes , CHENG CHIN,

JASON MONTY, ANDREW OOI, SIMON ILLINGWORTH, IVAN MARUSIC, ALEX SKVORTSOV, The University of Melbourne — Direct numerical simulation (DNS) of pulsatile turbulent pipe flow is carried out over three-dimensional sinusoidal surfaces mimicking surface roughness. The simulations are performed at a mean Reynolds number of $Re_\tau \approx 540$ (based on friction velocity, u_τ , and pipe radii, δ) and at various roughness profiles following the study of Chan et al. (JFM 771, 743 - 777, 2015), where the size of the roughness (roughness semi-amplitude height h^+ and wavelength λ^+) is increased geometrically while maintaining the height-to-wavelength ratio of the sinusoidal roughness element. Results from the pulsatile simulations are compared with non-pulsatile simulations to investigate the effects of pulsation on the Hama roughness function, ΔU^+ . Other turbulence statistics including mean turbulence intensities, Reynolds stresses and energy spectra are analysed. In addition, instantaneous phase (eg. at maximum and minimum flow velocities) and phase-averaged flow structures are presented and discussed.

9:57AM M33.00010 Generation of Turbulent Inflow Conditions for Pipe Flow via an

Annular Ribbed Turbulator , NIMA MOALLEMI, JOSHUA BRINKERHOFF, Univ of British Columbia — The generation of turbulent inflow conditions adds significant computational expense to direct numerical simulations (DNS) of turbulent pipe flows. Typical approaches involve introducing boxes of isotropic turbulence to the velocity field at the inlet of the pipe. In the present study, an alternative method is proposed that incurs a lower computational cost and allows the anisotropy observed in pipe turbulence to be physically captured. The method is based on a periodic DNS of a ribbed turbulator upstream of the inlet boundary of the pipe. The Reynolds number based on the bulk velocity and pipe diameter is 5300 and the blockage ratio (BR) is 0.06 based on the rib height and pipe diameter. The pitch ratio is defined as the ratio of rib streamwise spacing to rib height and is varied between 1.7 and 5.0. The generation of turbulent flow structures downstream of the ribbed turbulator are identified and discussed. Suitability of this method for accurate representation of turbulent inflow conditions is assessed through comparison of the turbulent mean properties, fluctuations, Reynolds stress profiles, and spectra with published pipe flow DNS studies. The DNS results achieve excellent agreement with the numerical and experimental data available in the literature.

Tuesday, November 22, 2016 8:00AM - 10:10AM –

Session M34 Turbulence: DNS Methods, Process, and Analysis Oregon Ballroom 203 -
Guillaume Blanquart, Caltech

8:00AM M34.00001 Two-step simulation of velocity and passive scalar mixing at high Schmidt number in turbulent jets¹ , K. JEFF RAH, GUILLAUME BLANQUART, California Institute of Technology

Simulation of passive scalar in the high Schmidt number turbulent mixing process requires higher computational cost than that of velocity fields, because the scalar is associated with smaller length scales than velocity. Thus, full simulation of both velocity and passive scalar with high Sc for a practical configuration is difficult to perform. In this work, a new approach to simulate velocity and passive scalar mixing at high Sc is suggested to reduce the computational cost. First, the velocity fields are resolved by Large Eddy Simulation (LES). Then, by extracting the velocity information from LES, the scalar inside a moving fluid blob is simulated by Direct Numerical Simulation (DNS). This two-step simulation method is applied to a turbulent jet and provides a new way to examine a scalar mixing process in a practical application with smaller computational cost.

¹NSF, Samsung Scholarship

8:13AM M34.00002 From laminar to fully-developed turbulence in a 500 radii long Osborne Reynolds pipe flow: A direct numerical simulation study.¹ , RONALD ADRIAN, Arizona

State University, XIAOHUA WU, Canadian Royal Military College, PARVIZ MOINN, Stanford University — We report our new direct numerical simulation results of the Osborne Reynolds' pipe transition problem in a 500 radii long configuration. The inlet disturbance is generated through a three degree narrow wedge. The present radial-mode inlet disturbance is in contrast to our earlier simulation design using a wire-ring at the inlet, which is circumferential-mode in nature (Wu et al, PNAS, 1509451112, 2015). The current mesh size is $16384 \times 201 \times 512$, and the simulation Reynolds number is 6500 based on the pipe diameter and bulk velocity. Statistics in the fully-developed turbulent region are in good agreement with those sampled from an auxiliary short turbulent pipe simulation using the streamwise periodic boundary condition. Frequency spectra of the turbulence kinetic energy are computed at six streamwise stations, namely, 30R, 60R, 90R, 350R, 400R and 450R downstream of the inlet. Surprisingly, spectra in the late transitional region (60R and 90R) exhibit stronger high frequency content than those in the fully-developed turbulent region. Contours of a passive scalar indicate the existence of patchy turbulent spot structures, even in the fully-developed turbulent region.

¹We acknowledge partial support from NSF award number CBET-1335731.

8:26AM M34.00003 Characteristics of the mixing volume model with the interactions among spatially distributed particles for Lagrangian simulations of turbulent mixing¹

, TOMOAKI WATANABE, KOJI NAGATA, Nagoya University — The mixing volume model (MVM), which is a mixing model for molecular diffusion in Lagrangian simulations of turbulent mixing problems, is proposed based on the interactions among spatially distributed particles in a finite volume. The mixing timescale in the MVM is derived by comparison between the model and the subgrid scale scalar variance equation. A-priori test of the MVM is conducted based on the direct numerical simulations of planar jets. The MVM is shown to predict well the mean effects of the molecular diffusion under various conditions. However, a predicted value of the molecular diffusion term is positively correlated to the exact value in the DNS only when the number of the mixing particles is larger than two. Furthermore, the MVM is tested in the hybrid implicit large-eddy-simulation/Lagrangian-particle-simulation (ILES/LPS). The ILES/LPS with the present mixing model predicts well the decay of the scalar variance in planar jets.

¹This work was supported by JSPS KAKENHI Nos. 25289030 and 16K18013. The numerical simulations presented in this manuscript were carried out on the high performance computing system (NEC SX-ACE) in the Japan Agency for Marine-Earth Science and Technology.

8:39AM M34.00004 Lagrangian study of acceleration in a turbulent channel flow , JUAN IGNACIO POLANCO, IVANA VINKOVIC, LMFA, Université Lyon 1, INSA Lyon, Ecole Centrale Lyon, CNRS UMR 5509, NICKOLAS STELZENMULLER, NICOLAS MORDANT, LEGI, Université Grenoble Alpes, CNRS UMR 5519 — A Lagrangian characterisation of a wall-bounded turbulent flow is presented. Tracking of fluid tracers is performed in experiments and direct numerical simulations of channel flow at $Re_T \approx 1450$. Near the channel walls, the presence of large-scale streamwise vortices is associated with high-magnitude centripetal accelerations of tracer particles. This presents a clear signature on time autocorrelations of particle acceleration. Temporal cross-correlations and joint probability density functions of acceleration are used to describe the dependency between acceleration components. It is found that, near the walls, negative streamwise accelerations are associated with wall-normal accelerations directed towards the channel centre. This is due to the combined influence of viscous effects and wall-confinement. Good quantitative agreement between experiments and simulations is obtained. Preliminary results on relative dispersion of fluid particle pairs in channel flow are presented and compared with theoretical scalings derived for homogeneous isotropic turbulence. Presented Lagrangian properties can serve as basis for stochastic modelling of transport and dispersion of pollutants by atmospheric flows.

8:52AM M34.00005 Direct numerical simulation of turbulent boundary layer with constant thickness¹ , YICHEN YAO, CHUNXIAO XU, WEIXI HUANG, Tsinghua Univ — Direct numerical simulation is performed to turbulent boundary layer (TBL) with constant thickness at $Re_\theta = 1420$. Periodic boundary condition is applied in the streamwise direction, and a mean body force equivalent to the convection term in the mean momentum equation is imposed in this direction. The body force is calculated using the published TBL data of Schlatter and Orlu (2010) at $Re_\theta = 1420$. The presently simulated TBL is compared with the conventional TBL and turbulent channel flow at the prescribed Reynolds number. The turbulent statistics agrees well with that of Schlatter and Orlu (2010). The pre-multiplied energy spectra in current simulation also present high similarity with the conventional TBL, while differ obviously with those in turbulent channel. The successful replication of turbulent boundary in the current simulation provides an alternative method for boundary layer simulation with much less computational cost. Meanwhile, in aspect of both turbulent statistics and flow structures, the current results indicate that the differences between turbulent channel and boundary layer flow mainly caused by the discrepancy in driving force distribution rather than the periodic boundary restriction.

¹National Natural Science Foundation of China (Project No. 11490551, 11472154, 11322221, 11132005)

9:05AM M34.00006 Non-Boussinesq effects on buoyancy-driven variable-density turbulence¹ , DENIS ASLANGIL, Lehigh University, DANIEL LIVESCU, Los Alamos National Laboratory, ARINDAM BANERJEE, Lehigh University — Non-Boussinesq effects on turbulent mixing of a heterogeneous mixture of two incompressible, miscible fluids with different densities are investigated in terms of properly normalized L^{2m} -norms of density gradient by means of high-resolution Direct Numerical Simulations. In a triply periodic three-dimensional domain, the mixing occurs in response to stirring induced by buoyancy-generated motions between two fluids which are initially segregated in random patches. During the flow evolution, the density gradient can reach high values even at low Atwood numbers indicating that non-Boussinesq effects play a crucial role within the flow. The results cover a broad range of Reynolds numbers and non-dimensional density ratios (Atwood numbers, A) including small ($A=0.05$), moderate ($A=0.25$ and 0.5), and high ($A=0.75$) values. An asymmetric behavior is detected on the probability density function of the density gradient at high Atwood numbers. The evolution of the density gradient and the hierarchy of its higher order norms are also investigated by decomposing the flow into the different flow regions by using density as a fluid marker. It is found that the density gradient is much larger in regions of light fluid compared to regions occupied by the heavier fluid, indicating a strong mixing asymmetry between light and heavy fluids. This shows that Boussinesq models may not be adequate even at low density ratios; contrary to what statistics based on the entire domain.

¹AB acknowledges NSF Career Award 1453056.

9:18AM M34.00007 Turbulent heat-and-mass transfer in channel flow at transcritical temperature conditions , KUKJIN KIM, CARLO SCALO, Purdue University, JEAN-PIERRE HICKEY, University of Waterloo — Turbulent heat and mass transfer at transcritical thermodynamic conditions is studied in turbulent channel flow using the high-fidelity DNS for solution to the compressible Navier-Stokes equations in the conservative form closed with the Peng-Robinson state equation. To isolate the real fluid effects on turbulent heat transfer, the bulk pressure is maintained at supercritical $p_b = 1.1p_c$ and the isothermal walls are set to $\Delta T/2$ above and below the local pseudo-boiling temperature T_{pb} of the fluid (R-134a) where ΔT is 5K, 10K, and 20K. This setup allows the flow to reach a statistically-steady state while capturing the highest thermodynamic gradients, thus allowing a detailed study on thermodynamics of transcritical turbulent heat transfer. All thermodynamic and turbulent scales are fully resolved which is shown through a careful grid convergence analysis. The time-averaged density and compressibility factor are highly dependent on the temperature field and their large near-wall gradient causes thermodynamically-induced peaks in the RMS quantities resulting in strong turbulent mixing. The ejection of heavy pseudo-liquid blobs by near-wall turbulent structures into the channel core leads to a third RMS peak which is not observable in ideal gas simulations.

9:31AM M34.00008 Quantitative evaluation on contributions of laminar, turbulence and secondary flow to momentum and heat transfer in rhombic ducts, NAOYA FUKUSHIMA, Tokyo University of Science — In the present study, Direct Numerical Simulation of turbulent flow in rhombic ducts have been carried out to investigate effects of the corner angle on the friction and heat transfer. Due to secondary flow of the second kind, the friction and heat transfer are enhanced in the corner, while turbulence enhances momentum and heat transfer near the wall away from the corner. In previous studies, turbulence and secondary flows are supposed to enhance momentum and heat transfer, qualitatively. The quantitative estimation of their contribution has not been clarified yet. Fukagata, Iwamoto and Kasagi (2002) have theoretically driven the FIK-identity to evaluate quantitative contributions of laminar and turbulence to the friction in turbulent channel. In this study, the FIK-identity has been numerically applied to DNS data in the rhombic ducts to evaluate quantitative contributions of laminar, turbulence and secondary flow to the momentum and heat transfer. From the results, it is quantitatively clarified that the contributions of turbulence and secondary flow to heat transfer are larger than that to friction in the rhombic ducts.

9:44AM M34.00009 Direct numerical simulation of chemical fouling in a turbulent channel flow, HANBYEOL KIM, HAECHON CHOI, Seoul Natl Univ — In heat exchanger industries, the fouling deposition on solid surfaces causes serious problems such as impaired heat transfer and increased pressure drop. Thus, the prediction and mitigation of fouling deposits has been an important issue. In the present study, we conduct direct numerical simulation of a fully developed channel flow. We assume that fluid flows as a slurry flow and the apparent viscosity is calculated using Thomas' equation. We also assume that a second order reaction with single soluble reactant occurs and a highly viscous product is accumulated at high-temperature wall. Two passive scalar equations are solved along with the Navier-Stokes equations to obtain the mass fraction of the reactant and the temperature in the channel. The reactant flows into the channel at the inlet, and the highly viscous product is stuck on the wall and forms a fouling layer.

9:57AM M34.00010 Computing Normal Shock-Isotropic Turbulence Interaction With Tetrahedral Meshes and the Space-Time CESE Method¹, BALAJI SHANKAR VENKATACHARI, National Institute of Aerospace, CHAU-LYAN CHANG, NASA Langley Research Center — The focus of this study is scale-resolving simulations of the canonical normal shock- isotropic turbulence interaction using unstructured tetrahedral meshes and the space-time conservation element solution element (CESE) method. Despite decades of development in unstructured mesh methods and its potential benefits of ease of mesh generation around complex geometries and mesh adaptation, direct numerical or large-eddy simulations of turbulent flows are predominantly carried out using structured hexahedral meshes. This is due to the lack of consistent multi-dimensional numerical formulations in conventional schemes for unstructured meshes that can resolve multiple physical scales and flow discontinuities simultaneously. The CESE method — due to its Riemann-solver-free shock capturing capabilities, non-dissipative baseline schemes, and flux conservation in time as well as space — has the potential to accurately simulate turbulent flows using tetrahedral meshes. As part of the study, various regimes of the shock-turbulence interaction (wrinkled and broken shock regimes) will be investigated along with a study on how adaptive refinement of tetrahedral meshes benefits this problem.

¹The research funding for this paper has been provided by Revolutionary Computational Aerosciences (RCA) subproject under the NASA Transformative Aeronautics Concepts Program (TACP).

Tuesday, November 22, 2016 8:00AM - 10:10AM –
Session M35 Turbulence: Theory and Semi-empirical Models Oregon Ballroom 204 - P.K. Yeung, Georgia Tech University

8:00AM M35.00001 Scaling properties of the mean equation for passive scalar in turbulent channel flow, ANG ZHOU, University of New Hampshire, SERGIO PIROZZOLI, University of Rome La Sapienza, JOSEPH KLEWICKI, University of New Hampshire, University of Melbourne — Data from numerical simulations of fully developed turbulent channel flows subjected to a uniform and constant heat generation are used to explore the scaling behaviors admitted by the mean equation for passive scalar transport. The analysis proceeds in a manner similar to previous studies of mean momentum transport. Based on the relative magnitude of terms, the leading order balances in the equation organize into a four layer structure. The wall-normal widths of the layers exhibit significant dependencies both on Reynolds and Prandtl number, and these dependencies are analytically surmised and empirically validated. The passive scalar equation also admits an invariant form on each of a hierarchy of scaling layers. As with the momentum case, this hierarchy is quantified by its inner-normalized widths. The present findings indicate that the layer width distribution is increasingly approximated by a linear function of wall normal position with increasing ratio of Reynolds number to Prandtl number on a domain of the hierarchy where the molecular diffusion effect loses leading order. The analysis indicates that across this domain the square of the slope of the width distribution function is equivalent to the scalar Karman constant as Reynolds number goes to infinity. The data provide convincing evidence in support of this finding.

8:13AM M35.00002 Three-dimensional structure of alternative Reynolds stresses in turbulent channels¹, KOSUKE OSAWA, Tokyo Institute of Technology and UPM, JAVIER JIMENEZ, Universidad Politecnica de Madrid — As explained in another talk in this meeting, the ambiguity of the fluxes in the momentum conservation law allows alternative definitions for the Reynolds stresses. We study here the three-dimensional structures of the tangential stress that minimises the total r.m.s. flux fluctuations in turbulent channels at several $Re_T \geq 10^3$. As in the case of the classical shear stress, it is found that the structures can be classified into wall-detached and wall-attached families. The latter carry most of the overall stress and are geometrically self-similar, although less elongated than for the classical ones. Although they span the full range of scales from viscous to the channel height, larger structures are less common than in the classical case, apparently missing very large 'global' modes. They are also less fractal ($D_F \approx 2.5$) than the 'sponges of flakes' of the classical quadrant structures ($D_F \approx 2.1$), and more inclined with respect to the wall, 45° versus 20° , suggesting that they may be related to the 'hairpin legs' discussed by several authors.

¹Funded by the Coturb project of the ERC and Erasmus Mundus

8:26AM M35.00003 Determination of the Reynolds stress in canonical flow geometries

, T.-W. LEE, Arizona State University — We present a new theoretical result for solving for the Reynolds stress in turbulent flows, and show how it works for canonical flow geometries: flow over a flat plate, channel flow, and axi-symmetric jets. The theory is based on fundamental physics of turbulence transport. Comparison of the current theoretical result with experimental and DNS (direct numerical simulation) data show good agreement, and various considerations of the results indicate that this is not a fortuitous coincidence, and point to radically new solutions for Reynolds stress. The theory leads to a closed-form formula for the Reynolds stress in terms of the root variables, such as the mean velocity, velocity gradient, turbulence kinetic energy and a viscous term. The form of the solution also provides insight on how the Reynolds stress is generated and distributed. This is *not* a modeling study, but a theoretical one based on physical principles although some of the nuances are still being examined. Details of the theory are submitted elsewhere, and also will be presented at the conference. The theoretical result for the Reynolds stress is compared with various experimental and DNS data. The agreement is nearly perfect at low Reynolds numbers, which gives some confidence that we have captured the true physics of turbulent transport, and that the results are not a fortuitous coincidence.

8:39AM M35.00004 Corrections to the 4/5-law for decaying turbulence

, JONAS BOSCHUNG, RWTH Aachen University, MICHAEL GAUDING, TU Freiberg, FABIAN HENNIG, DOMINIK DENKER, HEINZ PITTSCH, RWTH Aachen University — We examine finite Reynolds number contributions to the inertial range solution of the third order structure functions stemming from the unsteady and viscous terms for decaying turbulence. Under the assumption that the second order correlations f and g are self-similar under a coordinate change, we are able to rewrite the exact second order equations as function of a normalised scale r only with the decay exponent as a parameter. We close the resulting system of equations using a power law and an eddy-viscosity ansatz. If we further assume K41 scaling, we find the same Reynolds number dependence as previously in the literature.

8:52AM M35.00005 Wall-roughness induced ultimate Taylor-Couette turbulence

, XIAOJUE ZHU, ROBERTO VERZICCO, DETLEF LOHSE, Physics of Fluids Group, University of Twente — We use direct numerical simulations to examine the Taylor number (Ta) dependence of the torque required to drive rotating cylinders with smooth and/or rough walls in Taylor-Couette turbulence. With the introduction of both inner and outer wall roughness, the scaling of the dimensionless torque Nu_ω becomes $Nu_\omega \propto Ta^{0.5 \pm 0.01}$. We interpret this through an extension of the Grossmann-Lohse theory [Phys. Fluids 23, 045108 (2011)], by accounting for the log-law of the wall in the presence of roughness. The logarithmic correction $L(Re)$ in the relation $Nu_\omega \propto Ta^{1/2} \times L(Re)$, which leads to the effective scaling $Nu_\omega \propto Ta^{0.38}$ in the ultimate regime for the smooth case, gets canceled out and Kraichnan's pure ultimate scaling $Nu_\omega \propto Ta^{1/2}$ [R. H. Kraichnan, Phys. Fluids 5, 1374 (1962)] is recovered.

9:05AM M35.00006 Turbulent boundary-layer flow over a long plate with a uniformly rough surface , D.I. PULLIN, California Institute of Technology, N. HUTCHINS, D. CHUNG, University of Melbourne — We develop a semi-empirical model for a zero-pressure-gradient turbulent boundary layer flowing over a flat plate of length L and covered with homogeneous, uniform roughness of equivalent sand-grain roughness k_s . Use is made of the log-wake model for the stream-wise mean velocity that includes a transitional-asymptotic roughness correction together with the Kármán integral relation. For $Re_L = U_\infty L/\nu$ very large, the velocity ratio $S = U_\infty/u_\tau$ at $x = L$, the plate drag coefficient C_D and other mean-flow properties can be obtained for given Re_L and k_s/L . Three distinct cases are discussed; the smooth-wall, fully-rough and long-plate limits. Of these, the most important is the fully-rough case where k_s/L is fixed with $Re_L \rightarrow \infty$, giving that $C_D = f_1(k_s/L)$, $\delta_L/L = f_2(k_s/L)$ independent of Re_L . This agrees qualitatively with Granville (1958) although somewhat different $C_D(k_s/L)$ is obtained owing to the present use of a wake function. Thus for a given k_s and $x = L$ location on a fully rough vehicle, the boundary layer thickness and the drag coefficient is invariant with unit Reynolds number U_∞/ν .

9:18AM M35.00007 Sonic eddy model of the turbulent boundary layer , ROBERT BREIDENTHAL, PAUL DINTILHAC, OWEN WILLIAMS, University of Washington — A model of the compressible turbulent boundary layer is proposed. It is based on the notion that turbulent transport by an eddy requires that information of nonsteady events propagates across the diameter of that eddy during one rotation period. The finite acoustic signaling speed then controls the turbulent fluxes. As a consequence, the fluxes are limited by the largest eddies that satisfies this requirement. Therefore "sonic eddies" with a rotational Mach number of about unity would determine the skin friction, which is predicted to vary inversely with Mach number. This sonic eddy model contrasts with conventional models that are based on the energy equation and variations in the density. The effect of density variations is known to be weak in free shear flows, and the sonic eddy model assumes the same for the boundary layer. In general, Mach number plays two simultaneous roles in compressible flow, one related to signaling and the other related to the energy equation. The predictions of the model are compared with experimental data and DNS results from the literature.

9:31AM M35.00008 Decay of passive scalar fluctuations in axisymmetric turbulence , KATSUNORI YOSHIMATSU, Nagoya Univ, PETER A. DAVIDSON, University of Cambridge, YUKIO KANEDA, Aichi Institute of Technology — Passive scalar fluctuations in axisymmetric Saffman turbulence are examined theoretically and numerically. Theoretical predictions are verified by direct numerical simulation (DNS). According to the DNS, self-similar decay of the turbulence and the persistency of the large-scale anisotropy are found for its fully developed turbulence. The DNS confirms the time-independence of the Corrsin integral.

9:44AM M35.00009 Characteristics of space-time energy spectra in turbulent Shear flows , TING WU, CHENGHUI GENG, SHIZHAO WANG, GUOWEI HE, Institute of Mechanics, Chinese Academy of Sciences, TURBULENCE TEAM — An energy spectrum over wavenumbers is preliminarily characterized by its mean and standard deviation. The mean is corresponding to the characteristic wavenumber at the center of mass of energy spectrum and the standard deviation corresponding to the bandwidth of energy spectrum. In the present study, we derive the exact expressions for the characteristic wavenumbers and the bandwidths of space-time energy spectra at fixed frequencies. The characteristic wavenumbers are used to calculate the phase velocities that bridge from temporal spectra to space-time spectra. The bandwidths are used to measure the well-known spectral broadening. It is shown that phase velocities alone are insufficient to determine the bandwidths of energy spectra. As a result, Taylor's frozen-flow model and Kraichnan and Tannekes' random-sweeping model predict the narrower bandwidths of energy spectra. Therefore, in addition to phase velocities, bandwidths are introduced to rescale the space-time energy spectra that are obtained from phase velocities, leading to the correct bandwidths of energy spectra. Existing data from direct numerical simulation of turbulent channel flows is used to validate this rescaling technique.

9:57AM M35.00010 Irreversibility-inversions in 2D turbulence, ANDREW BRAGG, Duke University, FILIPPO DE LILLO, GUIDO BOFFETTA, Universit di Torino — We consider a recent theoretical prediction that for inertial particles in 2D turbulence, the nature of the irreversibility of their pair dispersion inverts when the particle inertia exceeds a certain value (Bragg et al., Phys. Fluids 28, 013305, 2016). In particular, when the particle Stokes number, St , is below a certain value, the forward-in-time (FIT) dispersion should be faster than the backward-in-time (BIT) dispersion, but for St above this value, this should invert so that BIT becomes faster than FIT dispersion. This non-trivial behavior arises because of the competition between two physically distinct irreversibility mechanisms that operate in different regimes of St . In 3D turbulence, both mechanisms act to produce faster BIT than FIT dispersion, but in 2D, the two mechanisms have opposite effects because of the inverse energy cascade in the turbulent velocity field. We supplement the qualitative argument given by Bragg et al. (Phys. Fluids 28, 013305, 2016) by deriving quantitative predictions of this effect in the short-time dispersion limit. These predictions are then confirmed by results of inertial particle dispersion in a direct numerical simulation of 2D turbulence.

Tuesday, November 22, 2016 8:00AM - 10:10AM –

Session M36 Drops: Wall-bound Portland Ballroom 251 - Brian Spencer, State University of NY - Buffalo

8:00AM M36.00001 Corner wetting during the vapor-liquid-solid growth of faceted nanowires, BRIAN SPENCER, University at Buffalo, STEPHEN DAVIS, Northwestern University — We consider the corner wetting of liquid drops in the context of vapor-liquid-solid growth of nanowires. Specifically, we construct numerical solutions for the equilibrium shape of a liquid drop on top of a faceted nanowire by solving the Laplace-Young equation with a free boundary determined by mixed boundary conditions. A key result for nanowire growth is that for a range of contact angles there is no equilibrium drop shape that completely wets the corner of the faceted nanowire. Based on our numerical solutions we determine the scaling behavior for the singular surface behavior near corners of the nanowire in terms of the Young contact angle and drop volume.

8:13AM M36.00002 Measuring static and dynamic contact angles using a liquid needle., RAYMOND SANEDRIN, KRUSS USA, MING JIN, DANIEL FRESE, CARSTEN SCHEITHAUER, THOMAS WILLERS, KRUSS GmbH — The optical determination of static and advancing contact angle is made on drops applied or extended, respectively, onto a substrate through the use of thin solid needles. Although this method has been used extensively, this method of dosing can be time consuming, cumbersome and if not meticulously performed can lead to erroneous contact angle results. Herein, we present an alternative way of applying drops onto substrates using a small liquid jet, which is produced by a liquid pressure dosing system acting as a “liquid needle.” A comparative static contact angle study on 14 different surfaces with two different liquids were performed utilizing two different ways of dosing: the conventional solid and a novel liquid needle based technique. We found, for all but one sample, that the obtained results were highly comparable. Observed differences can be explained by the characteristics of either way of dosing. In addition, we used the liquid pressure based dosing system for optical advancing contact angle measurement on two different samples. The liquid needle based method facilitates the expansion of a drop from 0.1 to 22 μL within less than 1.2 seconds, which provided constant contact angle versus drop base diameter curves. The obtained results were highly comparable with dynamic Wilhelmy contact angle measurements.

8:26AM M36.00003 Static 2D solutions for the profiles of liquids on rigid substrates, including the special case of droplets with finite-length precursor films¹, JUAN GOMBA, CIFICEN (UNCPBA-CICPBA-CONICET), Pinto 399, 7000 Tandil, Argentina , CARLOS A. PERAZZO, (Dept. de Física y Química, U. Favaloro, Buenos Aires, Argentina), J. R. MAC INTYRE, CIFICEN (UNCPBA-CICPBA-CONICET), Pinto 399, 7000 Tandil, Argentina — We present analytical solutions for the shape of static bidimensional profiles of a liquid resting on a substrate under partial-wetting conditions imposed by means of a two-term disjoiningconjoining pressure. In contrast with previous works where we studied the shape of droplets surrounded by infinite precursor films [Gomba, J. M. & Homsy, G. M *Langmuir* 25, 5684 (2009)], [Gomba, J. M. & Perazzo, C. A *Phys. Rev. E* 86, 056310 (2012)], [Mac Intyre, J. R. & Gomba, J. M. & Perazzo, C. A *J. Eng. Math.*, in press (2016)], here we do not impose restrictions on the shape of the profile. We show that for quite general disjoiningconjoining pressure, the free surface can adopt only 5 nontrivial static patterns. In particular we find solutions when the height goes to zero which describe satisfactorily the profile of a finite amount of fluid deposited on a substrate. Interestingly, one of the solutions represents the shape of a droplet surrounded by a finite length precursor film. We make a parametric study and identify the regions where each solution can be found. We compare the solutions with the corresponding ones obtained by more complete models (where the hypothesis of the lubrication approximation is not strictly valid) and with axisymmetric s

¹Conicet PIP 299, PIP 356, ANCyT PICT 1707

8:39AM M36.00004 Effects of induced vibration modes on droplet sliding phenomena¹, JOSE EDUARDO MEJIA, JORGE ALVARADO, Texas AM University, CHUN-WEI YAO, Lamar University, DROPWISE CONDENSATION COLLABORATION, ENGINEERED SURFACES COLLABORATION — An analytical and experimental investigation has been undertaken to understand the effects of induced vibration modes on droplet sliding phenomena. A mathematical model has been postulated which is capable of estimating accurately droplet sliding angles when using hydrophobic and hydrophilic surfaces. The model, which takes into account equilibrium contact angle, contact angle hysteresis, and droplet volume, has been validated using experimental data. The model has been modified to be able to estimate droplet sliding angle when different modes of vibrations are imposed on the surfaces. Experimental results to date reveal that when resonance modes of vibrations are imposed, the droplet sliding angles decrease considerably. The results also indicate that the modified model can be used effectively to relate imposed resonance frequencies to the critical sliding angle of droplets.

¹LSAMP sponsored NSF Fellowship

8:52AM M36.00005 Capillary adhesion forces between flexible fibers, CAMILLE DUPRAT, LadHyX, Ecole polytechnique, SUZIE PROTIRE, CNRS - Institut Jean le Rond d'Alembert — We consider the capillary adhesion produced by a drop placed between two elastic fibers. We measure the force exerted by the drop as we vary the inter-fiber distance, and report two types of wet adhesion: a weak capillary adhesion, where a liquid drop bridges the fibers, and a strong elastocapillary adhesion where the liquid is spread between two collapsed fibers. The weak adhesion is characterized by a force that increases linearly with the liquid length. With flexible fibers, the force exerted by the drop can induce deformation and rapid collapse, or zipping, of the fibers. This zipping results in a sudden increase of the wetted length and a force that departs from the linear evolution. As the inter-fiber distance is subsequently increased, the liquid length decreases while the fibers deformation increases, and the force actually reaches a plateau, i.e. remains constant until unzipping, or detachment of the fibers occurs. We measure the value of this plateau, i.e. the maximal adhesion force, as we vary the drop volume and the fibers elasticity. We also show that flexibility extends capillary adhesion to inter-fiber distances impossible to reach with rigid fibers, while keeping a constant pull-out force characteristic of the elastocapillary coupling.

9:05AM M36.00006 Molecular dynamics analysis of a equilibrium nanoscale droplet on a solid surface with periodic roughness, YUMA FURUTA, Department of Mechanical Engineering, Osaka University, DONATAS SURBLYS, Institute of Phisical and Chemical Research, YASTAKA YAMAGUCHI, Department of Mechanical Engineering, Osaka University — Molecular dynamics simulations of the equilibrium wetting behavior of hemi-cylindrical argon droplets on solid surfaces with a periodic roughness were carried out. The rough solid surface is located at the bottom of the calculation cell with periodic boundary conditions in surface lateral directions and mirror boundary condition at the top boundary. Similar to on a smooth surface, the change of the cosine of the droplet contact angle was linearly correlated to the potential well depth of the inter-atomic interaction between liquid and solid on a surface with a short roughness period while the correlation was deviated on one with a long roughness period. To further investigate this feature, solid-liquid, solid-vapor interfacial free energies per unit projected area of solid surface were evaluated by using the thermodynamic integration method in independent quasi-one-dimensional simulation systems with a liquid-solid interface or vapor-solid interface on various rough solid surfaces at a constant pressure. The cosine of the apparent contact angles estimated from the density profile of the droplet systems corresponded well with ones calculated from Young's equation using the interfacial energies evaluated in the quasi-one dimensional systems.

9:18AM M36.00007 Shapes of randomly placed droplets, MAHESH PANCHAGNULA, NACHIKETA JANARDAN, SRI VALLABHA DEEVI, Indian Inst of Tech-Madras — Surface characterization is essential for many industrial applications. Surface defects result in a range of contact angles, which lead to Contact Angle Hysteresis (CAH). We use shapes of randomly shaped drops on surfaces to study the family of shapes that may result from CAH. We image the triple line from these drops and extract additional information related to local contact angles as well as curvatures from these images. We perform a generalized extreme value analysis (GEV) on this microscopic contact angle data. From this analysis, we predict a range for extreme contact angles that are possible for a sessile drop. We have also measured the macroscopic advancing and receding contact angles using a Goniometer. From the extreme values of the contact line curvature, we estimate the pinning stress distribution responsible for the random shapes. It is seen that this range follows the same trend as the macroscopic CAH measured using a Goniometer, and can be used as a method of characterizing the surface.

9:31AM M36.00008 Three dimensional force balance of asymmetric droplets¹, YESEUL KIM, SU JIN LIM, SKKU Advanced Institute of Nanotechnology (SAINT), Sungkyunkwan University, KUN CHO, School of Advanced Materials Science and Engineering, Sungkyunkwan University, BYUNG MOOK WEON², School of Advanced Materials Science and Engineering, SKKU Advanced Institute of Nanotechnology (SAINT), Sungkyunkwan University — An equilibrium contact angle of a droplet is determined by a horizontal force balance among vapor, liquid, and solid, which is known as Young's law. Conventional wetting law is valid only for axis-symmetric droplets, whereas real droplets are often asymmetric. Here we show that three-dimensional geometry must be considered for a force balance for asymmetric droplets. By visualizing asymmetric droplets placed on a free-standing membrane in air with X-ray microscopy, we are able to identify that force balances in one side and in other side control pinning behaviors during evaporation of droplets. We find that X-ray microscopy is powerful for realizing the three-dimensional force balance, which would be essential in interpretation and manipulation of wetting, spreading, and drying dynamics for asymmetric droplets.

¹This research was supported by Basic Science Research Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Education (NRF-2016R1D1A1B01007133).

²E-mail: bmweon@skku.edu

9:44AM M36.00009 Nanodrop contact angles from molecular dynamics simulations, SRIKANTH RAVIPATI, BENJAMIN AYMARD, PETR YATSYSHIN, Complex Multiscale Systems Group, Department of Chemical Engineering, Imperial College London, AMPARO GALINDO, Centre for Process Systems Engineering Group, Department of Chemical Engineering, Imperial College London, SERAFIM KALLIADASIS, Complex Multiscale Systems Group, Department of Chemical Engineering, Imperial College London — The contact angle between three phases being in thermodynamic equilibrium is highly sensitive to the nature of the intermolecular forces as well as to various fluctuation effects. Determining the Young contact angle of a sessile drop sitting on a substrate from molecular dynamics (MD) simulations is a highly non-trivial task. Most commonly employed methods for finding droplet contact angles from MD simulation data either require large numbers of particles or are system-dependent. We propose a systematic geometry based methodology for extracting the contact angle from simulated sessile droplets by analysing an appropriately coarse-grained density field. To demonstrate the method, we consider Lennard-Jones (LJ) and SPC/E water nanodroplets of different sizes sitting on planar LJ walls. Our results are in good agreement with Young contact angle values computed employing test-area perturbation method.

9:57AM M36.00010 Footprint Geometry and Sessile Drop Resonance, CHUN-TI CHANG¹, National Taiwan University, SUSAN DANIEL, PAUL H. STEEN, Cornell Universitiy — How does a sessile drop resonate if its footprint is square (square drop)? In this talk, we discuss the two distinct families of observed modes in our experiments. One family (spherical modes) is identified with the natural modes of capillary spherical caps, and the other (grid modes) with Faraday waves on a square bath (square Faraday waves). A square drop exhibits grid or spherical modes depending on its volume, and the two families of modes arise depending on how wavenumber selection of footprint geometry and capillarity compete. For square drops, a dominant effect of footprint constraint leads to grid modes which are constrained response; otherwise the drops exhibit spherical modes, the characteristic of sessile drops on flat plates.

¹Chun-Ti Chang takes his new position at National Taiwan University on Aug. 15th, 2016. Until then, Chun-Ti Chang is affiliated with Technical University Dortmund, Germany

Tuesday, November 22, 2016 8:00AM - 9:57AM –
Session M37 Drops: Complex Fluids Portland Ballroom 252 - Carl D. Meinhart, University of California Santa Barbara

8:00AM M37.00001 Dynamics and shapes of ferrofluid drops under spatially uniform magnetic fields, PAYAM ROWGHANIAN, FRIEDHELM SERWANE, DAVID KEALHOFER, CARL D. MEINHART, OTGER CAMPAS, University of California Santa Barbara — We study the shape and dynamics of a Newtonian ferrofluid drop immersed in a Newtonian and non-magnetic viscous fluid under the action of a uniform external magnetic field. We obtain the exact equilibrium drop shapes for arbitrary ferrofluids which describe unexplained previous experiments, characterize the extent of deviations of the exact shape from the commonly assumed ellipsoidal shape, and analyze the smoothness of highly curved tips in elongated drops. We present a comprehensive study of drop deformation for a Langevin ferrofluid. Using a computational scheme that allows fast and accurate simulations of ferrofluid drop dynamics, we show that the dynamics of drop deformation by an applied magnetic field is described up to a numerical factor by the same time scale as drop relaxation in the absence of any magnetic field. The numerical factor depends on the ratio of viscosities and the ratio of magnetic to capillary stresses, but is independent of the nature of the ferrofluid in most practical cases. Finally, we use the shape and dynamics of the magnetic drops to measure the rheology of complex fluids.

8:13AM M37.00002 Pattern formation and temporal undulations of plane magnetic droplet¹, CHAMKOR SINGH, Indian Institute of Technology Kharagpur India, ARUP KUMAR DAS, Indian Institute of Technology Roorkee India, PRASANTA KUMAR DAS, Indian Institute of Technology Kharagpur India — In this study, we numerically investigate the time-dependent response of a ferrofluid droplet under an impulsively applied uniform magnetic field in a zero gravity environment. It is identified that two characteristic non-dimensional groups, namely, the Laplace number La and the magnetic Bond number Bo_m , primarily influence the response of the droplet. It is found that the nature of the time response can be either monotonic or undulating depending on the parameters. The transition between the two is smooth. In addition to the previously well-known regimes of elliptic and acicular ferrofluid droplet shapes, a new regime on the $La - Bo_m$ plane is found where we observe some unique bifurcating patterns at the poles of the droplet. The temporal aperiodic to periodic mode transition on the $La - Bo_m$ plane is found to be governed by La and the spatial droplet deformation and its final equilibrium configuration is found to be governed by Bo_m . The mechanism behind the elliptic to non-elliptic or elliptic to bifurcated shape transitions is discussed.

¹The authors gratefully acknowledge the support of the Council of Scientific and Industrial Research, New Delhi, India, for the present work.

8:26AM M37.00003 Magnetophoretic interaction of ferrofluid droplets in a rotating magnetic field, MINGFENG QIU, Department of Mathematics, University of British Columbia, SHAHRIAR AFKHAM, Department of Mathematical Sciences, New Jersey Institute of Technology, CHING-YAO CHEN, Department of Mechanical Engineering, National Chiao Tung University, JAMES FENG, Departments of Chemical & Biological Engineering and Mathematics, University of British Columbia — Recent experiments have discovered a mode of planetary motion of a pair of ferrofluid droplets in a rotating magnetic field. It consists of the self-spin of individual droplets and the global revolution of the pair with a phase lag from the rotating field. This talk describes a volume-of-fluid simulation that explores this phenomenon. By studying the magnetic and hydrodynamic interactions between the droplets, we determine the time scale of the planetary motion under different operating conditions. The numerical results are compared to predictions using a simple dipole interaction model and the experiments. Finally we simulate the motion of a multiple-droplet chain in a rotating field, and compare the results to experimental observations that the drops assemble into a regular and compact array that rotates with the field with a phase lag.

8:39AM M37.00004 A Theory of Shape-Shifting Droplets, PIERRE HAAS, RAYMOND GOLDSTEIN, STOYAN SMOUKOV, University of Cambridge, NIKOLAI DENKOV, University of Sofia — Recent observations of cooled oil emulsion droplets uncovered a remarkable array of shape transformations¹: the initially spherical droplets flatten into polygonal shapes, first hexagons, then triangles or quadrilaterals that ultimately grow thin protrusions from their corners. These transformations are driven by a partial phase transition of the bulk liquid phase. In this talk, we explore theoretically the simplest geometric competition between this phase transition and surface tension in planar polygons. We recover the experimental sequence of shapes and predict shape statistics in qualitative agreement with experiments. Extending the model to capture some of the three-dimensional structure of the droplets, we analyse the topological transition of droplet puncture observed in experiments.

¹N. Denkov, S. Tcholakova, I. Lesov, D. Cholakova, and S. K. Smoukov, Self-shaping of oil droplets via the formation of intermediate rotator phases upon cooling, *Nature* **528**, 392 (2015)

8:52AM M37.00005 Dynamics of spreading thixotropic droplets¹, ARAN UPPAL, RICHARD CRASTER, OMAR MATAR, Imperial College London — Thixotropy has become of increasing interest for a variety of applications in recent years. The lubrication approximation has been often used in the study of such fluids, especially in the presence of a free surface. The lubrication approximation aims to remove the explicit depth dependence from the resulting evolution equations by utilising the naturally occurring small aspect ratio. However, this is not possible with the inclusion of a structure parameter to describe the thixotropic behaviour. Thus, we consider a range of closures to simplify the evolution equations and compare against the full simulation results.

¹EPSRC UK Centre for Doctoral Training

9:05AM M37.00006 Drop dynamics on a stretched viscoelastic filament: An experimental study¹, JORGE PEIXINHO, MARIE-CHARLOTTE RENOULT, OLIVIER CRUMEYROLLE, INNOCENT MUTABAZI, Normandie Univ., UNIHAVRE, CNRS, LOMC, Le Havre, France — Capillary pressure can destabilize a thin liquid filament during breakup into a succession of drops. Besides, the addition of a linear, high molecular weight, flexible and soluble polymer is enough to modify the morphology of this instability. In the time period preceding the breakup, the development of beads-on-a-string structures where drops are connected by thin threads is monitored. The drops dynamics involve drop formation, drop migration and drop coalescence. Experiments using a high-speed camera on stretched bridges of viscoelastic polymeric solutions were conducted for a range of viscosities and polymer concentrations. The rheological properties of the solutions are also quantified through conventional shear rheology and normal stress difference. The overall goal of this experimental investigation is to gain more insight into the formation and time evolution of the drops.

¹The project BIOENGINE is co-financed by the European Union with the European regional development fund and by the Normandie Regional Council

9:18AM M37.00007 Rolling, sliding, and sticking of viscoplastic xanthan gum solution drops on a superhydrophobic surface, MINYOUNG KIM, Korea Institute of Science and Technology, EUNGJUN LEE, DO HYUN KIM, Korea Advanced Institute of Science and Technology, RHOKYUN KWAK, Korea Institute of Science and Technology — Dynamics of Newtonian fluid on a non-wettable substrate have been reported, but those of non-Newtonian fluid, especially of viscoplastic fluid showing a yield stress, are not fully characterized yet. Here, we investigate three distinct behaviors of a viscoplastic drop (xanthan gum solution) -rolling, sliding, and sticking- on an inclined superhydrophobic surface with various inclined angles (1-24 degree) and xanthan gum concentrations (0.25-1.5 %). At a low concentration of xanthan gum (low yield stress) and/or a high inclined angle (high gravitational stress), the drop rolls down the surface as the gravitational stress exceeds the yield stress. As the concentration increases, and thus the yield stress exceeds the gravitational stress, the drop stays on the surface like a solid (sticking). However, if we adjust the gravitational stress to induce an adhesive failure between the xanthan gum drop and the surface (but still lower than the yield stress), the drop slides down the surface without rolling. To the best of our knowledge, this is the first direct characterization of the behavior of the viscoplastic drops on an inclined surface considering gravitational stress, yield stress, and adhesive failure.

9:31AM M37.00008 Coalescence dynamics of a Non-Newtonian drop in a Hele-Shaw cell¹, PANAGIOTA ANGELI, VICTOR VOULGAROPOULOS, MAXIME CHINAUD, Department of Chemical Engineering, University College London, Torrington Place, London WC1E 7JE, UK — In this study, the quasi-2D coalescence of an aqueous droplet with a flat interface is studied. The two immiscible liquids used in the coalescence experiments are a water/glycerol mixture and low viscosity oil. Experiments are conducted in a Hele-Shaw cell with an aspect ratio of 14 by 8 by 0.125 cm while time resolved velocity fields are obtained by using high-speed bright field Particle Image Velocimetry measurements. The aim of the present study is to investigate the effect of a polymer (i.e. xanthan gum), which results in a shear-thinning behaviour of the aqueous phase. It was found that the time evolution of the neck at the initial stages of coalescence follows a linear trend, which suggests that the shear-thinning behaviour at the neck region at this stage of coalescence can be considered quasi-constant in time. In addition, velocity and vorticity fields are computed inside the coalescing droplet and the bulk homophase. It is found that two pair of counter rotating vortices are generated just after the rupture of the film which separates the drop from the homophase. Furthermore, the polymer addition reduced the magnitude of the corresponding vorticity peaks in the drop and slowed down the coalescence dynamics.

¹Project funded under the UK Engineering and Physical Sciences Research Council (EPSRC) Programme Grant MEMPHIS

9:44AM M37.00009 Regimes in secondary atomization of shear thinning inelastic drops, VARUN KULKARNI, JONATHAN ROCHA, PAUL SOJKA, Purdue University — Secondary atomization plays a key role in the determining the eventual drop size distribution of a spray from an injector. Studies until recently have primarily focused on drops made of Newtonian or viscoelastic fluids. However, little is known about the breakup of single drop made of a shear thinning fluid subjected to cross flow of air. In this study we attempt to classify, experimentally, the regimes of breakup and morphology of an inelastic, shear thinning drop. Six different CMC water/glycerine based solutions are tested. The morphological patterns are classified on a $We/Oh^2 - 1/Oh^2$ plot using the zero shear rate viscosity where We denotes the Weber number and Oh is the Ohnesorge number. The regime boundaries for the various breakup patterns are juxtaposed with their Newtonian counterparts to highlight the differences. We observe a dependence on the Carreau number, Cu and power law index, n . The results converge to the Newtonian limit for particular cases of Cu and n .

Tuesday, November 22, 2016 8:00AM - 10:10AM – Session M38 Flow Instability: Richtmyer-Meshkov I Portland Ballroom 255 - Kathy Prestridge, Los Alamos National Laboratory

8:00AM M38.00001 Oscillations of a standing shock in the Richtmyer-Meshkov instability (II)¹, KARNIG MIKAElian, Lawrence Livermore National laboratory — In a typical Richtmyer-Meshkov experiment a fast moving flat shock strikes a stationary perturbed interface between fluids A and B creating a transmitted and a reflected shock, both of which are perturbed. We propose shock tube experiments in which the reflected shock is stationary in the laboratory. Such a standing shock undergoes well known damped oscillations. We present the conditions required for producing such a standing shock wave which greatly facilitates the measurement of the oscillations and their rate of damping.

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

8:13AM M38.00002 Dynamics of Richtmyer-Meshkov (RM) mixing with reshock, SWATHI MULA, STUART CRAIG, KATHY PRESTRIDGE, Los Alamos National Laboratory — Variable density mixing plays a very important role in a number of applications, including inertial confinement fusion, supernovae, and supersonic combustion ramjet engines. To better understand the dynamics of variable density mixing, experiments are developed at the Vertical Shock Tube (VST) facility at Los Alamos National Laboratory. At this facility, an initially perturbed density interface (air-SF6, Atwood number = 0.6) is impulsively accelerated by a low Mach shock wave (Mach <3), which induces Richtmyer-Meshkov (RM) mixing of the two fluids. Initial perturbations on the air-SF6 interface are generated by an oscillating flapper that initially separates the two fluids. The time evolution of RM mixing is studied by way of simultaneous density and velocity measurements using Planar Laser Induced Fluorescence (PLIF) and Particle Image Velocimetry (PIV) techniques. For two separate initial conditions, the measurements capture the air-SF6 interface, at multiple time locations, before and after the passage of shock and reshock at Mach = 1.3. At each time location, multiple instantaneous shots are acquired. From these measurements, we study the evolution of RM instability along with the dependence of mixing flow features (post-shock and reshock) on the initial conditions.

8:26AM M38.00003 Circulation in blast driven instabilities, MARC HENRY DE FRAHAN, ERIC JOHNSEN, Univ of Michigan - Ann Arbor — Mixing in many natural phenomena (e.g. supernova collapse) and engineering applications (e.g. inertial confinement fusion) is often initiated through hydrodynamic instabilities. Explosions in these systems give rise to blast waves which can interact with perturbations at interfaces between different fluids. Blast waves are formed by a shock followed by a rarefaction. This wave profile leads to complex time histories of interface acceleration. In addition to the instabilities induced by the acceleration field, the rarefaction from the blast wave decompresses the material at the interface, further increasing the perturbation growth. After the passage of the wave, circulation circulation generated by the blast wave through baroclinic vorticity continues to act upon the interface. In this talk, we provide scaling laws for the circulation and amplitude growth induced by the blast wave. Numerical simulations of the multifluid Euler equations solved using a high-order accurate Discontinuous Galerkin method are used to validate the theoretical results.

8:39AM M38.00004 Mixing and Turbulence Statistics in an Inclined Interface Richtmyer-Meshkov Instability¹, AKSHAY SUBRAMANIAM, SANJIVA LELE, Stanford Univ — The interaction of a Mach 1.55 shockwave with a nominally inclined interface is considered. Unlike the classical Richtmyer-Meshkov problem, the interface evolution is non-linear from early time and large highly correlated vortical structures are observed even after reshock. The simulations target the experiment of McFarland et. al. (2014). Simulations are performed using the Miranda code (Cook et. al., 2005) that uses high-order spectral-like numerics (Lele, 1992). Results from multiple grid resolutions up to 4 billion grid points establish grid convergence. Comparisons to the experiments show that the simulations adequately capture the physics of the problem. Analysis of the data from the simulations based on variable density turbulence equations in the Favre averaged form will be presented. Statistics of unclosed terms in the variable density RANS equations will also be presented and compared to standard closure models. It is observed that the Reynolds Stresses have a non-monotonic return to isotropy after reshock and that compressibility effects are important long after reshock. The effect of numerics are also quantified and presented.

¹Computer time for this work was provided by NSF PRAC award “Multi-material turbulent mixing” on the Blue Waters system.

8:52AM M38.00005 Emergence of power-law scalings in shock-driven mixing transition¹, PETER VOROBIEFF, PATRICK WAYNE, DELL OLMSTEAD, DYLAN SIMONS, C. RANDALL TRUMAN, University of New Mexico, SANJAY KUMAR, Indian Institute of Technology - Kanpur — We present an experimental study of transition to turbulence due to shock-driven instability evolving on an initially cylindrical, diffuse density interface between air and a mixture of sulfur hexafluoride (SF_6) and acetone. The plane of the shock is at an initial angle θ with the axis of the heavy-gas cylinder. We present the cases of planar normal ($\theta = 0$) and oblique ($\theta = 20^\circ$) shock interaction with the initial conditions. Flow is visualized in two perpendicular planes with planar laser-induced fluorescence (PLIF) triggered in acetone with a pulsed ultraviolet laser. Statistics of the flow are characterized in terms of the second-order structure function of the PLIF intensity. As instabilities in the flow evolve, the structure functions begin to develop power-law scalings, at late times manifesting over a range of scales spanning more than two orders of magnitude. We discuss the effects of the initial conditions on the emergence of these scalings, comparing the fully three-dimensional case (oblique shock interaction) with the quasi-two-dimensional case (planar normal shock interaction). We also discuss the flow anisotropy apparent in statistical differences in data from the two visualization planes.

¹This work is funded by NNSA grant DE-NA0002913

9:05AM M38.00006 Time-resolved particle image velocimetry measurements of the 3D single-mode Richtmyer-Meshkov instability, QIAN XU, VITALIY V. KRIVETS, EVEREST G. SEWELL, JEFFREY W. JACOBS, Univ of Arizona — A vertical shock tube is used to perform experiments on the single-mode three-dimensional Richtmyer-Meshkov Instability (RMI). The light gas (Air) and the heavy gas (SF_6) enter from the top and the bottom of the shock tube driven section to form the interface. The initial perturbation is then generated by oscillating the gases vertically. Both gases are seeded with particles generated through vaporizing propylene glycol. An incident shock wave ($M \approx 1.2$) impacts the interface to create an impulsive acceleration. The seeded particles are illuminated by a dual cavity 75W, Nd: YLF laser. Three high-speed CMOS cameras record time sequences of image pairs at a rate of 2 kHz. The initial perturbation used is that of a single, square-mode perturbation with either a single spike or a single bubble positioned at the center of the shock tube. The full time dependent velocity field is obtained allowing the determination of the circulation versus time. In addition, the evolution of time dependent amplitude is also determined. The results are compared with PIV measurements from previous two-dimensional single mode experiments along with PLIF measurements from previous three-dimensional single mode experiments.

9:18AM M38.00007 Simulations of the Richtmyer-Meshkov Instability with experimentally measured volumetric initial conditions, KEVIN FERGUSON, EVEREST SEWELL, VITALIY KRIVETS, The University of Arizona, JEFFREY GREENOUGH, Lawrence Livermore National Laboratory, JEFFREY JACOBS, The University of Arizona — Initial conditions for the Richtmyer-Meshkov instability (RMI) are measured in three dimensions in the University of Arizona Vertical Shock Tube using a moving magnet galvanometer system. The resulting volumetric data is used as initial conditions for the simulation of the RMI using ARES at Lawrence-Livermore National Laboratory (LLNL). The heavy gas is sulfur hexafluoride (SF_6), and the light gas is air. The perturbations are generated by harmonically oscillating the gasses vertically using two loudspeakers mounted to the shock tube which cause Faraday resonance, producing a random short wavelength perturbation on the interface. Planar Mie scattering is used to illuminate the flow field through the addition of propylene glycol particles seeded in the heavy gas. An $M=1.2$ shock impulsively accelerates the interface, initiating instability growth. Images of the initial condition and instability growth are captured at a rate of 6 kHz using high speed cameras. Comparisons between experimental and simulation results, mixing diagnostics, and mixing zone growth are presented.

9:31AM M38.00008 Simulations and experiments of ejecta generation in twice-shocked metals¹, VARAD KARKHANIS, PRAVEEN RAMAPRABHU, University of North Carolina, Charlotte, WILLIAM BUTTLER, JAMES HAMMERBERG, FRANK CHERNE, MALCOLM ANDREWS, Los Alamos National Laboratory, New Mexico — Using continuum hydrodynamics embedded in the FLASH code, we model ejecta generation in recent target experiments [1], where a metallic surface was loaded by two successive shock waves. The experimental data were obtained from a two-shockwave, high-explosive tool at Los Alamos National Laboratory, capable of generating ejecta from a shocked tin surface in to a vacuum. In both simulations and experiment, linear growth is observed following the first shock event, while the second shock strikes a finite-amplitude interface leading to nonlinear growth. The timing of the second incident shock was varied systematically in our simulations to realize a finite-amplitude re-initialization of the RM instability driving the ejecta. We find the shape of the interface at the event of second shock is critical in determining the amount of ejecta, and thus must be used as an initial condition to evaluate subsequent ejected mass using a source model[2]. In particular, the agreement between simulations, experiments and the mass model is improved when shape effects associated with the interface at second shock are incorporated. [1] W. T. Buttler et al., J. Appl. Phys., 116 (2014). [2] F.J. Cherne et al., J. Appl. Phys., 118, 185901 (2015).

¹This work was supported in part by the (U.S.) Department of Energy (DOE) under Contract No. DE-AC52-06NA2-5396.

9:44AM M38.00009 Experimental Investigation of the Richtmyer-Meshkov Instability Through Simultaneous Measurements of Concentration and Velocity, DANIEL REESE, ALEX AMES, CHRIS NOBLE, JASON OAKLEY, DAVE ROTHAMER, RICCARDO BONAZZA, University of Wisconsin — The present work investigates the evolution of the Richtmyer-Meshkov instability through simultaneous measurements of concentration and velocity. 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 (Atwood number $A = 0.7$). The helium is seeded with acetone vapor for use in planar laser-induced fluorescence (PLIF), while each gas in the shear layer cross flow 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 $M = 1.57$ shock wave, the interface is imaged twice in close succession using a planar laser sheet containing both the second and fourth harmonic output (532 nm and 266 nm, respectively) of a dual-cavity Nd:YAG laser. Particle image pairs are captured on a dual-frame CCD camera, for use in particle image velocimetry (PIV), while PLIF images are corrected to show concentration. Velocity fields are obtained from particle images using the Insight 4G software package by TSI, and velocity field structure is investigated and compared against concentration images. Probability density functions (PDFs) and planar energy spectra (of both velocity fluctuations and concentration) are then calculated and results are discussed.

9:57AM M38.00010 Experimental investigation of the effect of multimodal inclined interface on Richtmyer-Meshkov instability evolution, MOHAMMAD MOHAGHAR, JOHN CARTER, BENJAMIN MUSCI, DEVESH RANJAN, Georgia Institute of Technology — In the Georgia Tech Shock Tube and Advanced Mixing Laboratory, the evolution of Richtmyer-Meshkov instability (RMI) which arises from two initial conditions, namely, a predominantly single mode, inclined interface between two gases, and a perturbed, multimodal, inclined interface are studied. The gas combination of nitrogen-acetone as light gas and carbon dioxide as heavy gas (Atwood number of 0.23) with an inclination angle of 80 degrees ($\eta/\lambda = 0.097$) was chosen in this set of experiments. The interface is visualized using planar laser diagnostics (simultaneous PLIF/PIV measurements), once impulsively accelerated by a Mach~1.55. The ensemble-averaged turbulence measurements of the density, velocity and density-velocity cross-statistics are used to investigate the effects of added secondary modes to the interface on the correlation between turbulence and mixing quantities.

Tuesday, November 22, 2016 8:00AM - 10:10AM –

Session M39 Bio: Bacterial Accumulation and Growth Portland Ballroom 256 - Michele Guala,
University of Minnesota

8:00AM M39.00001 Understanding Abiotic Triggers For Cyanobacteria Blooms in Lakes Using a Long Term In-situ Monitoring Research Station , ANNE WILKINSON, MIKI HONZO, CHRISTINE SALOMON, SHAHRAM MISSAGHI, MICHELE GUALA, University of Minnesota — Harmful Algal Blooms (HAB) are ubiquitous ecological and public health hazards. HAB are made up of potentially toxic freshwater cyanobacteria. The occurrences of toxic HAB are unpredictable and highly spatially/temporary variable in freshwater ecosystems. To study the abiotic triggers for toxic HAB, a research station has been deployed in a eutrophic lake from June-October 2016. This station provides hourly water quality profiles and meteorological (every 5 minutes) monitoring with real time access. Water quality monitoring is performed by an autonomously traversed sonde that provides chemical, physical and biological measurements; including phycocyanin, a light-absorbing pigment distinct to cyanobacteria. The research station is a sentinel for HAB accumulation, prompting focused HAB analysis, including: phytoplankton and toxin composition/concentration, and turbulent kinetic energy dissipation rates. We will discuss how mixing conditions, temperature stratification, light intensity, surface wind magnitude and energy dissipation mediate a)HAB formation/composition b)toxicity and c)cyanobacteria stratification. The results will help illuminate abiotic processes that trigger HAB accumulation/toxicity, which can direct timely toxic HAB prediction and prevention efforts.

8:13AM M39.00002 Clustering of floating particles in stratified turbulence , GUIDO BOFFETTA, FILIPPO DE LILLO, University of Torino, STEFANO MUSACCHIO, University of Nice Sophia Antipolis, ALESSANDRO SOZZA, University of Torino — We study the dynamics of small floating particles transported by stratified turbulence in presence of a mean linear density profile as a simple model for the confinement and the accumulation of plankton in the ocean. By means of extensive direct numerical simulations we investigate the statistical distribution of floaters as a function of the two dimensionless parameters of the problem. We find that vertical confinement of particles is mainly ruled by the degree of stratification, with a weak dependency on the particle properties. Conversely, small scale fractal clustering, typical of non-neutral particles in turbulence, depends on the particle relaxation time and is only weakly dependent on the flow stratification. The implications of our findings for the formation of thin phytoplankton layers are discussed.

8:26AM M39.00003 Bacterial finite-size effects for population expansion under flow , FEDERICO TOSCHI, FRANCESCA TESSER, JOS C.H. ZEEGERS, HERMAN J.H. CLERCX, LUC BRUNSVELD, Eindhoven Univ of Tech — For organisms living in a liquid ecosystem, flow and flow gradients have a dual role as they transport nutrient while, at the same time, dispersing the individuals. In absence of flow and under homogeneous conditions, the growth of a population towards an empty region is usually described by a reaction-diffusion equation. The effect of fluid flow is not yet well understood and the interplay between transport of individuals and growth opens a wide scenario of possible behaviors. In this work, we study experimentally the dynamics of non-motile *E. coli* bacteria colonies spreading inside rectangular channels, in PDMS microfluidic devices. By use of a fluorescent microscope we analyze the dynamics of the population density subjected to different co- and counter-flow conditions and shear rates. A simple model incorporating growth, dispersion and drift of finite size beads is able to explain the experimental findings. This indicates that models based on the Fisher-Kolmogorov-Petrovsky-Piscounov equation (FKPP) may have to be supplemented with bacterial finite-size effects in order to be able to accurately reproduce experimental results for population spatial growth.

8:39AM M39.00004 Dynamics of water uptake in spreading bacterial colonies , C. NADIR KAPLAN, L. MAHADEVAN, Harvard University — Bacteria can colonize a moist, nutrient-rich surface by secreting osmolytes to recruit water from the underlying substrate. We consider the outermost region of an expanding *Escherichia coli* biofilm, where the rim width is set by the cell growth rate and the colony expansion speed. Based on the hypothesis that sliding due to the mechanical contact between cells governs their speed, we model the interplay between the flow of cells and the water uptake via osmolyte production. This allows us to determine the front expansion speed and the non-uniform biofilm thickness, in agreement with experiments.

8:52AM M39.00005 ABSTRACT WITHDRAWN –

9:05AM M39.00006 Bacterial accumulation in viscosity gradients , NICOLAS WAISBORD, JEFFREY GUASTO, Tufts University — Cell motility is greatly modified by fluid rheology. In particular, the physical environments in which cells function, are often characterized by gradients of viscous biopolymers, such as mucus and extracellular matrix, which impact processes ranging from reproduction to digestion to biofilm formation. To understand how spatial heterogeneity of fluid rheology affects the motility and transport of swimming cells, we use hydrogel microfluidic devices to generate viscosity gradients in a simple, polymeric, Newtonian fluid. Using video microscopy, we characterize the random walk motility patterns of model bacteria (*Bacillus subtilis*), showing that both wild-type ('run-and-tumble') cells and smooth-swimming mutants accumulate in the viscous region of the fluid. Through statistical analysis of individual cell trajectories and body kinematics in both homogeneous and heterogeneous viscous environments, we discriminate passive, physical effects from active sensing processes to explain the observed cell accumulation at the ensemble level.

9:18AM M39.00007 Investigation of cyanobacteria in a controlled hyperbolic straining flow¹ , FARZAN AKBARIDOUST, JIMMY PHILIP, IVAN MARUSIC, Univ of Melbourne — Here we report a systematic study on the effect of straining flow on cyanobacteria, which are a cause of significant water contamination issues worldwide. We focus on the species *Anabaena Circinalis*. A micro-cross channel equipped with two online computer-controlled on-chip membrane valves was designed and fabricated using standard soft-lithography. The device produces a hyperbolic straining flow on a micron-scaled region similar to G. I. Taylor's four-roll mill at larger scale. It was used to investigate the behaviour of a single filament of cyanobacteria in a crowded medium under an increasing uniform strain rate flow. The velocity field and the resulting uniform strain-rate was measured in the absence of bacteria filaments using micro-PIV. A large number of single filaments of bacteria were trapped and exposed to stain-rates over 2 to 15 s⁻¹. Previous studies have reported anecdotal evidence of suspected mechanical damage to *Anabaena Circinalis* for strain rates considerably lower than the maximum values studied here. In our case, no mechanical damage was observed.

¹This work was performed in part at the Melbourne Centre for Nanofabrication (MCN) in the Victorian Node of the Australian National Fabrication Facility (ANFF)

9:31AM M39.00008 Measurements of fluid transport by controllable vertical migrations of plankton¹, ISABEL A. HOUGHTON, JOHN O. DABIRI, Stanford University — Diel vertical migration of zooplankton has been proposed to be a significant contributor to local and possibly large-scale fluid transport in the ocean. However, studies of this problem to date have been limited to order-of-magnitude estimates based on first principles and a small number of field observations. In this work, we leverage the phototactic behavior of zooplankton to stimulate controllable vertical migrations in the laboratory and to study the associated fluid transport and mixing. Building upon a previous prototype system, a laser guidance system induces vertical swimming of brine shrimp (*Artemia salina*) in a 2.1 meter tall, density-stratified water tank. The animal swimming speed and spacing during the controlled vertical migration is characterized with video analysis. A schlieren imaging system is utilized to visualize density perturbations to a stable stratification for quantification of fluid displacement length scales and restratification timescales. These experiments can add to our understanding of the dynamics of active particles in stratified flows.

¹NSF and US-Israel Binational Science Foundation

9:44AM M39.00009 Fluid dynamics of two-dimensional pollination in *Ruppia maritima*¹, NAGA MUSUNURI, DANIEL BUNKER, New Jersey Institute of Technology, SUSAN PELL, United States Botanic Garden, FISCHER PELL, PUSHPENDRA SINGH, New Jersey Institute of Technology — The aim of this work is to understand the physics underlying the mechanisms of two-dimensional aquatic pollen dispersal, known as hydrophily. We observed two mechanisms by which the pollen released from male inflorescences of *Ruppia maritima* is adsorbed on a water surface: (i) inflorescences rise above the surface and after they mature their pollen mass falls onto the surface as clumps and disperses on the surface; (ii) inflorescences remain below the surface and produce air bubbles which carry their pollen mass to the surface where it disperses. In both cases dispersed pollen masses combined under the action of capillary forces to form pollen rafts. This increases the probability of pollination since the capillary force on a pollen raft towards a stigma is much larger than on a single pollen grain. The presence of a trace amount of surfactant can disrupt the pollination process so that the pollen is not transported or captured on the water surface.

¹National Science Foundation

9:57AM M39.00010 Stochastic cycle selection in active flow networks, FRANCIS WOODHOUSE, University of Cambridge, ADEN FORROW, Massachusetts Institute of Technology, JOANNA FAWCETT, The University of Western Australia, JORN DUNKEL, Massachusetts Institute of Technology — Active biological flow networks pervade nature and span a wide range of scales, from arterial blood vessels and bronchial mucus transport in humans to bacterial flow through porous media or plasmodial shuttle streaming in slime molds. Despite their ubiquity, little is known about the self-organization principles that govern flow statistics in such non-equilibrium networks. By connecting concepts from lattice field theory, graph theory and transition rate theory, we show how topology controls dynamics in a generic model for actively driven flow on a network. Through theoretical and numerical analysis we identify symmetry-based rules to classify and predict the selection statistics of complex flow cycles from the network topology. Our conceptual framework is applicable to a broad class of biological and non-biological far-from-equilibrium networks, including actively controlled information flows, and establishes a new correspondence between active flow networks and generalized ice-type models.

Tuesday, November 22, 2016 8:00AM - 10:10AM –
Session M40 Porous Media Flows: CO₂ Sequestration and Convection Portland Ballroom
253-258-254-257 - Farzan Kazemifar, University of Notre Dame

8:00AM M40.00001 Experimental quantification of pore-scale flow of water and liquid CO₂ in 2D heterogeneous porous micromodels at reservoir conditions¹, YAOFA LI, FARZAN KAZEMIFAR, GIANLUCA BLOIS, KENNETH CHRISTENSEN, University of Notre Dame — Pore-scale flow interactions between water and supercritical CO₂ is relevant to large-scale geologic sequestration of CO₂. Recent studies have provided evidence of strong instabilities at the meniscus resulting in burst events and onset of inertial effects. This supports the notion that pore-scale physics cannot be captured by Darcian models and unsteady events play a defining role in CO₂ transport/trapping processes and such burst events may generate pressure fluctuations that can be linked to micro-seismic events in the pore structure. To this end, the pore-scale flow of water and liquid/supercritical CO₂ is investigated under reservoir-relevant conditions in 2D heterogeneous porous micro-models that reflect the complexity of a real sandstone. Fluorescent microscopy and micro-PIV are complemented by a fast differential pressure transmitter, allowing for simultaneous quantification of the flow field within and the instantaneous pressure drop across the micromodels. A number of CO₂ invasion patterns and corresponding pressure drop variations are observed over a range of wettability conditions, yielding a more comprehensive picture of the CO₂ drainage processes.

¹This work was primarily supported as part of the Center for Geologic Storage of CO₂, an EFRC funded by the U.S. Department of Energy, Office of Science and partially supported by WPI-I2CNER based at Kyushu University, Japan.

8:13AM M40.00002 A new approach to the stability analysis of transient natural convection in porous media, NILS TILTON, Colorado School of Mines — Onset of natural convection due to transient diffusion in porous media has attracted considerable attention for its applications to CO₂ sequestration. Stability analyses typically investigate onset of convection using an initial value problem approach in which a perturbation is introduced to the concentration field at an initial time $t = t_p$. This leads to debate concerning physically appropriate perturbations, the critical time t_c for linear instability, and to the counter-intuitive notion of an optimal initial time t_p that maximizes perturbation growth. We propose a new approach in which transient diffusion is continuously perturbed by small variations in the porosity. With this approach, instability occurs immediately ($t_c = 0$) without violating any physical constraints, such that the concepts of initial time t_p and critical time t_c have less relevance. We argue that the onset time for nonlinear convection is a more physically relevant parameter, and show that it can be predicted using a simple asymptotic expansion. Using the expansion, we consider porosity perturbations that vary sinusoidally in the horizontal and vertical directions, and show there are optimal combinations of wavelengths that minimize the onset time of nonlinear convection.

8:26AM M40.00003 Modeling the convective stability of CO_2 sequestration by a discontinuous and unstably stratified density profile, TABER WANSTALL, LAYACHI HADJI, The University of Alabama — The convective stability associated with carbon sequestration is modeled by adopting an unstably stratified basic profile having a step function density with top heavy carbon saturated layer overlying a lighter carbon free layer. The model takes into account the anisotropy in both permeability and carbon dioxide diffusion, and chemical reactions between the CO_2 rich brine and host mineralogy. We carry out a linear stability analysis to derive the instability threshold parameters for a variety of CO_2 boundary conditions. We solve for the minimum thickness of the carbon-rich layer at which convection sets in and quantify how its value is influenced by diffusion, anisotropy, permeability, reaction and type of boundary conditions. The discontinuity leads to convective concentration contours that have the shape of an asymmetric lens which we quantify by deriving and making use of the CO_2 flux expressions at the interface. The linear problem is extended to the nonlinear regime, the analysis of which leads to the determination of a uniformly valid super critical steady solution.

8:39AM M40.00004 Density driven convection with dissolution in porous media: experiment, simulation and linear stability analysis, XUHUI MENG, Huazhong University of Science and Technology, XIAOFAN YANG, Pacific Northwest National Laboratory, ZHAOLI GUO, Huazhong University of Science and Technology — Geological storage of the CO_2 in subsurface saline aquifers is a promising way to reduce CO_2 emissions. During this process, CO_2 first dissolves into pure brine. Then the acidic and denser mixture falls down under the gravity and reacts with the rock. In the present work, a microfluidic experiment is conducted to investigate the density-driven convection with dissolution in porous media. Moreover, the linear stability analysis and numerical simulations are further performed to investigate the interfacial instability. The results demonstrate that front instability can be triggered by the density contrast between the two miscible fluids, leading to the Rayleigh-Taylor instability. While this type of instability can be suppressed by the surface reaction between the fluid and solid phases, which prevents the transport of the denser fluid to the deeper region at the beginning. Over the long term, it is found that the interfacial instability can be influenced by the evolution of the porosity due to the dissolution, which will drive the transport of denser fluid further down. Our investigation shows that the transport of the reactive fluid in porous media depends on the competition among the density contrast, the chemical reaction rate and the evolution of the porosity/permeability.

8:52AM M40.00005 Dynamics and mass transport of solutal convection in a closed porous media system, BAOLE WEN, DARIA AKHBARI, MARC HESSE, The University of Texas at Austin — Most of the recent studies of CO_2 sequestration are performed in open systems where the constant partial pressure of CO_2 in the vapor phase results in a time-invariant saturated concentration of CO_2 in the brine (C_s). However, in some closed natural CO_2 reservoirs, e.g., Bravo Dome in New Mexico, the continuous dissolution of CO_2 leads to a pressure drop in the gas that is accompanied by a reduction of C_s and thereby affects the dynamics and mass transport of convection in the brine. In this talk, I discuss the characteristics of convective CO_2 dissolution in a closed system. The gas is assumed to be ideal and its solubility given by Henry's law. An analytical solution shows that the diffusive base state is no longer self-similar and that diffusive mass transfer declines rapidly. Scaling analysis reveals that the volume ratio of brine and gas η determines the behavior of the system. DNS show that no constant flux regime exists for $\eta > 0$; nevertheless, the quantity F/C_s^2 remains constant, where F is the dissolution flux. The onset time is only affected by η when the Rayleigh number Ra is small. In this case, the drop in C_s during the initial diffusive regime significantly reduces the effective Ra and therefore delays the onset.

9:05AM M40.00006 Convective dissolution in anisotropic porous media¹, MARCO DE PAOLI, Univ of Udine, FRANCESCO ZONTA, TU Wien, ALFREDO SOLDATI, Univ of Udine and TU Wien — Solute convection in porous media at high Rayleigh-Darcy numbers has important fundamental features and may also bear implications for geological CO_2 sequestration processes. With the aid of direct numerical simulations, we examine the role of anisotropic permeability γ (the vertical-to-horizontal permeability ratio) on the distribution of solutal concentration in fluid saturated porous medium. Interestingly, we find that the finite-time (short-term) amount of solute that can be dissolved in anisotropic sedimentary rocks ($\gamma < 1$, i.e. vertical permeability smaller than horizontal permeability) is much larger than in isotropic rocks. We link this seemingly counterintuitive effect with the occurring modifications to the flow topology in the anisotropic conditions.

¹CINECA Supercomputing Centre and ISCRA Computing Initiative are gratefully acknowledged for generous allowance of computer resources. Support from Regione Autonoma Friuli Venezia Giulia under grant PAR FSC 2007/2013 is also gratefully acknowledged.

9:18AM M40.00007 Non-Boussinesq Dissolution-Driven Convection in Two- and Three-Dimensional Porous Media at Partially-Saturated Condition, MOHAMMAD AMIN AMOOIE, MOHAMMAD REZA SOLTANIAN, JOACHIM MOORTGAT, The Ohio State University — Sequestered carbon dioxide (CO_2) into saline aquifers, increases brine density through dissolution, and leads to gravitational instability and convective mixing. Traditionally, only the underlying brine-saturated subdomain is studied to avoid two-phase systems while replacing the gas cap atop with a constant, fully-saturated boundary condition. This violates the interface movement, neglects the capillary transition zone across original phases, and imposes constant density at top boundary insensitive to convective downwelling flow. Moreover, dissolution causes volume swelling, reflected as pressure build-up in absence of interface (movement), which further increases the fluid density –not captured under Boussinesq approximation. Here we accurately model the nonlinear phase behavior of brine- CO_2 mixture, altered by dissolution and compressibility. We inject CO_2 at a sufficiently low injection rate to maintain the single, partially-saturated phase, with no constraint on pressure and composition, so that density at top is free to change against the rate at which dissolved CO_2 migrates downwards. We discover new flow regimes and present quantitative scaling relations for their temporal evolution in both two- and three-dimensional porous media.

9:31AM M40.00008 Transient buoyant convection from a discrete source in porous media¹, ALI MORADI, MORRIS FLYNN, Dept. of Mech. Eng., Univ of Alberta — The study of porous media filling box flows informs (i) the dissolution of non-aqueous phase liquids or sequestered CO_2 into potable groundwater, (ii) leakage of contaminants from waste piles, and (iii) enhanced oil recovery technologies. Here we examine the flow of a negatively buoyant, laminar plume in a box filled with a porous medium, which is connected to an infinite external ambient via upper and lower fissures. As $t \rightarrow \infty$, the box contains two uniform layers of different densities. However, the approach towards steady state is characterized by a lower (contaminated) layer that is continuously stratified and is governed by the ratios of the virtual origin correction and lower fissure depth to the box height, and the ratio, μ , of the draining timescale to the filling timescale. Whereas the presence of a continuous stratification in the contaminated layer for finite time poses analytical challenges, we show that it is possible to derive bounds on the range of possible solutions. A separate component of our study considers time-variable forcing where the plume source strength is either abruptly altered or turned on and off with fixed half-period. Throughout, comparisons are drawn against filling boxes driven by turbulent free plumes.

¹NSERC, Carbon Management Canada

9:44AM M40.00009 Thermo-miscible fluid displacement in a porous media flow, THOMAS WARD, TEJASWI SOORI, Iowa State University — The an-isoviscous displacement of a generalized Newtonian liquid in an impulsively heated axisymmetric pipe geometry is studied at low to moderate Reynolds numbers using computational analysis. The temperature dependent viscosity is modeled using an empirical correlation that has been shown to fit experimental data for a range of temperature values. The governing Cauchy momentum equations for the generalized Newtonian fluid are solved in primitive variables using a 4th order Runge-Kutta method. For viscous liquids with a high Prandtl number radial and axial variations in temperature are significant leading to modification of the steady state pressure loss when compared to isoviscous displacements. We characterize the steady state pressure loss and average Nusselt number using the Reynolds $0.1 < Re < 10$, viscous Atwood $0 < At < 0.8$, and Peclet $100 < Pe < 10,000$ numbers.

9:57AM M40.00010 Saffman-Taylor instability in a rough fracture: beyond Darcys law, AMIR PAHLAVAN, LUIS CUETO-FELGUEROZO, GARETH MCKINLEY, RUBEN JUANES, Massachusetts Inst of Tech-MIT — The interplay between wetting and disorder, as well as the inherent complexity of porous media renders the description of immiscible flows a daunting task. To shed light on this problem, we conduct experiments on rough radial Hele-Shaw cells. We fill the cell with a viscous glycerol, and then inject a less viscous silicone oil at the center of the cell. The surfaces are treated to alter their wettability, allowing us to study both drainage and imbibition regimes. Viscous forces tend to destabilize the interfaces, whereas capillary forces play a stabilizing role; however, the disorder in the medium complicates this balance: 1) it leads to heterogeneities in the permeability field and capillary pressure distribution, and 2) it changes the effective wettability of the medium and leads to contact line pinning and hysteresis. We observe that at high capillary numbers, the disorder only weakly modulates the patterns, whereas at low capillary numbers, it affects the flow pattern significantly; the disorder leads to preferential flow paths in the drainage regime, and pinning and intermittent avalanche-like behavior in the imbibition regime. Inspired by these observations, we construct a phase-field model that takes the chemical potential gradients into account, going beyond Darcy's law.

Tuesday, November 22, 2016 10:45AM - 11:20AM –
Session N32 An Explanation for the Bedforms on Comet 67P/Churyumov-Gerasimenko Oregon Ballroom 201-202-203 - James Riley, University of Washington

10:45AM N32.00001 An explanation for the bedforms on comet 67P/Churyumov-Gerasimenko, PHILIPPE CLAUDIN, cole Suprieure de Physique et Chimie Industrielles — The recent approach of comet 67P/Churyumov-Gerasimenko by the European Space Agency spacecraft Rosetta has revealed the presence of geomorphologic features at its surface. In particular, surface patterns resembling aeolian ripples or dunes have been observed, especially in the 'neck' region. Erosion/deposition processes are unexpected on a comet because of the absence of an atmosphere, that would generate a wind to transport the grains at the surface. However, it is well known that comets experience outgassing when approaching the sun: the solar heat flux induces ice sublimation, generating dust/gas jets, which is at the origin of the comet's coma. Combining a description of sediment transport and hydrodynamics with a thermal model of the comets surface and outgassing, we show that, albeit generated by a rarefied atmosphere, these bedforms are paradoxically analogous to ripplemarks emerging on granular beds submitted to viscous shear flows.

Tuesday, November 22, 2016 10:45AM - 11:20AM –
Session N40 Ants, eyelashes, and the 2015 Ig Nobel Prize in Physics Portland Ballroom 253-258-254-257 - John Dabiri, Stanford University

10:45AM N40.00001 Ants, eyelashes, and the 2015 Ig Nobel Prize in Physics, DAVID HU, Georgia Institute of Technology — The zoo can be a source of recreation and rich scientific investigation. In this lecture, I will give an overview of my recent research with animals at the Atlanta Zoo. We will talk about how to make ant hamburgers, how eyelashes reduce evaporation of your eyes by a factor of two, and why mammals urinate for the same duration of 21 seconds. Although animal-inspired research can sound trendy, it can lead the way toward potential future directions in fluid mechanics, including the dynamics of active materials, flow through hairy surfaces, and the physics of digestion and excretion.

Tuesday, November 22, 2016 11:30AM - 11:50AM –
Session P32 Francois N. Frenkiel Award Lecture Oregon Ballroom 201-202-203 - Emmanuel Villermaux, Aix-Marseille Universite

11:30AM P32.00001 High-Speed Turbulent Reacting Flows: Intrinsic Flame Instability and its Effects on the Turbulent Cascade¹, ALEXEI POLUDNENKO, Texas AM University — Turbulent reacting flows are pervasive both in our daily lives on Earth and in the Universe. They power modern society being at the heart of many energy generation and propulsion systems, such as gas turbines, internal combustion and jet engines. On astronomical scales, thermonuclear turbulent flames are the driver of some of the most powerful explosions in the Universe, known as Type Ia supernovae. Despite this ubiquity in Nature, turbulent reacting flows still pose a number of fundamental questions often exhibiting surprising and unexpected behavior. In this talk, we will discuss several such phenomena observed in direct numerical simulations of high-speed, premixed, turbulent flames. We show that turbulent flames in certain regimes are intrinsically unstable even in the absence of the surrounding combustor walls or obstacles, which can support the thermoacoustic feedback. Such instability can fundamentally change the structure and dynamics of the turbulent cascade, resulting in a significant (and anisotropic) redistribution of kinetic energy from small to large scales. In particular, three effects are observed. 1) The turbulent burning velocity can develop pulsations with significant peak-to-peak amplitudes. 2) Unstable burning can result in pressure build-up and the formation of pressure waves or shocks when the flame speed approaches or exceeds the speed of a Chapman-Jouguet deflagration. 3) Coupling of pressure and density gradients across the flame can lead to the anisotropic generation of turbulence inside the flame volume and flame acceleration. We extend our earlier analysis, which relied on a simplified single-step reaction model, by demonstrating existence of these effects in realistic chemical flames (hydrogen and methane) and in thermonuclear flames in degenerate, relativistic plasmas found in stellar interiors. Finally, we discuss the implications of these results for subgrid-scale LES combustion models.

¹This work was supported by the Air Force Office of Scientific Research (AFOSR) under Award No. F4FGA06055G001, and the Department of Defense (DoD) High Performance Computing Modernization Program (HPCMP) under a Frontier project award.

Tuesday, November 22, 2016 11:30AM - 11:50AM –

Session P40 Andreas Acrivos Dissertation Award Lecture Portland Ballroom 253-258-254-257
- Antonino Ferrante, University of Washington

11:30AM P40.00001 How flexibility and dynamic ground effect could improve bio-inspired propulsion¹, DANIEL QUINN, Stanford University — Swimming animals use complex fin motions to reach remarkable levels of efficiency, maneuverability, and stealth. Propulsion systems inspired by these motions could usher in a new generation of advanced underwater vehicles. Two aspects of bio-inspired propulsion are discussed here: flexibility and near-boundary swimming. Experimental work on flexible propulsors shows that swimming efficiency depends on wake vortex timing and boundary layer attachment, but also on fluid-structure resonance. As a result, flexible vehicles or animals could potentially improve their performance by tracking their resonance properties. Bio-inspired propulsors were also found to produce more thrust with no loss in efficiency when swimming near a solid boundary. Higher lift-to-drag ratios for near-ground fixed-wing gliders is commonly known as ground effect. This newly observed “dynamic ground effect” suggests that bio-inspired vehicles and animals could save energy by harnessing the performance gains associated with near-boundary swimming.

¹This work was supported by the Office of Naval Research (MURI N00014-08-1-0642, Program Director Dr. Bob Brizzolara) and the National Science Foundation (DBI-1062052, PI Lisa Fauci; EFRI-0938043, PI George Lauder).

Tuesday, November 22, 2016 1:30PM - 2:48PM –

Session R1 Mini-Symposium - Multiphase Flows in Biomedicine II A105 - Kausik Sarkar, George Washington University

1:30PM R1.00001 The Subharmonic Behavior and Thresholds of High Frequency Ultrasound Contrast Agents¹, JOHN ALLEN, University of Hawaii - Manoa — Ultrasound contrast agents are encapsulated micro-bubbles used for diagnostic and therapeutic biomedical ultrasound. The agents oscillate nonlinearly about their equilibrium radii upon sufficient acoustic forcing and produce unique acoustic signatures that allow them to be distinguished from scattering from the surrounding tissue. The subharmonic response occurs below the fundamental and is associated with an acoustic pressure threshold. Subharmonic imaging using ultrasound contrast agents has been established for clinical applications at standard diagnostic frequencies typically below 20 MHz. However, for emerging applications of high frequency applications (above 20 MHz) subharmonic imaging is an area of on-going research. The effects of attenuation from tissue are more significant and the characterization of agents is not as well understood. Due to specificity and control production, polymer agents are useful for high frequency applications. In this study, we highlight novel measurement techniques to measure and characterize the mechanical properties of the shell of polymer contrast agents. The definition of the subharmonic threshold is investigated with respect to mono-frequency and chirp forcing waveforms which have been used to achieve optimal subharmonic content in the backscattered signal. Time frequency analysis using the Empirical Mode Decomposition (EMD) and the Hilbert-Huang transform facilitates a more sensitive and robust methodology for characterization of subharmonic content with respect to non-stationary forcing. A new definition of the subharmonic threshold is proposed with respect to the energy content of the associated adaptive basis decomposition. Additional studies with respect to targeted agent behavior and cardiovascular disease are discussed.

¹NIH, ONR

1:56PM R1.00002 Dynamics of encapsulated microbubbles for contrast ultrasound imaging and drug delivery: from pressure dependent subharmonic to collapsing jet and acoustic streaming¹, KAUSIK SARKAR, George Washington Univ — Intravenously injected microbubbles used as ultrasound contrast enhancing agents are encapsulated by a nanometer-thick layer of lipids, proteins or polymers to stabilize them against premature dissolution. Over the years, we have developed interfacial rheological models for the encapsulation and used them to characterize several contrast agents by acoustic means. We will present an overview of our research emphasizing recent efforts in two directions. The first is on using subharmonic signals from the contrast microbubbles for non-invasive pressure estimation. Experimental measurement and modeling show that the subharmonic signal can both increase or decrease with pressure depending on frequency. Secondly, we will discuss boundary element (BEM) simulation of the collapse of an encapsulated microbubbles forming a jet near a blood vessel wall. Different rheology models of the encapsulation have been rigorously implemented in the BEM formulation. We will discuss the resulting stresses and the acoustic streaming near the wall leading to sonoporation and other bioeffects.

¹Partially supported by Natinal Science Foundation

2:22PM R1.00003 Nonspherical dynamics and shape mode stability of ultrasound contrast agent microbubbles, MICHAEL CALVISI, University of Colorado, Colorado Springs — Ultrasound contrast agents (UCAs) are shell encapsulated microbubbles developed originally for ultrasound imaging enhancement. UCAs are more recently being exploited for therapeutic applications, such as for drug delivery, gene therapy, and tissue ablation. Ultrasound transducer pulses can induce spherical (radial) UCA oscillations, translation, and nonspherical shape oscillations, the dynamics of which are highly coupled. If driven sufficiently strongly, the ultrasound can induce breakup of UCAs, which can facilitate drug or gene delivery but should be minimized for imaging purposes to increase residence time and maximize diagnostic effect. Therefore, an understanding of the interplay between the acoustic driving and nonspherical shape mode stability of UCAs is essential for both diagnostic and therapeutic applications. In this work, we use both analytical and numerical methods to analyze shape mode stability for cases of small and large nonspherical oscillations, respectively. To analyze shape mode stability in the limit of small nonspherical perturbations, we couple a radial model of a lipid-coated microbubble with a model for bubble translation and nonspherical shape oscillation. This hybrid model is used to predict shape mode stability for ultrasound driving frequencies and pressure amplitudes of clinical interest. In addition, calculations of the stability of individual shape modes, residence time, maximum radius, and translation are provided with respect to acoustic driving parameters and compared to an unshelled bubble. The effects of shell elasticity, shell viscosity, and initial radius on stability are investigated. Furthermore, the well-established boundary element method (BEM) is used to investigate the dynamics and shape stability of large amplitude nonspherical oscillations of an ultrasonically-forced, polymer-coated microbubble near a rigid boundary. Different instability modes are identified based on the degree of jetting and proximity to the boundary. This insight is used to develop diagrams that delineate regions of stability from instability based on the breakup mechanism, in parameter ranges of ultrasound frequency and amplitude relevant to medical applications.

Tuesday, November 22, 2016 1:30PM - 3:14PM – Session R2 Industrial Application: General A106 - Sara Rimer, University of Michigan

1:30PM R2.00001 Flow Control of Hazardous Contaminants to Protect Evacuees in Civil Infrastructure Emergency Scenarios, SARA RIMER, University of Michigan — 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. An agent-based model is developed to simulate agents (i.e. building occupants) as they evacuate a public space. The agent-based evacuation model is coupled with the computational flow control model such that agents must interact with a dynamic, threatening environment. Results demonstrate how flow control can be achieved via feedback sensing of location of occupants with desire to minimize contaminant exposure.

1:43PM R2.00002 Computational study of liquid-gas cross-flow within structured packing cells¹, GIANLUCA LAVALLE, MATHIEU LUCQUIAUD, PRASHANT VALLURI, The University of Edinburgh — Absorption columns used in the carbon capture processes and filled with structured packings are crucial to foster the exchanges and the transfers between the absorber liquid and the flue gas. However, flow reversal can occur under special flow conditions, resulting in a dramatic drop of the technological performances. We investigate numerically the liquid-gas pattern within a cross-flow packing cell. The cell is a complex geometry with two connected channels, where the two phases flow co- or counter-currently. We show that an increase of both the gas speed and the liquid load leads to an increase of the pressure drop. Particular focus is also given to the analysis of flow repartition and flooding delay. We reveal that tilting the unit cell helps to delay the flooding and extends the operational capability. The pressure drop of the cross-flow unit cell is also compared to the Mellapak packing which is widely used in carbon capture applications. Finally, we support this study by performing numerical simulations on simpler geometries by means of a low-dimensional film-gas model, in order to investigate the two-phase dynamics and predict the flooding onset with a low computational cost.

¹The authors gratefully acknowledge EPSRC Grant No. EP/M001482/1

1:56PM R2.00003 Design of container velocity profile for the suppression of liquid sloshing¹, DONGJOO KIM, Kumoh National Institute of Technology — In many industrial applications, high-speed position control of a liquid container causes undesirable liquid vibrations called 'sloshing' which poses a control challenge in fast maneuvering and accurate positioning of containers. Recently, it has been shown that a control theory called 'input shaping' is successfully applied to reduce the sloshing, but its success comes at a cost of longer process time. Therefore, we aim to minimize liquid sloshing without increasing the process time when a container moves horizontally by a target distance within a limited time. In this study, sensing and feedback actuation are not permitted but the container velocity is allowed to be modified from a given triangular profile. A new design is proposed by applying input shaping to the container velocity with carefully selected acceleration time. That is, the acceleration time is chosen to be the 1st mode natural period, and the input shaper is determined based on the 3rd mode natural frequency. The proposed approach is validated by performing numerical simulations, which show that the simple modification of container velocity reduces the sloshing significantly without additional process time in a feedforward manner.

¹Supported by the NRF programs (NRF-2015R1D1A1A01059675) of Korean government

2:09PM R2.00004 Sloshing motion dynamics of a free surface in the draft tube cone of a Francis turbine operating in synchronous condenser mode, ELENA VAGNONI, LOïC ANDOLFATTO, ARTHUR FAREL, FRANÇOIS AVELLAN, École polytechnique fédérale de Lausanne — The penetration of the electrical grid by intermittent renewable energy sources induces grid fluctuations which must be compensated in order to guarantee the stability of the grid. Hydropower plants can supply reactive power to ensure the grid stabilization by operating in condenser mode. In this operating mode, the turbine operates with the tail water depressed to let the runner spin in air to reduce the power consumption. Pressurized air is injected in the draft tube cone to maintain the water level below the runner and this induces air-water interaction phenomena which cause important power losses. Flow visualization and pressure fluctuation measurements are performed in a reduced scale physical model of a Francis turbine operating in condenser mode to investigate the dynamics of the air-water interaction in the draft tube cone which causes the sloshing motion of the free surface. An image post-processing method is developed, enabling a quantitative description of the sloshing motion. The latter depends on the Froude number. By increasing the value of the Froude number, the amplitude of the sloshing motion decreases, as well as the amplitude of the pressure fluctuations. The frequency of the sloshing motion corresponds to the first natural frequency of the water volume.

2:22PM R2.00005 Design and development of low pressure evaporator/condenser unit for water-based adsorption type climate control systems, ARJUN VENKATARAMANAN, CARLOS A. RIOS PEREZ, CARLOS H. HIDROVO, Northeastern Univ — Electric vehicles (EVs) are the future of clean transportation and driving range is one of the important parameters which dictates its marketability. In order to increase driving range, electrical battery energy consumption should be minimized. Vapor-compression refrigeration systems currently employed in EVs for climate control consume a significant fraction of the battery charge. Thus, by replacing this traditional heating ventilation and air-conditioning system with an adsorption based climate control system one can have the capability of increasing the drive range of EVs. The Advanced Thermo-adsorptive Battery (ATB) for climate control is a water-based adsorption type refrigeration cycle. An essential component of the ATB is a low pressure evaporator/condenser unit (ECU) which facilitates both the evaporation and condensation processes. The thermal design of the ECU relies predominantly on the accurate prediction of evaporation/boiling heat transfer coefficients since the standard correlations for predicting boiling heat transfer coefficients have large uncertainty at the low operating pressures of the ATB. This work describes the design and development of a low pressure ECU as well as the thermal performance of the actual ECU prototype.

2:35PM R2.00006 Controlled evaporative cooling on a superhydrophilic surface: building a green wall, SUIN SHIM, SANGWOO SHIN, FORREST MEGGERS, ELIE BOU-ZEID, HOWARD A. STONE, Princeton University — We propose a way to design of an evaporative cooling device using continuous water flow on a superhydrophilic surface. Continuous flow helps prevent contaminant fouling on the surface of the cooler, which is a major challenge for conventional evaporative (swamp) coolers. A superhydrophilic surface leads to a reduced coolant flow rate, allowing for a maximum ratio of evaporative heat transfer to coolant thermal mass. Also, a staggered structure increases the surface area of the thin film flow of water which results in higher cooling efficiency. We performed both experimental and theoretical studies on the temperature change in the thin film flow of water. By keeping the water film thickness below 100 μm , ~ 5 K of temperature drop in the device was achieved. The cooling device can be manufactured using conventional cost-effective processes, offering practical applications in energy-efficient buildings.

2:48PM R2.00007 A new desalination technique using capacitive deionization, MO-HAMMAD SAJJAD ROSTAMY, MORTEZA KHASHECHI, EHSAN PIPELZADEH, None, DESALINATION TEAM — Capacitive deionization (CDI) is an emerging energy efficient, low pressure and low capital intensive desalination process where ions are separated by a pure electrostatic force imposed by a small bias potential as low as 1 V that is funded by an external Renewable (Solar) power supply to materials with high specific surface area. The main objective of this configuration is to separate the cation and anions on oppositely charged electrodes. Various electrode materials have been developed in the past, which have suffered from instability and lack of performance. Preliminary experimental results using carbon black, graphite powder, graphene\graphite\PTFE (Active\Conductive\binder), show that the graphene reduced via urea method is a suitable method to develop CDI electrode materials with capacitance as high as 52.2 mg/g for free standing graphene electrode. The focus of these studies has been mainly on developing electrodes with high specific surface area, high capacitance, excellent electronic conductivity and fast charge discharge cycles for desalination. Although some progress has been made, production of efficient and stable carbon based electrode materials for large scale desalination has not been fully realized.

3:01PM R2.00008 ABSTRACT WITHDRAWN —

Tuesday, November 22, 2016 1:30PM - 3:14PM —

Session R4 Applied Thermodynamics and Heat Transfer B112 - Mahboobe Mahdavi, Gannon University

1:30PM R4.00001 Experimental Analysis of the Effects of Inclination Angle and Working Fluid Amount on the Performance of a Heat Pipe¹, MAHBOOBE MAHDAMI, SAEED TIARI, Mechanical Engineering Department, Gannon University, Erie, PA, SONGGANG QIU, Department of Mechanical and Aerospace Engineering, West Virginia University, Morgantown, WV — Heat pipes are two-phase heat transfer devices, which operate based on evaporation and condensation of a working fluid inside a sealed container. In the current work, an experimental study was conducted to investigate the performance of a copper-water heat pipe. The performance was evaluated by calculating the corresponding thermal resistance as the ratio of temperature difference between evaporator and condenser to heat input. The effects of inclination angle and the amount of working fluid were studied on the equivalent thermal resistance. The results showed that if the heat pipe is under-filled with the working fluid, energy transferring capacity of the heat pipe decreases dramatically. However, overfilling heat pipe causes over flood and degrades heat pipe performance. The minimum thermal resistances were obtained for the case that 30% of the heat pipe volume was filled with working fluid. It was also found that in gravity-assisted orientations, the inclination angle does not have significant effect on the performance of the heat pipe. However, for gravity-opposed orientations, as the inclination angle increases, the temperature difference between the evaporator and condensation increases and higher thermal resistances are obtained.

¹ Authors appreciate the financial support by a research grant from Temple University

1:43PM R4.00002 Neutron Radiography for Determining the Evaporation/Condensation Coefficients of Cryogenic Propellants¹, K. BELLUR, E.F. MEDICI, M. KUL-SHRESHTHA, V. KONDURU, D. TYREWALA, C.-K. CHOI, J.S. ALLEN, Michigan Technological University, A. TAMILARASAN, J.C. HERMANSON, University of Washington, J.B. MCQUEILLEN, NASA Glenn Research Center, J. LEAO, D.S. HUSSEY, D.L. JACOBSON, J. SCHERSCHLIGT, National Institute of Standards and Technology — A novel, combined experimental and computational approach was used to determine the accommodation coefficients for liquid hydrogen and liquid methane in aluminum and stainless steel containers. The experimental effort utilized the NIST Neutron Imaging Facility to image the evaporation and condensation of cryogenic, hydrogenated propellants inside metallic containers. The computational effort included a numerical solution of a model for phase change in the contact line and thin film regions as well as a CFD effort for determining the appropriate thermal boundary conditions for the numerical solution of the evaporating and condensing liquid. These three methods in combination allow for extracting the accommodation coefficients from the experimental observations. The condensation and evaporation were controlled by adjusting the system temperature and pressure. The computational thermal model was shown to accurately track the transient thermal response of the test cells. The meniscus shape determination suggests the presence of a finite contact angle, albeit very small, between liquid hydrogen and an aluminum oxide surface.

¹ Research supported by the NASA Space Technology Research Grants Program (Grant NNX14AB05G).

1:56PM R4.00003 A heat transfer model for slug flow boiling within microchannels, MIRCO MAGNINI, JOHN THOME, Ecole Polytech Fed de Lausanne — We propose a novel physics-based model for the fluid mechanics and heat transfer associated with slug flow boiling in horizontal circular microchannels, to update the widely used three-zone model for the design of multi-microchannel evaporators. The flow is modelled as the cyclic passage of a liquid slug, an elongated bubble which traps a thin liquid film against the channel wall, and a dry vapor plug. The capillary flow theory, extended to incorporate evaporation effects, is applied to estimate the bubble velocity along the channel. A liquid film thickness prediction method considering bubble proximity effects, which may limit the radial extension of the film, is included. Theoretical heat transfer models accounting for the thermal inertia of the liquid film and for the recirculating flow within the liquid slug are utilized. The heat transfer model is compared to experimental data taken from three independent studies: 833 slug flow boiling data points covering R134a, R245fa and R236fa and channel diameters from 0.4 mm to 1 mm. The new model predicts more than 80% of the database to within $\pm 30\%$ and it represents an important step toward a complete physics-based modelling of bubble dynamics and heat transfer within microchannels under evaporating flow conditions.

2:09PM R4.00004 Nonlinear optimization of buoyancy-driven ventilation flow , SALEH NABI, PIYUSH GROVER, Mitsubishi Electric Research Labs, C.P. CAULFIELD, BP Institute & DAMTP, University of Cambridge

We consider the optimization of buoyancy-driven flows governed by Boussinesq equations using the Direct-Adjoint-Looping method. We use incompressible Reynolds-averaged Navier-Stokes (RANS) equations, derive the corresponding adjoint equations and solve the resulting sensitivity equations with respect to inlet conditions. For validation, we solve a series of inverse-design problems, for which we recover known globally optimal solutions. For a displacement ventilation scenario with a line source, the numerical results are compared with analytically obtained optimal inlet conditions available from classical plume theory. Our results show that depending on Archimedes number, defined as the ratio of the inlet Reynolds number to the Rayleigh number associated with the plume, qualitatively different optimal solutions are obtained. For steady and transient plumes, and subject to an enthalpy constraint on the incoming flow, we identify boundary conditions leading to 'optimal temperature distributions in the occupied zone.

2:22PM R4.00005 Numerical simulation of natural convection in vertical cylinders partially cooled from above , JOSE NUNEZ GONZALEZ, Escuela Nacional de Estudios Superiores, Unidad Morelia,UNAM, ALBERTO BELTRAN MORALES, Instituto de Investigaciones en Materiales, Unidad Morelia, UNAM, SERGIO CUEVAS, Instituto de Energias Renovables, UNAM

— Steady natural convection in vertical cylinders heated from below and partially cooled from above is studied from a numerical point of view. The governing equations for natural convection are discretized employing a mixed Fourier - Finite volume method using the SIMPLEC algorithm as velocity decoupling strategy. Calculations are performed for constant Prandtl number, $Pr=6.667$, and Rayleigh number over a range of $10^3 \leq Ra \leq 10^5$ and cooler size $0.125 \leq \gamma \leq 1$ and for an aspect ratio (height/diameter) $0.5 \leq a \leq 1.25$. Convective complex three-dimensional flow structures are presented.

2:35PM R4.00006 Experimental validation of a solar-chimney power plant model , NIMA FATHI, PATRICK WAYNE, IGNACIO TRUEBA MONJE, PETER VOROBIEFF, University of New Mexico

— In a solar chimney power plant system (SCPPS), the energy of buoyant hot air is converted to electrical energy. SCPPS includes a collector at ground level covered with a transparent roof. Solar radiation heats the air inside and the ground underneath. There is a tall chimney at the center of the collector, and a turbine located at the base of the chimney. Lack of detailed experimental data for validation is one of the important issues in modeling this type of power plants. We present a small-scale experimental prototype developed to perform validation analysis for modeling and simulation of SCPPS. Detailed velocity measurements are acquired using particle image velocimetry (PIV) at a prescribed Reynolds number. Convection is driven by a temperature-controlled hot plate at the bottom of the prototype. Velocity field data are used to perform validation analysis and measure any mismatch of the experimental results and the CFD data. CFD Code verification is also performed, to assess the uncertainty of the numerical model with respect to our grid and the applied mathematical model. The dimensionless output power of the prototype is calculated and compared with a recent analytical solution and the experimental results.

2:48PM R4.00007 Micro-Scale Thermoacoustics¹ , AVSHALOM OFFNER, GUY Z. RAMON, TECHNION

— Thermoacoustic phenomena conversion of heat to acoustic oscillations - may be harnessed for construction of reliable, practically maintenance-free engines and heat pumps. Specifically, miniaturization of thermoacoustic devices holds great promise for cooling of micro-electronic components. However, as devices size is pushed down to micro-meter scale it is expected that non-negligible slip effects will exist at the solid-fluid interface. Accordingly, new theoretical models for thermoacoustic engines and heat pumps were derived, accounting for a slip boundary condition. These models are essential for the design process of micro-scale thermoacoustic devices that will operate under ultrasonic frequencies. Stability curves for engines - representing the onset of self-sustained oscillations - were calculated with both no-slip and slip boundary conditions, revealing improvement in the performance of engines with slip at the resonance frequency range applicable for micro-scale devices. Maximum achievable temperature differences curves for thermoacoustic heat pumps were calculated, revealing the negative effect of slip on the ability to pump heat up a temperature gradient.

¹The authors acknowledge the support from the Nancy and Stephen Grand Technion Energy Program (GTEP).

3:01PM R4.00008 Mechanism and Structure of Subsurface Explosions in Granular Media , SHUYUE LAI, RYAN HOUIM, ELAINE ORAN, Univ of Maryland-College Park

— Numerical simulations of explosions in granular media were performed with an unsteady multidimensional fully compressible model, which solves two sets of coupled Euler equations, one for the gas and one for the granular medium. An explosive charge, buried in the granular medium, is modeled by a pocket of high-pressure and high-temperature gas. The initial conditions were determined based on an estimate of subsurface conditions on a comet. A series of simulations were performed in which the charge was buried at 3 m and 1.5 m and the particle volume fractions and the coefficient of restitution varied in the ranges 0.25 to 0.45 and 0 to 1, respectively. The simulations show the process of granular shock formation and propagation as a blast wave is created during the explosion. The blast wave initiates the particle motion and the particles accumulate to form a granular shock. The granular shock, in turn, produces a weak gas shock following it. There is a power law that relates the granular-shock radius to the explosion time: $R \sim t^{0.4}$, which is consistent with the results found by G. I. Taylor for 3-D spherical shock waves. The exponent of the power law remains at 0.4 regardless of the volume fraction and the elasticity of the granular material. For denser granular flows, the intergranular stress becomes stronger, and so the granular shock propagates at a higher velocity.

Tuesday, November 22, 2016 1:30PM - 3:14PM – Session R5 Compressible Flow: Shock-boundary Layer Interaction (computational)

B113 - Javier Urzay, Stanford University

1:30PM R5.00001 Upstream and Downstream Influence in STBLI Instability , PINO MARTIN, STEPHAN PRIEBE, CLARA HELM, University of Maryland

— Priebe and Martín (JFM, 2012) show that the low-frequency unsteadiness in shockwave and turbulent boundary layer interactions (STBLI) is governed by an inviscid instability. Priebe, Tu, Martín and Rowley (JFM, 2016) show that the instability is an inviscid centrifugal one, i.e Görtlerlike vortices. Previous works had given differing conclusions as to whether the low-frequency unsteadiness in STBLI is caused by an upstream or downstream mechanism. In this paper, we reconcile these opposite views and show that upstream and downstream correlations co-exist in the context of the nature of Görtler vortices. We find that the instability is similar to that in separated subsonic and laminar flows. Since the turbulence is modulated but passive to the global mode, the turbulent separated flows are amenable to linear global analysis. As such, the characteristic length and time scales, and the receptivity of the global mode might be determined, and low-order models that represent the low-frequency dynamics in STBLI might be developed. The centrifugal instability persists even under hypersonic conditions. This work is funded by the AFOSR Grant Number AF9550-15-1-0284 with Dr. Ivett Leyva.

1:43PM R5.00002 Transition in oblique shock/boundary layer interactions at Mach

5.92¹, ANUBHAV DWIVEDI, PRAKASH SHRESTHA, NATHANIEL HILDEBRAND, J.W. NICHOLS, M.R. JOVANOVIC, G.V. CANDLER, Univ of Minn - Minneapolis — We use the compressible flow solver US3D to perform DNS of an oblique shock wave interacting with a laminar boundary layer over an adiabatic flat plate at Mach 5.92. Simulations are repeated with different spanwise extents. The adverse pressure gradient created by the shock causes the boundary layer to separate, leading to the formation of a recirculation bubble downstream. We consider interactions of various strengths by varying the shock angle. A sufficiently strong interaction causes the flow to become $3 - D$, unsteady and eventually transition to turbulence. We observe long streamwise streaks downstream of the reattachment point which eventually break into turbulence. In the present work, we characterize the spatio-temporal dynamics of the unsteady separation bubble and these streaks using Fourier analysis and Sparsity Promoting Dynamic Mode Decomposition. To investigate the origin of these streaks we also analyze the role of linear Görtler instability resulting from the curvature of the streamlines induced by the separation bubble.

¹Supported by ONR, Grant No. N00014-15-1-2522

1:56PM R5.00003 Global stability analysis of oblique shock/boundary layer interactions at Mach 5.92¹

NATHANIEL HILDEBRAND, ANUBHAV DWIVEDI, PRAKASH SHRESTHA, JOSEPH W. NICHOLS, MIHAIGO R. JOVANOVIC, GRAHAM V. CANDLER, University of Minnesota - Twin Cities — We investigate the mechanisms by which an oblique shock impinging on a hypersonic, laminar boundary layer can transition to turbulence. As the shock angle increases, the initially stable flow undergoes a three-dimensional bifurcation to instability. We apply Direct Numerical Simulation (DNS) and global stability analysis to characterize the frequency and spanwise wavenumber selected by this bifurcation. The compressible flow solver US3D was used to perform DNS as well as to construct steady, two-dimensional base flows. Direct and adjoint global modes were extracted about each base flow with the shift-and-invert Arnoldi method. Linear stability analysis was repeated for various shock angles to identify when the bifurcation occurs. An angle of 14 degrees resulted in unstable eigenvalues for spanwise wavenumbers around 0.32. The most unstable mode resides in the shear layer and creates streaks downstream. Multiplying this direct mode by its corresponding adjoint, we find the wave maker for this instability and show it's sensitive to changes near the reattachment point. We also use the adjoint modes to project DNS data on the direct modes to see their physical relevance.

¹We are grateful to the Office of Naval Research for supporting this study through grant number N00014-15-1-2522.

2:09PM R5.00004 Confinement effects in shock/turbulent-boundary-layer interaction through wall-modeled LES¹, IVAN BERMEJO-MORENO, University of Southern California, LAURA CAMPO, Stanford University, JÖHAN LARSSON, University of Maryland, College Park, JULIEN BODART, Université de Toulouse, ISAE, France, DAVID HELMER, JOHN EATON, Stanford University — Wall-modeled large-eddy simulations (WMLES) are used to investigate three-dimensional effects imposed by lateral confinement on the interaction of oblique shock waves impinging on turbulent boundary layers (TBLs) developed along the walls of a nearly-square duct. A constant Mach number, $M = 2.05$, of the incoming air stream is considered, with a Reynolds number based on the incoming turbulent boundary layer momentum thickness $Re_\theta \approx 14,000$. The strength of the impinging shock is varied by increasing the height of a compression wedge located at a constant streamwise location that spans the top wall of the duct at a 20 angle. Simulation results are first validated with particle image velocimetry (PIV) experimental data obtained at several vertical planes. Emphasis is placed on the study of the instantaneous and time-averaged structure of the flow for the stronger-interaction case, which shows mean flow reversal. By performing additional spanwise-periodic simulations, it is found that the structure and location of the shock system and separation bubble are significantly modified by the lateral confinement. Low-frequency unsteadiness and downstream evolution of corner flows are also investigated.

¹Financial support from the United States Department of Energy under the PSAAP program is gratefully acknowledged.

2:22PM R5.00005 Separation control in a hypersonic shock wave / turbulent boundary-layer interaction¹, ANNE-MARIE SCHREYER, Institute of Fluid Mechanics, TU Braunschweig, Germany, IVAN BERMEJO-MORENO, Aerospace and Mechanical Engineering, University of Southern California, JEONGLAE KIM, JAVIER URZAY, Center for Turbulence Research, Stanford University — Hypersonic vehicles play a key role for affordable access to space. The associated flow fields are strongly affected by shock wave/turbulent boundary-layer interactions, and the inherent separation causes flow distortion and low-frequency unsteadiness. Microramp sub-boundary layer vortex generators are a promising means to control separation and diminish associated detrimental effects. We investigate the effect of a microramp on the low-frequency unsteadiness in a fully separated interaction. A large eddy simulation of a 33°-compression-ramp interaction was performed for an inflow Mach number of 7.2 and a Reynolds number based on momentum thickness of $Re_\theta = 3500$, matching the experiment of Schreyer et al.(2011). For the control case, we introduced a counter-rotating vortex pair, as induced by a single microramp, into the boundary layer through the inflow conditions. We applied a dynamic mode decomposition (DMD) on both cases to identify coherent structures that are responsible for the dynamic behavior. Based on the DMD, we discuss the reduction of the separation zone and the stabilization of the shock motion achieved by the microramp, and contribute to the description of the governing mechanisms.

¹Pursued during the 2016 CTR Summer Program at Stanford University.

2:35PM R5.00006 LES of shock wave/turbulent boundary layer interaction affected by microramp vortex generators, LAURENT JOLY, ARNAUD GREBERT, STPHANE JAMME, JULIEN BODART, Institut Supérieur de l'Aéronautique et de l'Espace, Université de Toulouse, AERODYNAMICS, ENERGETICS AND PROPULSION DEP. TEAM — At large Mach numbers, the interaction of an oblique shock wave with a turbulent boundary layer (SWTBLI) developing over a flat plate gives rise to a separation bubble known to exhibit low-frequency streamwise oscillations around $St_L = 0.03$ (a Strouhal number based on the separated region length). Because these oscillations yield wall pressure or load fluctuations, efforts are made to reduce their amplitude. We perform large eddy simulations to reproduce the experiments by Wang et al (2012) where a rake of microramp vortex generators (MVGs) were inserted upstream the SWTBLI with consequences yet to be fully understood. There is no consensus on the flow structure downstream MVGs and this is first clarified in the case of MVGs protruding by 0.47δ in a TBL at Mach number $M = 2.7$ and Reynolds number $Re_\theta = 3600$. Large-scale vortices intermittently shed downstream the MVGs are characterized by a streamwise period close to twice the TBL thickness and a frequency $f \approx 0.5U_e/\delta$, two orders of magnitude higher than the one of the uncontrolled SWTBLI. We then characterize the interaction between the unsteady wake of the MVGs with the SWTBLI resulting in the reduction of the interaction length and the high-frequency modulation of the shock foot motions.

2:48PM R5.00007 Characterization of the low-frequency unsteadiness in LES data of supersonic and hypersonic STBLI, CLARA HELM, PINO MARTIN, University of Maryland — In a recent study, Priebe *et al.* (JFM 2016) used Dynamic Mode Decomposition (DMD) to analyze DNS data of a Mach 3 ramp-generated shock and turbulent boundary layer interaction (STBLI). The authors found that the reconstructed low-frequency DMD modes took on the form of Görtler-like vortices downstream of separation. The five reconstructed modes reproduced the low-frequency dynamics of the separation bubble accurately. Martín *et al.* (AIAA2016-3341) and Martín *et al.* (APS, DFD 2016) show that the low-frequency unsteadiness in STBLI results from an inviscid centrifugal instability similar to that found in separated subsonic and laminar flows, and that the turbulence is modulated but passive to the global mode. In this work we further characterize the Görtler-like vortices using LES data of Mach 3 and Mach 7 separated STBLIs. We find that the Görtler-like vortices are unsteady, and we quantify the wavelength, amplitude and the aperiodic development of these structures. This work is supported by the Air Force Office of Scientific Research under grant AF9550-15-1-0284.

3:01PM R5.00008 An explicit filtering framework based on Perona-Malik anisotropic diffusion for shock capturing and subgrid scale modeling of Burgers' turbulence, ROMIT MAULIK, OMER SAN, Oklahoma State University - Stillwater — In this work, we introduce a relaxation filtering closure approach to account for subgrid scale effects in explicitly filtered large eddy simulations using the concept of anisotropic diffusion. We utilize the Perona-Malik diffusion model and demonstrate its shock capturing ability and spectral performance for solving the Burgers turbulence problem, which is a simplified prototype for more realistic turbulent flows showing the same quadratic nonlinearity. Our numerical assessments present the behavior of various diffusivity functions in conjunction with a detailed sensitivity analysis with respect to the free modeling parameters. In comparison to direct numerical simulation (DNS) and under-resolved DNS results, we find that the proposed closure model is efficient in the prevention of energy accumulation at grid cut-off and is also adept at preventing any possible spurious numerical oscillations due to shock formation under the optimal parameter choices. In contrast to other relaxation filtering approaches, it is also shown that a larger inertial range can be obtained by the proposed anisotropic diffusion model using a compact stencil scheme in an efficient way.

Tuesday, November 22, 2016 1:30PM - 3:40PM –
Session R6 Aerodynamics: Wing/Vortex Interaction B114 - Adam DeVoria, University of Florida
- Gainesville

1:30PM R6.00001 High lift generation of low-aspect-ratio wings, ADAM DEVORIA, KAMRAN MOHSENI, Univ of Florida - Gainesville — The time-averaged flow field in the center-span of low-aspect-ratio rectangular wings is experimentally measured. It is shown that lift stall is preceded by shedding of strong trailing-edge vorticity. The induced downwash of the tip vortices delays the growth of the attached boundary layer as well as leading-edge separation. Reattached flow occurs for sufficiently low aspect ratios and results in a smooth merging of the flow at the trailing edge thus assisting in satisfying a Kutta condition there. As a consequence, the strength of vorticity shed from the trailing edge is decreased and allows for continued lift generation at high angles of attack. When the reattachment point passes beyond the trailing edge, a strong shear layer is generated there and represents negative lift, leading to stall with a slight increase in angle of attack or aspect ratio.

1:43PM R6.00002 Identification of separate flow features in the shear layer, KAREN MULLENERS, SWATHI KRISHNA, EPFL, MELISSA GREEN, Syracuse University — Analyzing unsteady flow fields primarily involves the identification of dynamically significant regions of vorticity in the flow. Detection of all the flow features is essential for an accurate description of the physics of the flow, which eventually helps in improving flow modeling and predictions. Eulerian criteria such as λ_2 and Γ_2 successfully identify large scale structures based on local velocity gradients and topology but do not detect the coherent vortices with the concentrated vorticity in a shear layer. The identification of these smaller structures within the shear layer is important when predicting the overall circulatory contribution to the aerodynamic forces produced, in applications such as flapping wing design. In order to detect the smaller flow features along with the prominent large scale vortices, an alternative method of vortex identification is proposed in which the flow structures are detected based on the vorticity contours. This method is applied to numerical and experimental data of a pitching panel to highlight its robustness. In addition, the finite time Lyapunov exponent (FTLE) is calculated to show that the boundaries of the material lines and identified vorticity contours coincide.

1:56PM R6.00003 Potential flow predictions for a flapping flat plate wing, SWATHI KRISHNA, KAREN MULLENERS, EPFL, MELISSA GREEN, Syracuse University — It is well established that the leading edge vortex is one of the major contributors to the generation of lift on a flapping insect wing. However, the contributions of the trailing edge vortices and the shear layer to unsteady force production mechanisms needs more investigation. The individual contribution of different flow structures is especially important if reliable theoretical predictions of lift and drag are to be made, that eventually assist in the design of micro air vehicles. The current work aims to distinguish different flow features of an unsteady flow field generated by a flapping wing in hover and to quantify the role played by them in the generation of aerodynamic forces. This is achieved by employing a semi-empirical potential flow model that allows for the calculation of lift by theoretically recreating the potential flow field based on the vortex strengths and locations obtained from phase-averaged particle image velocimetry (PIV) data. Individual flow structures are detected in the PIV data based on the vorticity contours. The theoretically predicted lift is compared with direct force measurements to demonstrate the utility and limitations of the model.

2:09PM R6.00004 The Structure of a Trailing Vortex from a Perturbed Wing, GREGORY FISHMAN, DONALD ROCKWELL, Lehigh University — The unsteady structure of a trailing vortex may be interpreted as a three-dimensional gust. Such a vortex, or gust, potentially impinges upon a follower wing positioned on or near its trajectory, thereby giving rise to unsteady buffeting of its surface and/or disruption of its flight path. Stereoscopic particle image velocimetry and a three-dimensional construction technique are employed to characterize the structure of a trailing vortex from a wing subjected to displacement perturbations in the heaving mode with an amplitude an order of magnitude smaller than the diameter of the vortex and a wavelength two orders of magnitude greater than the diameter of the vortex. This perturbation leads to relatively large undulations of axial velocity deficit within, and circulation of, the vortex. Along the axis of the vortex, these fluctuations are associated with alternating regions of low and high values of swirl ratio. This results in an internal vortex structure comprised of successive regions of instability separated by stabilized regions. These mechanisms are therefore linked to the origin of the large gust-like fluctuations of axial velocity deficit and circulation.

2:22PM R6.00005 Characterization of vortical gusts produced by a heaving plate¹

, ESTEBAN HUFSTEDLER, BEVERLEY J. MCKEON, California Institute of Technology — To experimentally investigate the interaction between a wing and a spanwise vortical gust, a simple gust generator has been built and tested. This consists of a transversely heaving flat plate that changes direction to release a vortex, which then convects downstream to interact with a wing. Previous experiments have shown that, immediately downstream of the plate, the circulation of the generated vortex is proportional to the heaving speed of the plate. The forces that the gusts exert on a downstream wing were shown to be strongly repeatable and consistent with a passing vortex. This presentation will discuss the properties of the vortical gusts as they move downstream, and relate those properties to the important dimensionless parameters of the flow. These properties include the convection speed and circulation of the vortex, as well as the enstrophy due to the wake of the plate.

¹This research is funded by the Gordon and Betty Moore Foundation through Grant GBMF2645 to the California Institute of Technology.

2:35PM R6.00006 Measurement and Modelling of a Heaving Airfoil Flow , VICTOR TROSHIN, AVRAHAM SEIFERT, Tel Aviv Univ

— An outline of a low order modelling procedure of a heaving airfoil in still fluid using experimental measurements is provided. Due to its relative simplicity, the proposed procedure is applicable for the analysis of flow fields within complex and unsteady geometries and it is ideal for analysing the data obtained by experimentation. Currently, this procedure is used to model and predict the flow field evolution using small number of low profile load sensors and flow field measurements. The time delay neural networks are used in order to estimate the flow field. The neural networks estimate the amplitudes of the most energetic modes using four sensory inputs. The modes are calculated using proper orthogonal decomposition (POD) of the flow field data obtained experimentally by time-resolved, phase-locked particle imaging velocimetry. In order to permit the use of proper orthogonal decomposition, the measured flow field is mapped onto a stationary domain using volume preserving transformation. The analysis performed by the model showed good estimation quality within the parameter range used in the training procedure. However, the performance deteriorates for cases out of this range. This state indicates that, in order to improve the robustness of the model, both the decomposition and the training data sets must be diverse.

2:48PM R6.00007 Evaluating low order models for force prediction in high-amplitude gusts , GINO PERROTTA, ANYA JONES, University of Maryland

— The unsteady forces on a plunging wing were measured for high-amplitude transient motions. The plunging velocity paralleled the canonical sine-squared transverse gust profile, including cases with plunging velocities far greater than the free stream velocity. The ratio of plunging velocity to free stream velocity was varied from one-sixth to 24 which allowed for quantification and demonstration of the increasing error in typical force prediction models. Each velocity ratio was tested at multiple values of free stream velocity. All cases were tested with a free stream to wing incidence angle of zero degrees. Forces and moments were measured on the same rigid flat plate wing for all cases. Measured forces during the gust were compared to forces predicted by various models, and the error between them was quantified. The parameter space defining the sine-squared gust was then partitioned into regions of high accuracy for unsteady force prediction models such as potential flow, quasi-steady based on steady measurements, and indicial functions. This highlights the strengths and weaknesses of each model, and identifies gust conditions that are not adequately modeled by any of these tools.

3:01PM R6.00008 A versatile low-dimensional vortex model for investigating unsteady aerodynamics¹ , DARWIN DARAKANANDA, JEFF D. ELDREDGE, University of California, Los Angeles — In previous work, we demonstrated a hybrid vortex sheet/point vortex model that captures the non-linear aerodynamics of a plate translating at a high angle of attack. We used vortex sheets to model the shear layers emerging from the plate, and point vortices to capture the effect of the coherent vortex structures. In this work, we introduce modifications that allow the model to work for a larger range of plate kinematics over longer periods of time. First, following the example of Ramesh et al., we relax the Kutta condition at the leading edge and determine vorticity flux based on a suction parameter instead. To prevent the vortex sheet from becoming unstable near the resulting singular edge, we explicitly filter out short-wave disturbances along the sheet while redistributing the sheet's control points. Second, by looking for intersections between the vortex sheets and any repelling Lagrangian coherent structures, the model can detect the formation of new coherent vortices. Trailing portions of the sheets that become dynamically distinct from the shear layers are rolled up into point vortices. We test these modifications on a variety of problems, including pitch-up, impulsive translation at low angles of attack, as well as flow response to pulse actuation near the leading edge.

¹This work has been supported by AFOSR, under award FA9550-14-1-0328.

3:14PM R6.00009 Balance equations for triple-joint vortex-sheet structures , XI XIA, KAMRAN MOHSENI, University of Florida

— A vortex sheet is the limiting case for a viscous shear layer as the thickness approaches zero. Recently, vortex-sheet based flow models have been demonstrated to provide significant reduction for numerical simulations of viscous and inviscid flows. In such modeling approaches, a prominent phenomenon is the formation of a new vortex sheet from existing vortex sheets, thereby creating a triple-joint vortex-sheet structure. In this study, the formation of the new vortex sheet is analytically determined by applying conservation laws of mass and momentum to flow surrounding the entire triple-joint vortex-sheet structure, together with the boundary conditions specific to any application. As a result, a general condition is obtained to determine the angle, strength, and velocity of the new vortex sheet. This model is validated by simulating airfoils in steady and unsteady background flows and comparing the flow structures and force calculations with experimental data. While the performance of this model is demonstrated in this study for the vortex shedding problem at the trailing edge, its future applications could be extended to flow separation on a smooth surface and triple contact point of multi-phase flows.

3:27PM R6.00010 On the leading edge vortex of thin wings¹ , ABEL ARREDONDO, IGNACIO MARIA VIOLA, University of Edinburgh — On thin wings, the sharp leading edge triggers laminar separation followed by reattachment, forming a Leading Edge Vortex (LEV). This flow feature is of paramount importance because, if periodically shed, it leads to large amplitude load fluctuations, while if stably attached to the wing, it can provide lift augmentation. We found that on asymmetric-spinnaker-type yacht sails, the LEV can be stable despite the relatively low sweep (30°). This finding, which was recently predicted numerically by Viola et al. (Ocean Eng., 2014; 90:93-103), has been confirmed through current flume tests on a 1:115th model scale sail. Forces were measured and Particle Image Velocimetry was performed on four horizontal sail sections at a Reynolds number of 1.7×10^4 . Vortex detection revealed that the LEV becomes progressively larger and more stable towards the highest sections, where its axis has a smaller angle with respect to the freestream velocity. Mapping the sail section on a rotating cylinder through a Joukowski transformation, we quantified the lift augmentation provided by the LEV on each sail section. These results open up new sail design strategies based on the manipulation of the LEV and can be applicable to the wings of unmanned aerial vehicles and underwater vehicles.

¹Project funded by Conacyt.

Tuesday, November 22, 2016 1:30PM - 3:27PM –

Session R9 General Fluid Dynamics: Instability, Surfaces and Cavitation B117 - Michelle Driscoll, New York University

1:30PM R9.00001 Shape-based separation of microparticles with magnetic fields

CHENG WANG, RAN ZHOU, Missouri Univ of Sci & Tech — Precise manipulations, e.g., sorting and focusing, of nonspherical microparticles in fluidic environment has important applications in the fields of biology sciences and biomedical engineering. However, non-spherical microparticles are hard to manipulate because they tumble in shear flows. Most of existing techniques, including traditional filtration and centrifugation, and recent microfluidic technology, have difficulty in separating microparticles by shape. We demonstrate a novel shape-based separation technique by combining external magnetic fields with pressure-driven flows in a microchannel. Due to the magnetic field, prolate ellipsoidal particles migrate laterally at different speeds than the spherical ones, leading to effective separation. Our experimental investigations reveal the underlying physical mechanism of the observed shape-dependent migration. We find that the magnetic field breaks the rotational symmetry of the nonspherical particles, and induces shape-dependent lift force and migration velocity.

Ideally, this abstract should be just before the oral presentation of Merlin Etzold.

Many thanks,

1:43PM R9.00002 Decontamination of chemical tracers in droplets by a submerged thin film flow¹

JULIEN R. LANDEL, DAMTP, University of Cambridge, HARRY MCEVOY, Dstl, STUART B. DALZIEL, University of Cambridge — We investigate the decontamination of chemical tracers contained in small viscous drops by a submerged thin film. This problem has applications in the decontamination of hazardous chemicals, following accidental releases or terrorist attacks. Droplets lying on surfaces are cleaned by spraying a liquid decontaminant over the surface. The decontaminant film submerges the droplets without detaching them, in order to neutralize toxic chemicals in the droplets. The decontamination process is controlled by advection, and reaction processes near the drop-film interface. Chemical tracers dissolve into the film flow forming a thin diffusive boundary layer at the interface. The chemical tracers are then neutralized through a reaction with a chemical decontaminant transported in the film. We analyze this work that the decontamination process occurs mainly in the film phase owing to low solubility of the decontaminant in the droplet. We analyze the impact of the reaction time scale, assuming first-order reaction, in relation with the characteristic advection and diffusion scales in the case of a single droplet. Using theoretical, numerical and experimental means, we find that the reaction time scale is significantly smaller than the characteristic time scale in the diffusive boundary layer in order to enhance noticeably the decontamination of a single toxic droplet. We discuss these results in the more general case of the decontamination of a large number of droplets.

Julien

¹This material is based upon work supported by the Defense Threat Reduction Agency under Contract No. HDTRA1-12-D-0002.

1:56PM R9.00003 Chemical decontamination of façade cracks

MERLIN A. ETZOLD, JULIEN R. LANDEL, STUART B. DALZIEL, DAMTP, University of Cambridge — The problem of cleaning and decontamination of buildings arises in the context of chemical spillages, terrorist attacks, industrial applications and in day-to-day situations such as the removal of graffiti. A common feature of all buildings is the existence of cracks and fissures, which act as contaminant traps. This contribution reports experiments and modelling of the removal of a water-soluble contaminant from the bottom of an idealised V-shaped crack. The contaminant is dissolved in a polymer thickened droplet. The surface washing techniques commonly used in industrial decontamination induce a flow in the crack which is mostly controlled by the crack geometry. Rinsing with pure water is compared against the situation in which a neutralising chemical is present. The cleaning process is modelled by solving the time-dependent diffusion equation within the droplet coupled to the steady state advection-diffusion equation outside the droplet. This approach is similar to the work of Landel et al. on decontaminating plane surfaces beneath falling films [JFM (2016), vol. 789, pp. 630-668]. Our results indicate that the proposed model describes successfully the earlier stages of decontamination. In later stages the dissolution of the thickened matrix may contribute to the process.

2:09PM R9.00004 Flow visualization of the trapping induced by vortex breakdown at a junction¹

DANIELE VIGOLO, University of Birmingham (UK), MARCO RICCOMI, Universita' di Pisa (Italy), FEDERICO ALBERINI, University of Birmingham (UK), ELISABETTA BRUNAZZI, Universita' di Pisa (Italy), JESSE AULT, HOWARD STONE, Princeton University — Here we present experimental investigations of the vortex breakdown happening at a T-, Y- or "arrow" shaped junction responsible for the trapping of light material suspended in solution. Considering the ubiquitous nature of T-junctions and bifurcation in general, in industrial as well as biological environments, it is extremely interesting to better understand how this trapping phenomenon happens. In particular, we observed the flow profiles at different sections in order to perform a three-dimensional study of complex structures, such as vortices and recirculation zones, that develop at a bifurcation. We explored Reynolds number ranging from 50 to about 500 for different milli-fluidic devices. Thus we compared standard micro-PIV and a novel optical technique, the Ghost Particle Velocimetry (GPV), that was recently introduced, to investigate the onset of vortex breakdown. Moreover, the experimental results were compared with single-phase OpenFoam numerical simulations performed in the same flow conditions. Finally, we studied the mutual influence of a trapped particle on the flow field inside the recirculation zone by fully exploiting the capability of GPV to produce 3D flow field with a spatial resolution of few tens of microns.

¹This work was partially supported by the Wellcome Trust [1516ISSFFIRA23]

2:22PM R9.00005 An experimental and computational study of large bubble coalescence in stagnant highly viscous liquids¹

SARA MOHAMMED, EZEKIEL AGUNLEJIKA, University of Nottingham, ZHIHUA XIE, Imperial College London, BUDDHIKA HEWKANDAMBY, BARRY AZZOPARDI, University of Nottingham, OMAR MATAR, Imperial College London — The coalescence of two, and sometimes, three large bubbles rising in columns of viscous, stagnant liquids is studied experimentally and computationally. Two cases are considered: a 38 mm diameter column with a 0.12 Pa s liquid (aqueous solution of glycerol/potassium chloride); a 290 mm diameter column with a silicone oil of 330 Pa s viscosity. High-speed videos are taken of the coalescence process, which are characterised by the acceleration of the trailing bubble into the rear of the leading one. There is significant penetration of the trailing bubble into the leading one, with a noticeable delay prior to rupture of the thin film separating the bubbles. The velocities of the individual bubbles, as well as the bubble shapes are measured accurately. Numerical simulations of the bubble rise and coalescence process are also carried out using the parallelised, control-volume, finite-element code, *Fluidity*, which uses adaptive, unstructured meshing. The numerical results are compared with the experimental observations in terms of single bubble shape and speed, as well as the entire dynamics of two-bubble coalescence process; particular attention is focused on bubble penetration and the final stages of coalescence for the very large viscosity ratio case.

¹EPSRC UK Programme Grant MEMPHIS (EP/K003976/1)

2:35PM R9.00006 Cavitation during wire brushing. , BO LI, JUN ZOU, CHEN JI, Zhejiang University —

In our daily life, brush is often used to scrub the surface of objects, for example, teeth, pots, shoes, pool, etc. And cleaning rust and stripping paint are accomplished using wire brush. Wire brushes also can be used to clean the teeth for large animals, such as horses, crocodiles. By observing brushing process in water, we capture the cavitation phenomenon on the track of moving brush wire. It shows that the cavitation also can affect the surface. In order to take clear and entire pictures of cavity, a simplified model of one stainless steel wire brushing a boss is adopted in our experiment. A transparent organic tank filled with deionized water is used as a view box. And a high speed video camera is used to record the sequences. In experiment, ambient pressure is atmospheric pressure and deionized water temperature is kept at home temperature. An obvious beautiful flabellate cavity zone appears behind the moving steel wire. The fluctuation of pressure near cavity is recorded by a hydrophone. More movies and pictures are used to show the behaviors of cavitation bubble following a restoring wire. Beautiful tracking cavitation bubble cluster is captured and recorded to show.

2:48PM R9.00007 A purely elastic upstream instability in channel flows , BOYANG QIN,

PAULO ARRATIA, University of Pennsylvania — In this talk, the flow of a viscoelastic fluid is experimentally investigated using particle velocimetry methods in a microfluidic device. The microchannel is 100-micron wide and deep and consists of a long (3-cm) straight region followed by a linear array of cylinders. Velocimetry measurements show the presence of flow instabilities far upstream of the linear array of cylinders. Velocity fluctuations are found up to 10 channel widths upstream the perturbation (linear array of cylinders); they extend far beyond the unsteady vortex that develops immediately in front the cylinder and are excited at many time scales. These fluctuations increase as the Weissenberg number (or flow rate) is increased. However, beyond a certain value of Wi , these fluctuations sharply decrease. We believe that the observed decrease in velocity fluctuations at high Wi correspond to a regime in which the flow speed (convection) is larger than the elastic wave speed, that is, the elastic Mach number exceeds unity.

3:01PM R9.00008 Thermally aware, durable nanoengineered surfaces with high speed liquid impalement resistance¹ , MANISH TIWARI, CHAOYI PENG, ZHUYANG CHEN, Nanoengineered Systems Laboratory, UCL Mechanical Engineering, University College London, London WC1E 7JE, UK — Highly hydrophobic nanoengineered surfaces delaying freezing down to -20 degrees Centigrade for a day, sustaining dropwise steam condensation under high rate steam shear for several days, sustaining mechanical abrasion and high strains have attracted strong interest recently. Particularly, anti-icing and dropwise condensation promotion require thermally conductive surfaces with careful nucleation control – of ice germs or droplets, respectively – using precise surface nanotexture. Scalability of surface manufacture is an additional challenge. In the current presentation, we will demonstrate a pathway to address these needs. Anodisation of metallic substrate is first used to obtain nanotextured surfaces with a precision of approx. 200 nm. Next, rationally formulated nanocomposites comprising solution processed fluorinated copolymers and nanoparticle dispersions were spray coated on the anodized metals. The resulting nanocomposite coatings were superhydrophobic with approx. 20 nm precision in surface texture. The surface durability is assessed using tape peel, sand abrasion, and droplet and water jet impact tests up to 30 m/s. High speed jet splashing is recorded at speeds >10 m/s to demonstrate the influence of jet diameter on splashing characteristics.

¹This work was partly supported by EPSRC grant EP/N006577/1.

3:14PM R9.00009 Predicting the onset of high-frequency self-excited oscillations in a channel with an elastic wall , THOMAS WARD, ROBERT WHITTAKER, University of East Anglia — Flow-induced oscillations of fluid-conveying elastic-walled channels arise in many industrial and biological systems including the oscillation of the vocal cords during phonation. We derive a system of equations that describes the wall displacement in response to the steady and oscillatory components of the fluid pressure derived by Whittaker et. al. (2010). We show that the steady pressure component results in a base state deformation assumed to be small in magnitude relative to the length of the channel. The oscillation frequency of the elastic wall is determined by an eigenvalue problem parameterised by the shape of the base state deformation, the strength of axial tension relative to azimuthal bending, \mathcal{F} , and the size of non-linear stretching effects from the wall's initial deformation, K . We determine the slow growth or decay of the normal modes in each by considering the energy budget of the system. The amplitude of the oscillations grow or decay exponentially with a growth rate Λ , which may be expressed in terms of a critical Reynolds number Re_c . We use numerical simulations to identify three distinct regions in parameter regimes space and determine the stability of oscillations in each.

Tuesday, November 22, 2016 1:30PM - 3:40PM –

Session R10 DFD GPC: Convection and Buoyancy Driven Flows: Planetary & Exoplanetary Dynamics B118-119 - Ben Brown, University of Colorado

1:30PM R10.00001 Convective overshoot at stiffly stable interfaces , BENJAMIN BROWN,

University of Colorado, JEFFREY OISHI, Bates College, DANIEL LECOANET, Princeton University, KEATON BURNS, Massachusetts Institute of Technology, GEOFFREY VASIL, University of Sydney — Convective overshoot is an important non-local mixing and transport process in stars, extending the influence of turbulent stellar convection beyond the unstable portions of the atmosphere. In the Sun, overshoot into the tachocline at the base of the convection zone has been ascribed a major role in the storage and organization of the global-scale magnetic fields within the solar dynamo. In massive stars, overshooting convection plays an important role in setting the lifespan of the star by mixing fuel into the nuclear burning core. Here we narrowly consider the properties of convective overshoot across very stiff interfaces within fully compressible dynamics across convection zones with significant stratification. We conduct these studies using the Dedalus pseudospectral framework. We extend prior studies of overshoot substantially and find that the depth of overshoot in DNS simulations of a typical plume is well-predicted by a simple buoyancy equilibration model. The implications of this model, extended into the stellar regime, are that very little overshoot should occur under solar conditions. This would seem to sharply limit the role of the tachocline within the global solar dynamo.

1:43PM R10.00002 Sustained shear flows in stratified convection , EVAN ANDERS, TAYLER

QUIST, BENJAMIN BROWN, University of Colorado - Boulder, JEFFREY OISHI, Bates College — Highly stratified convection is ubiquitous among natural systems including planetary atmospheres and stellar envelopes. Here we study fully compressible convection within plane-parallel, polytropically stratified layers using the Dedalus pseudospectral framework at moderate to high Rayleigh number. We find that large scale shearing "zonal" flows can naturally arise in such systems, as previously found in incompressible Rayleigh-Benard convection. These zonal winds can strongly influence the dynamics of convection. We study the forces that drive and dissipate large-scale shear flows and the bulk properties of sustained flows. We find naturally occurring shear flows at moderate aspect ratio and explore methods to achieve similar, convectively-driven shear flows at larger aspect ratios.

1:56PM R10.00003 The stability of buoyancy-driven gaseous boundary layers over inclined semi-infinite hot plates, PRABAKARAN RAJAMANICKAM, WILFRIED COENEN, ANTONIO L SANCHEZ,

University of California San Diego — The free-convective boundary-layer flow that develops over a semi-infinite inclined hot plate is known to become unstable at a finite distance from the leading edge, characterized by a critical value of the Grashof number Gr_δ based on the local boundary-layer thickness δ . The character of the instability depends on the inclination angle ϕ , measured from the vertical direction. For values of ϕ below a critical value ϕ_c the instability is characterized by the appearance of spanwise vortices, whereas for $\phi > \phi_c$ the bifurcated flow displays Görtler-like streamwise vortices. The Boussinesq approximation, employed in previous linear stability analyses, ceases to be valid for gaseous flow when the wall-to-ambient temperature ratio $\theta_w = T_w/T_\infty$ is not close to unity. The corresponding non-Boussinesq analysis is presented here, accounting also for the variation with temperature of the different transport properties. The base-flow profiles are used in a parallel-flow temporal stability analysis to delineate the dependence of the critical Grashof numbers Gr_δ on the inclination angles ϕ and on the temperature ratio θ_w . The analysis provides in particular the values of the crossover inclination angles $\phi_c(\theta_w)$.

2:09PM R10.00004 The formation of thermohaline staircases for large salt concentration differences in double diffusive convection¹, 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 — In the upper layers of the tropical and subtropical ocean, step-like mean profiles for both temperature and salinity are often observed, a phenomenon referred to as thermohaline staircase. It consists of alternatively stacked mixing layers, and finger layers with sharp gradients in both mean temperature and salinity. It is believed that thermohaline staircases are caused by double diffusive convection (DDC), i.e. the convection flow with fluid density affected by two different scalars. Here we conducted direct numerical simulations of DDC bounded by two parallel plates and aimed to realise the multi-layer state similar to the oceanic thermohaline staircase. We applied an unstable salinity difference and a stable temperature difference across the two plates. We gradually increased the salinity Rayleigh number Ra_S , i.e. the strength of salinity difference, and fixed the relative strength of temperature difference. When Ra_S is high enough the flow undergoes a transition from a single finger layer to a triple layer state, where one mixing layer emerges between two finger layers. Such triple layer state is stable up to the turbulent diffusive time scale. The finger-layer height is larger for higher Ra_S . The dependences of the scalar fluxes on Ra_S were also investigated.

¹Supported by Dutch FOM Foundation and NWO programme MCEC; Computing resources from SURFSara and PRACE project 2015133124

2:22PM R10.00005 Drifting localized structures in doubly diffusive convection, EDGAR KNOBLOCH, University of California at Berkeley, DAVID LO JACONO, ALAIN BERGEON, IMFT, Université de Toulouse, UPS-INP — We use numerical continuation to compute a multiplicity of spatially localized states in doubly diffusive convection in a vertical slot driven by imposed horizontal temperature and concentration differences. The calculations focus on the so-called opposing case, in which the resulting gradients are in balance. No-slip boundary conditions are used at the sides and periodic boundary conditions with large spatial period are used in the vertical direction. This system exhibits homoclinic snaking of stationary spatially localized structures with point symmetry [1,2]. In this talk we demonstrate the existence, near threshold, of drifting pulses of spatially localized convection that appear when mixed concentration boundary conditions are used, and use homotopic continuation to identify similar states in the case of fixed concentration boundary conditions. We show that these states persist to large values of the Grasshof number and provide a detailed study of their properties. [1] A. Bergeon & E. Knobloch, Phys. Fluids 20, 034102 (2008). [2] C. Beaume, A. Bergeon & E. Knobloch, Phys. Fluids 25, 114102 (2013).

2:35PM R10.00006 A reduced model for salt-fingering convection in the small Lewis number limit¹, JIN-HAN XIE, EDGAR KNOBLOCH, Univ of California - Berkeley — We derive a reduced model that captures key features of salt-fingering convection, including secondary instabilities, in the asymptotic limit of small Lewis number and large flux ratio. In the infinite Prandtl number limit, this model combines a prognostic equation for the evolution of the salinity field with a novel diagnostic relation between the streamfunction and salinity. When the salinity and temperature Rayleigh numbers Ra_S and Ra_T are large, simulations reveal the existence of statistically steady saturated states, characterized by fluxes and kinetic energy that scale as powers of $(Ra_S/Ra_T) - 1$. Three distinguished regimes are identified: a weakly nonlinear regime and two strongly nonlinear regimes characterized by distinct exponents. The processes responsible for saturation are described in detail and the probability density function of the saturated fields is determined.

¹This work was supported by the National Science Foundation under grant No. DMS-1317666.

2:48PM R10.00007 Numerical studies of a confined volatile binary fluid subject to a horizontal temperature gradient¹, TONGRAN QIN, ROMAN GRIGORIEV, Georgia Institute of Technology — Our fundamental understanding of convection in a layer of nonisothermal binary fluid with free surface in the presence of noncondensable gases, such as air, is still limited. In relatively thick liquid layers, the flow is driven by a combination of three different forces: buoyancy, thermocapillarity, and solutocapillarity in the liquid layer. Unlike buoyancy, both thermocapillarity and solutocapillarity depend sensitively on the boundary conditions at the liquid-vapor interface. Recent experimental studies showed that the composition of both the liquid and the gas phases have significant effects on the convection pattern. In particular, in a methanol-water mixture, four different flow regimes were identified on a map spanned by the concentration of methanol in the liquid and the concentration of air in the gas, which are thermocapillarity-dominated flow (TDF), solutocapillarity-dominated flow (SDF), unsteady flow (UF) and reversed flow (RF). This talk will present a comprehensive numerical model for a confined volatile binary fluid subject to a horizontal temperature gradient in the presence of noncondensable gases, and illustrate how the composition of both phases affect thermocapillarity and solutocapillarity. The numerical results will also be compared with experiments.

¹Supported by NSF

3:01PM R10.00008 Convection-driven dynamos in the limit of rapid rotation, MICHAEL CALKINS, LOUIE LONG, DAVID NIEVES, KEITH JULIEN, Univ of Colorado - Boulder, STEVEN TOBIAS, University of Leeds — Most large-scale planetary magnetic fields are thought to be driven by rapidly rotating convection. Direct numerical simulation (DNS) remains an important tool for investigating the physics of dynamos, but remains severely restricted in parameter space relative to geo- and astrophysical systems. Asymptotic models provide a complimentary approach to DNS that have the ability to access planetary-like magnetohydrodynamical regimes. We utilize an asymptotic dynamo model to investigate the influence of convective flow regime on dynamo action. We find that the spatial characteristics of the large-scale magnetic field are dependent only weakly on changes in flow behavior. In contrast, the behavior of the small-scale magnetic field is directly dependent on, and therefore shows significant variations with, the small-scale convective flow field. These results may suggest why many previous DNS studies, which reside in a vastly different parameter space relative to planets, are nonetheless successful in reproducing many of the observed features of planetary magnetic fields.

3:14PM R10.00009 Direct measurement of the heat exchanges in a buoyancy driven two-phase flow; application to the planets core formation, JEAN-BAPTISTE WACHEUL, MICHAEL LE BARS, IRPHE UMR 7342 CNRS, Aix-Marseille Univ and ECM, Marseille, France, ECOULEMENT TOURNANT ET GEOPHYSIQUE TEAM — Telluric planet formation involved the settling of large amounts of liquid iron coming from impacting planetesimals into a viscous magma ocean as deep as thousands of kilometers. During this "iron rain", the initial state of planets was mostly determined by exchanges of heat and elements between the two phases. Most models of planet formation simply assume that the metal rapidly equilibrated with the whole mantle. Here we report the results of experiments on which we performed measurements of the diffusive exchanges integrated during the fall, in addition to measuring the dynamical variables of the flow on high speed videos recordings. Using a balloon filled with liquid gallium alloy as an analogue for the iron core of the impactor and a viscous fluid as an analogue for the silicate magma, we were able to produce flows matching the dynamical regime of the geophysical inspiration. We find that the early representations of this flow as an iron "rain" is far from the experiments, both in terms of fluid mechanics and diffusive exchanges during the phase where most of the equilibration is accomplished. Indeed, the equilibration coefficient at a given depth depends both on the size of the metal diapir and on the viscosity of the ambient fluid, whereas the falling speed is only controlled by the size. Various scalings chosen in the literature for the diffusive exchanges, and we find good agreement with the hypothesis and scaling of a turbulent thermal.

3:27PM R10.00010 Chemically reacting fluid flow in exoplanet and brown dwarf atmospheres, BAYLEE BORDWELL, BENJAMIN P. BROWN, Department of Astrophysical and Planetary Sciences, University of Colorado Boulder, JEFFREY S. OISHI, Department of Physics and Astronomy, Bates College — In the past few decades, spectral observations of planets and brown dwarfs have demonstrated significant deviations from predictions in certain chemical abundances. Starting with Jupiter, these deviations were successfully explained to be the effect of fast dynamics on comparatively slow chemical reactions. These dynamical effects are treated using mixing length theory in what is known as the "quench" approximation. In these objects, however, both radiative and convective zones are present, and it is not clear that this approximation applies. To resolve this issue, we solve the fully compressible equations of fluid dynamics in a matched polytropic atmosphere using the state-of-the-art pseudospectral simulation framework Dedalus. Through the inclusion of passive tracers, we explore the transport properties of convective and radiative zones, and verify the classical eddy diffusion parameterization. With the addition of active tracers, we examine the interactions between dynamical and chemical processes using abstract chemical reactions. By locating the quench point (the point at which the dynamical and chemical timescales are the same) in different dynamical regimes, we test the quench approximation, and generate prescriptions for the exoplanet and brown dwarf communities.

Tuesday, November 22, 2016 1:30PM - 2:48PM –
Session R12 Multiphase Flows: Level Set Method and Applications C123 - Mario Trujillo, University of Wisconsin-Madison

1:30PM R12.00001 A Performance Comparison Between a Level Set Method and an Unsplit Volume of Fluid Method, OLIVIER DESJARDINS, ROBERT CHIODI, Cornell Univ, MARK OWKES, Montana State Univ — The simulation of high density ratio liquid-gas flows presents many numerical difficulties due to the necessity to track the interface and the discontinuities in physical properties associated with the interface. Two main categories of methods used to track the interface are level set methods and volume of fluid (VOF) methods. In particular, conservative level set methods track and transport the interface using a scalar field, with the interface profile represented by a hyperbolic tangent function of a finite thickness. Volume of fluid methods, on the other hand, store the percentage of each fluid in the computational cells. Both methods offer distinct advantages, however, the strengths and weaknesses of each method relative to each other have yet to be thoroughly investigated. This work compares the accuracy and computational efficiency for an accurate conservative level set method and an unsplit VOF method using canonical test cases, such as Zalesak's disk, the deformation of a circle, and the deformation of a sphere. The mass conservation and ability to correctly predict instability for a more complex case of an air-blast atomization of a planar liquid layer will also be presented.

1:43PM R12.00002 Comparing volume of fluid and level set methods for evaporating liquid-gas flows, JOHN PALMORE, OLIVIER DESJARDINS, Cornell University — This presentation demonstrates three numerical strategies for simulating liquid-gas flows undergoing evaporation. The practical aim of this work is to choose a framework capable of simulating the combustion of liquid fuels in an internal combustion engine. Each framework is analyzed with respect to its accuracy and computational cost. All simulations are performed using a conservative, finite volume code for simulating reacting, multiphase flows under the low-Mach assumption. The strategies used in this study correspond to different methods for tracking the liquid-gas interface and handling the transport of the discontinuous momentum and vapor mass fractions fields. The first two strategies are based on conservative, geometric volume of fluid schemes using directionally split and un-split advection, respectively. The third strategy is the accurate conservative level set method. For all strategies, special attention is given to ensuring the consistency between the fluxes of mass, momentum, and vapor fractions. The study performs three-dimensional simulations of an isolated droplet of a single component fuel evaporating into air. Evaporation rates and vapor mass fractions are compared to analytical results.

1:56PM R12.00003 Numerical Simulation of Two-phase flow with Phase Change Using the Level-set Method, HONGYING LI, JING LOU, LUNSHENG PAN, IHPC A*STAR, YITFATT YAP, Department of Mechanical Engineering, The Petroleum Institute — Multiphase flow with phase change is widely encountered in many engineering applications. A distinct feature involves in these applications is the phase transition from one phase to another due to the non-uniform temperature distribution. Such kind of process generally releases or absorbs large amount of energy with mass transfer happened simultaneously. It demands great cautions occasionally such as the high pressure due to evaporation. This article presents a numerical model for simulation of two-fluid flow with phase change problem. In these two fluids, one of them changes its state due to phase change. Such a problem then involves two substances with three phases as well as two different interfaces, i.e. the interface between two substances and the interface of one substance between its two phases. Two level-set functions are used to capture the two interfaces in the current problem. The current model is validated against one-dimensional and two-dimensional liquid evaporation. With the code validated, it is applied to different phase change problems including (1) a falling evaporating droplet and the rising of one bubble and (2) two-fluid stratified flow with solidification of one fluid. Comparisons on the bubble and droplet topologies, flow and temperature fields are made for the first case between the falling evaporating droplet and the falling droplet without evaporation. For the second demonstration case, the effect of the superheated temperature on the solidification process is investigated.

2:09PM R12.00004 Stabilized Conservative Level Set Method with Adaptive Wavelet-based Mesh Refinement, NAVID SHERVANI-TABAR, OLEG V. VASILYEV, Univ of Colorado - Boulder —

This paper addresses one of the main challenges of the conservative level set method, namely the ill-conditioned behavior of the normal vector away from the interface. An alternative formulation for reconstruction of the interface is proposed. Unlike the commonly used methods which rely on the unit normal vector, Stabilized Conservative Level Set (SCLS) uses a modified renormalization vector with diminishing magnitude away from the interface. With the new formulation, in the vicinity of the interface the reinitialization procedure utilizes compressive flux and diffusive terms only in the normal direction to the interface, thus, preserving the conservative level set properties, while away from the interfaces the directional diffusion mechanism automatically switches to homogeneous diffusion. The proposed formulation is robust and general. It is especially well suited for use with adaptive mesh refinement (AMR) approaches due to need for a finer resolution in the vicinity of the interface in comparison with the rest of the domain. All of the results were obtained using the Adaptive Wavelet Collocation Method, a general AMR-type method, which utilizes wavelet decomposition to adapt on steep gradients in the solution while retaining a predetermined order of accuracy.

2:22PM R12.00005 Gradient Augmented Level Set Method for Two Phase Flow Simulations with Phase Change, C.R. LAKSHMAN ANUMOLU, MARIO F. TRUJILLO, University of Wisconsin-Madison

— A sharp interface capturing approach is presented for two-phase flow simulations with phase change. The Gradient Augmented Levelset method is coupled with the two-phase momentum and energy equations to advect the liquid-gas interface and predict heat transfer with phase change. The Ghost Fluid Method (GFM) is adopted for velocity to discretize the advection and diffusion terms in the interfacial region. Furthermore, the GFM is employed to treat the discontinuity in the stress tensor, velocity, and temperature gradient yielding an accurate treatment in handling jump conditions. Thermal convection and diffusion terms are approximated by explicitly identifying the interface location, resulting in a sharp treatment for the energy solution. This sharp treatment is extended to estimate the interfacial mass transfer rate. At the computational cell, a d-cubic Hermite interpolating polynomial is employed to describe the interface location, which is locally fourth-order accurate. This extent of subgrid level description provides an accurate methodology for treating various interfacial processes with a high degree of sharpness. The ability to predict the interface and temperature evolutions accurately is illustrated by comparing numerical results with existing 1D to 3D analytical solutions.

2:35PM R12.00006 Multiscale level-set method for accurate modeling of two-phase immiscible flow with deposited thin-films on solid surfaces, MOTAZ ABU ALSAUD, Stanford Univ, AMIR RIAZ, University of Maryland, College Park, HAMDI TCHELEPI, Stanford Univ, UNIVERSITY OF MARYLAND, COLLEGE PARK COLLABORATION, STANFORD UNIVERSITY TEAM

— We developed a multiscale sharp interface method based on the level-set for two-phase immiscible flow with pre-existing thin-films on solid surfaces. The lubrication approximation theory is used to model the thin-film equation efficiently. The incompressible Navier-Stokes, level-set, and thin-film evolution equation are coupled sequentially to capture the physics occurring at multiple length scales. The proposed multiscale method is validated through comparison with the augmented Young-Laplace equation that includes the Van der Waals intermolecular force for a static meniscus in a capillary tube. The viscous bending in the advancing interface over precursor film problem is captured by the numerical method and agrees with the Cox-Voinov theory. The problem of a moving-bubble inside a capillary tube is modeled, and the results compare well with both theory and experiments. In addition, the performance of the new approach is assessed by studying the spurious currents for capillary-dominated flows at low capillary numbers. The method is applicable for flows with a capillary number as low as $Ca = 10^6$.

Tuesday, November 22, 2016 1:30PM - 3:40PM –

Session R13 DFD GPC: Geophysical Fluid Dynamics: Cryosphere C124 - Colin Meyer,
Harvard University

1:30PM R13.00001 Meltwater percolation and refreezing in compacting snow¹, COLIN MEYER, Harvard University, IAN HEWITT, Oxford University

— Meltwater is produced on the surface of glaciers and ice sheets when the seasonal surface energy forcing warms the ice above its melting temperature. This meltwater percolates through the porous snow matrix and potentially refreezes, thereby warming the surrounding ice by the release of latent heat. Here we model this process from first principles using a continuum model. We determine the internal ice temperature and glacier surface height based on the surface forcing and the accumulation of snow. When the surface temperature exceeds the melting temperature, we compute the amount of meltwater produced and lower the glacier surface accordingly. As the meltwater is produced, we solve for its percolation through the snow. Our model results in traveling regions of meltwater with sharp fronts where refreezing occurs. We also allow the snow to compact mechanically and we analyze the interplay of compaction with meltwater percolation. We compare these models to observations of the temperature and porosity structure of the surface of glaciers and ice sheets and find excellent agreement. Our models help constrain the role that meltwater percolation and refreezing will have on ice-sheet mass balance and hence sea level.

¹Thanks to the 2016 WHOI GFD Program, which is supported by the National Science Foundation and the Office of Naval Research

1:43PM R13.00002 Formation of wind-driven ice bridges in narrow straits, BHARGAV RALLABANDI, ZHONG ZHENG, Princeton University, MICHAEL WINTON, NOAA/Geophysical Fluid Dynamics Laboratory, HOWARD A. STONE, Princeton University

— An ice bridge is a static arch made of tightly packed ice that can be formed when sea ice flows through a narrow strait between landmasses. The formation of a stable ice arch prevents the further flow of sea ice into warmer oceans, and therefore plays an important role in the regulation of the local climate and ecology and to an extent, the mass balance of Arctic ice. While ice bridges are a seasonal phenomenon in many parts of the Canadian Archipelago, the process of their formation and breakup is poorly understood. Using thin-layer theory along with dynamic sea ice models widely used in climate modeling, we develop a reduced-order description of wind-driven ice bridge formation in long, narrow straits. Our theory predicts a critical static condition for arrested flow that involves the ice properties (thickness and compactness), the geometry of the channel, and the magnitude of the wind stress. Further, we show that in a channel of varying shape and under a constant wind stress, a spatially uniform ice field evolves towards a steady state with discontinuities in its properties, consistent with observed mechanisms of ice bridge formation. The reduced-order model thus provides a predictive tool for the flow and stoppage of sea ice in straits.

1:56PM R13.00003 Snowflakes aggregation in turbulent flows: a case limit under dynamically critical Stokes conditions, MICHELE GUALA, St. Anthony Falls Laboratory, CEGE, University of Minnesota, JIARONG HONG, St. Anthony Falls Laboratory, ME, University of Minnesota — A simple theory, based on observations of snowflake distribution in a turbulent flow, is proposed to model the growth of inertial particles as a result of dynamic clustering at scales larger than the Kolmogorov length scale. Particles able to stick or coalesce are expected to grow in size in flow regions where preferential concentration is predicted by a critical Stokes number $St = \tau_p/\tau_f \simeq 1$. We postulate that, during growth, St remains critical, with the particle response time τ_p evolving according to the specific flow time scale τ_f defined by the vortices around which progressively larger particles end up orbiting, colliding and aggregating. This mechanism leads to the prediction of the limiting size of droplets and snowflakes in a turbulent flow. Such limit, determined by the extent of the turbulent inertial range, can be formulated as a function of the r.m.s. velocity fluctuation and the integral length scale. The proposed dynamically critical Stokes growth provides a framework to interpret hydrometeor aggregation and, in general, particle size growth in geophysical multi-phase flows.

2:09PM R13.00004 Snowflake Impact on the Air-Sea Interface, DAVID MURPHY, University of South Florida — The air-sea interface is the site of globally important exchanges of mass, momentum, and heat between the sea and atmosphere. These climate-driving exchanges occur through small-scale processes such as bubble entrainment and bursting, raindrop impact, and wind-wave creation. The physics of snowflakes falling on the sea surface has not been previously considered. High speed imaging of natural snowflakes of characteristic size up to 6.5 mm falling at a mean speed of 1 m/s into an aquarium of chilled seawater reveals a complex multiphase flow. Snowflakes impacting and crossing the air-seawater interface appear to entrain a thin air film which forms micro-bubbles as the snowflake melts. Large, morphologically complex snowflakes may entrain hundreds of micro-bubbles which are up to 0.15 mm in diameter. Large snowflakes melt milliseconds after entry and subsequently form a downward-moving vortex ring of freshwater, evident from the motion of the bubbles it contains, which may penetrate up to 16 mm below the surface. Buoyant freshwater and bubbles then rise, with larger bubbles escaping from the downward flow more quickly than the smaller bubbles. The dissolution and popping of these bubbles represent previously unrecognized sources of air-sea gas transfer and marine aerosol droplet creation, respectively.

2:22PM R13.00005 The formation of grounding zone wedges, KATARZYNA KOWAL¹, GRAE WORSTER, DAMTP, University of Cambridge — Ice sheets are generally lubricated by a layer of sub-glacial sediment, or till, which plays a central role in determining their large-scale dynamics. Sub-glacial till has been found to accumulate into distinctive sedimentary wedges at ice-sheet grounding zones, separating floating ice shelves from grounded ice sheets. These grounding-zone wedges have important implications for stabilizing ice sheets against grounding-zone retreat in response to rising sea levels. We develop a theoretical model of wedge formation in which we treat both ice and till as viscous fluids spreading under gravity into an inviscid ocean and present a fluid-mechanical explanation of the formation of these wedges in terms of the jump in hydrostatic loading and unloading of till across the grounding zone. We also conduct a series of fluid-mechanical experiments in a confined setting in which we find that the underlying layer of less viscous fluid accumulates spontaneously in a similar wedge-shaped region at the experimental grounding line. We also extend our theory to more natural, unconfined settings in two dynamical regimes in which the overlying ice is resisted dominantly either by vertical shear or by extensional stresses and compare our findings with available geophysical data.

¹Currently at Northwestern University

2:35PM R13.00006 Sidewall-driven convection in a thermally and compositionally stratified fluid, KEATON BURNS, GLENN FLIERL, Massachusetts Institute of Technology, ANDREW WELLS, University of Oxford — We present direct numerical simulations of incompressible turbulent convection along a heated sidewall in a thermally and compositionally stratified fluid, as a simplified model of meltwater flows along marine-terminating glaciers. Our model considers a 2D domain that is horizontally bounded and vertically periodic, with constant background thermal and compositional buoyancy gradients. We apply a fixed thermal perturbation along one sidewall, driving upward convective plumes and horizontally spreading layers with compensating thermal and compositional buoyancy perturbations. We examine the formation and structure of these layers as the background stratification is varied from thermally to compositionally dominated, and as the sidewall is tilted away from vertical. We also examine the variations in heat flux along the sidewall that arise with the layers.

2:48PM R13.00007 Ice sheets on plastically-yielding beds¹, IAN HEWITT, University of Oxford — Many fast flowing regions of ice sheets are underlain by a layer of water-saturated sediments, or till. The rheology of the till has been the subject of some controversy, with laboratory tests suggesting almost perfectly plastic behaviour (stress independent of strain rate), but many models adopting a pseudo-viscous description. In this work, we consider the behaviour of glaciers underlain by a plastic bed. The ice is treated as a viscous gravity current, on a bed that allows unconstrained slip above a critical yield stress. This simplified description allows rapid sliding, and aims to investigate 'worst-case' scenarios of possible ice-sheet disintegration. The plastic bed results in an approximate ice-sheet geometry that is primarily controlled by force balance, whilst ice velocity is determined from mass conservation (rather than the other way around, as standard models would hold). The stability of various states is considered, and particular attention is given to the pace at which transitions between unstable states can occur. Finally, we observe that the strength of basal tills depends strongly on pore pressure, and combine the model with a description of subglacial hydrology. Implications for the present-day ice sheets in Greenland and Antarctica will be discussed.

¹Funding: ERC Marie Curie FP7 Career Integration Grant

3:01PM R13.00008 Turbulent convection and dissolution under sloping Ice-shelves in Saline water, MAINAK MONDAL, BISHAKHADATTA GAYEN, The Australian National University, ROSS GRIFFITHS, Retired, ROSS KERR, The Australian National University — We have carried out numerical experiments with geophysically relevant slope angles (5° - 20°) over domains of $O(1\text{ m})$ - $O(10\text{ m})$ bracketing Antarctic conditions. Contact of the ice with cold and saltywater drives a turbulentbuoyant flow predominantly due to freshening from melting ice-interface in the upslope direction. The dissolution rate decreases with shallower angles and is predicted by our theory. Thickness of the thermal and salinity boundary layer increases with decreasing slope angle whereas the interface conditions remains insensitive to the inclination. The dissolution rate is independent of slope length when boundary layer is turbulent. For steeper angles turbulent kinetic energy is mainly produced by the buoyancy flux but its contribution rapidly decreases with shallower angles. For shallower slopes turbulence is sustained by shear production due to decrease of velocity boundary layer. Our results can be used to explain many dynamical processes under inclined ice shelves around Antarctica.

3:14PM R13.00009 Glacial uplift: fluid injection beneath an elastic sheet on a poroelastic substrate, JEROME NEUFELD, DUNCAN HEWITT, University of Cambridge, GREG CHINI, University of New Hampshire — Supraglacial lakes can drain to the base of glaciers extremely rapidly, causing localised uplift of the surrounding glacier and affecting its sliding velocity. The means by which large volumes of drained water interact with and leak into the subglacial hydrological system is unclear, as is the role of the basal till. A theoretical study of the spread of fluid injected below an elastic sheet (the ice) is presented, where the ice lies above, and initially compresses, a deformable poroelastic layer. As pressurized fluid is injected, the deformable layer swells to accommodate more fluid. If sufficient fluid is injected, a 'blister' of fluid forms above the layer, causing the overburden to lift off the base. The flow is controlled by the local pressure drop across the tip of this blister, which depends subtly on both the flow of fluid through the porous layer below the tip, and on poroelastic deformation in the till ahead of the tip. The spreading behaviour and dependence on key parameters is analysed. Predictions of the model are compared to field measurements of uplift from draining glacial lakes in Greenland.

3:27PM R13.00010 An Experimental Investigation of Ice-melting and heat transfer rates from submerged warm water jets upward impinging into ice-blocks as analogous for water-filled cavities formed during subglacial eruptions.., HAMIDREZA JAMSHIDNIA, MAGNUS TUMI GUDMUNDSSON, University of Iceland — Rates of energy transfer in water-filled cavities formed under glaciers by geothermal and volcanic activity are investigated by conducting experiments in which hot water jets (10- 90C) impinging into an ice block for jet Reynolds numbers in turbulent regime of 10000 -70000. It is found that heat flux is linearly dependent on jet flow temperature. Water jet melts a cavity into an ice block. Cavities had steep to vertical sides with a doming roof. Some of ice blocks used had trapped air bubbles. In these cases that melting of the ice could have led to trapping of air at the top of cavity, partially insulating the roof from hot water jet. The overall heat transfer rate in cavity formation varied with jet temperature from $<100 \text{ kW m}^{-2}$ to $\sim 900 \text{ kW m}^{-2}$ while melting rates in the vertical direction yield heat transfer rates of 200-1200 kW m^{-2} . Experimental heat transfer rates can be compared to data on subglacial melting observed for ice cauldrons in Iceland. For lowest temperatures the numbers are comparable to those for geothermal water in cool, subglacial water bodies and above subglacial flowpaths of jökulhlaups. Highest experimental rates for 80-90C jets are 3-10 times less than inferred from observations of recent subglacial eruptions ($2000-4000 \text{ kW m}^{-2}$). This can indicate that single phase liquid water convection alone may not be sufficient to explain the rates seen in recent subglacial eruptions, suggesting that forced 2 or 3 phase convection can be common.

Tuesday, November 22, 2016 1:30PM - 3:40PM –
Session R14 Waves: Surface Gravity Waves and Shear Currents C125-126 - Miguel Bustamante, University College Dublin

1:30PM R14.00001 Precession resonance mechanism in deep-water gravity surface waves¹, MIGUEL BUSTAMANTE, University College Dublin, DAN LUCAS, University of Cambridge — Discovered by Bustamante *et al.* in 2014 and published in *Phys. Rev. Lett.* in the same year², precession resonance is a mechanism whereby strong nonlinear energy transfers occur between modes of oscillations whose frequencies are detuned: the amplitude-dependent precession frequencies of the phases help restore the resonance, hence the name "precession resonance". After explaining how this mechanism works and how robust it is, we will discuss new applications of this effect in systems of technological interest, focusing on deep-water gravity surface waves. We report transfer efficiencies of up to 40%, depending on the numerical-experimental setup. All evidence gathered so far points to the conclusion that, to leading order, this effect is dominated by triad interactions at small (but finite) amplitudes. Joint work with Dan Lucas (DAMTP, Cambridge).

¹Financially supported by Science Foundation Ireland (SFI) under research grant no. 12/IP/1491.

²M.D.Bustamante, B. Quinn and D. Lucas, *Phys. Rev. Lett.* **113**, 084502 (2014)

1:43PM R14.00002 On the interaction of gravity-capillary lumps in deep water, NAEEM MASNAIDI, JAMES DUNCAN, University of Maryland — The nonlinear response of a water surface to a pressure source moving at a speed just below the minimum phase speed of linear gravity-capillary waves in deep water ($c_{min} = 23.1 \text{ cm/s}$) consists of periodic generation of pairs of three-dimensional solitary waves (lumps) in a V-shaped pattern downstream of the source. In the reference frame of the laboratory, these unsteady lumps propagate in a direction oblique to the motion of the source and are damped by viscosity. In the current study, the interaction of lumps generated by two equal strength pressure sources moving side by side in parallel straight lines is investigated experimentally via photography-based techniques. The first lump generated by each source, collides with the lump from the other source in the center-plane of the two sources. It is observed that a steep depression is formed during the collision. Soon after the collision, this depression radiates energy in the form of small-amplitude radial waves. After the radiation, a quasi-stable pattern is formed with several rows of localized depressions that are qualitatively similar to lumps but exhibit periodic oscillations in depth, similar to a "breather". The shape of the wave pattern and the period of oscillations depend strongly on the distance between the sources.

1:56PM R14.00003 ABSTRACT WITHDRAWN –

2:09PM R14.00004 ABSTRACT WITHDRAWN –

2:22PM R14.00005 An experimental study of wave coupling in gravity surface wave turbulence¹, QUENTIN AUBOURG, Univ Joseph Fourier, JOEL SOMMERIA, SAMUEL VIBOUD, CNRS, NICOLAS MORDANT, Univ Joseph Fourier — Weak turbulence is a theoretical framework aimed at describing wave turbulence (in the weakly nonlinear limit) i.e. a statistical state involving a large number of nonlinearly coupled waves. For gravity waves at the surface of water, it provides a phenomenology that may describe the formation of the spectrum of the ocean surface. Analytical predictions of the spectra are made based on the fact that energy transfer occurs through 4-wave coupling. By using an advanced stereoscopic imaging technique, we measure in time the deformation of the water surface. We obtain a state of wave turbulence by using two small wedge wavemakers in a 13-m diameter wavetank. We then use high order correlator (bi- and tri-coherence) in order to get evidence of the active wave coupling present in our system as used successfully for gravity-capillary wave turbulence [1]. At odds with the weak turbulence theory we observe 3-wave interaction involving 2 quasi linear wave and a bound wave whose frequency lies on the first harmonics of the linear dispersion relation. We do not observe 4-wave coupling within the accuracy of our measurement.

[1] Aubourg & Mordant, *Phys. Rev. Fluids* 1, 2016

¹This project has received funding from the European Research Council (ERC) under the European Union's Horizon 2020 research and innovation programme (grant agreement No 647018-WATU)

2:35PM R14.00006 Numerical simulation of the capillary-gravity waves excited by an obstacle, HIDESHI HANAZAKI, RYOSUKE INOMATA, Kyoto University — Capillary gravity waves excited by an obstacle are investigated by the unsteady numerical solution of the Euler equations. It is well known that the large-amplitude upstream advancing solitary waves are generated periodically under the resonant condition of $Fr=1$ (Fr : Froude number), i.e., when the phase velocity of the long surface waves agrees with the mean flow speed. With capillary effects ($Bo>0$), short waves are newly generated by the upstream solitary waves of large amplitude. In this study it is investigated how the characteristics of the solitary waves and the short waves, especially their amplitudes, change due to the variation of the obstacle height and the Froude number. The results will be compared also with the solutions of the forced KdV-type equations.

2:48PM R14.00007 Kinematic criterion for breaking of shoaling waves¹, DAN LIBERZON, URI ITAY, Technion - Israel Institute of Technology — Validity of a kinematic criterion for breaking of shoaling waves was examined experimentally. Results obtained by simultaneous measurements of water surface velocity by PTV and of the propagation velocity of a steep crest up to the point of breaking inception during shoaling will be reported. The experiments performed in a large wave tank examining breaking behavior of gentle spillers during shoaling on three different slopes suggest a validity of the recently proposed kinematic criterion. The breaking inception was found to occur when the horizontal velocity of the water surface on the steep (local steepness of 0.41-0.6) crest reaches a threshold value of 0.85-0.95 of that of the crest propagation. The exact moment and position of breaking inception detected using a Phase Time Method (PTM), characterizing a unique shape of the local frequency fluctuations at the inception. Future implementation of the PTM method for detection of breaking events in irregular wave fields will be discussed.

¹Supported by German-Israeli Foundation for Scientific Research and Development (GIF) grant 2019392

3:01PM R14.00008 The generation of symmetric and asymmetric lump solitons by a bottom topography¹, ZHIMING LU, Shanghai University — A group of Lump solutions to the (2+1)-dimensional Kadomtsev-Petviashvili (KP) equation is obtained analytically by making use of Hirota bilinear transform method. Then the generation of symmetric and asymmetric lump solitons by an obliquely-placed three-dimensional bottom topography is numerically investigated using the forced Kadomtsev-Petviashvili-I (fKP-I) equation. The numerical method is based on the third order Runge-Kutta method and the Crank-Nicolson scheme. The main result is the asymmetric generation of asymmetric lump-type solitons downstream of the obstacle. The lump soliton with a smaller amplitude is generated with a longer period and moves in a larger angle with respect to the positive x-axis than the one with a larger amplitude. The amplitude of the lump solitons strongly depend on the volume of the obstacle rather than the shape. Finally the effects of the detuning parameter on the generation of lump solitons is also studied.

¹Project supported by NSFC with No. 11272196

3:14PM R14.00009 Identifying features of Kelvin ship wakes via spectrogram analysis¹, SCOTT MCCUE, RAVINDRA PETHIYAGODA, TIMOTHY MORONEY, Queensland University of Technology — A method for observing and measuring ship wakes is to employ an echo sounder to record the surface elevation over time as a ship passes nearby. The resulting output signal corresponds to the cross-section of the ship wake taken in the direction of travel. The surface elevation at the echo sounder can be visualised as a spectrogram through the use of many short-time discrete Fourier transforms. In this study, we identify and explain features of spectrograms of ship wakes, concentrating on the differing effects that linearity and nonlinearity have on the wave time-frequency signal. These results have the potential to contribute to practical scenarios in which spectrograms are used to calculate the energy contained within a given ship wake and the effect that the propagating wake wash will have when it interacts with the coastal zone.

¹We acknowledge support of the Australian Research Council via the Discovery Project DP140100933

3:27PM R14.00010 Random coupling of acoustic-gravity waves in the atmosphere, CHRISTOPHE MILLET, CEA, DAM, DIF, FRANCOIS LOTT, LMD, ENS, CHRISTOPHE HAYNES, University of Queensland — In numerical modeling of long-range acoustic propagation in the atmosphere, the effect of gravity waves on low-frequency acoustic waves is often ignored. As the sound speed far exceeds the gravity wave phase speed, these two types of waves present different spatial scales and their linear coupling is weak. It is possible, however, to obtain relatively strong couplings via sound speed profile changes with altitude. In the present study, this scenario is analyzed for realistic gravity wave fields and the incident acoustic wave is modeled as a narrow-banded acoustic pulse. The gravity waves are represented as a random field using a stochastic multiwave parameterization of non-orographic gravity waves. The parameterization provides independent monochromatic gravity waves, and the gravity wave field is obtained as the linear superposition of the waves produced. When the random terms are retained, a more generalized wave equation is obtained that both qualitatively and quantitatively agrees with the observations of several highly dispersed stratospheric wavetrains. Here, we show that the cumulative effect of gravity wave breakings makes the sensitivity of ground-based acoustic signals large, in that small changes in the parameterization can create or destroy an acoustic wavetrain.

Tuesday, November 22, 2016 1:30PM - 3:27PM –
Session R15 Bio: Pumping/Bacterial Oil Remediation E143/144 - Jian Sheng, Texas AM University

1:30PM R15.00001 Drinking with a hairy tongue: viscous entrainment by dipping hairy surfaces, ALICE NASTO, PIERRE-THOMAS BRUN, JOS ALVARADO, JOHN BUSH, ANETTE HOSOI, MIT — Nectar-drinking bats have tongues covered with hair-like papillae, enhancing their ability to take up viscous nectar by dipping. Using a combination of model experiments and theory reminiscent of Landau-Levich-Derjaguin dip coating, we rationalize this mechanism of viscous entrainment in a hairy texture. For the model experiments, hairy surfaces are fabricated using laser cut molds and casting samples with PDMS elastomer. Modeling the liquid trapped within the texture using a Darcy-Brinkman like approach, we derive the drainage flow solution. The amount of fluid that is entrained is dependent on the viscosity of the fluid, the density of the hairs, and the dipping speed. We find that there is an optimal hair density to maximize fluid uptake.

1:43PM R15.00002 A fully coupled bolus-esophageal-gastric model for esophageal emptying based on the immersed boundary method¹, WENJUN KOU, Theoretical and Applied Mechanics, Northwestern University, JOHN E. PANDOLFINO, PETER J. KAHRILAS, Feinberg School of Medicine, Northwestern University, NEELESH A. PATANKAR, Department of Mechanical Engineering, Northwestern University — In this work, we develop a fully coupled bolus-esophageal-gastric model to study esophageal emptying based on the immersed boundary method. The model includes an esophageal segment, an ellipsoid-shaped stomach, and a bolus. It can easily handle the passive and active function of the lower esophageal sphincter (LES). Two groups of case studies are presented. The first group is about the influence from tissue anisotropy. Simulation shows that the weaker (or more compliant) part suffers from a higher wall shear stress and higher pressure load when the bolus is filled in and emptied from the LES segment. This implies a degradation cycle in which a weaker tissue becomes much weaker due to an increased load, a possible pathway to the esophageal lower diverticulum. The second group is about bulge formation resulting from asymmetric anatomy and a compliant LES. In particular, we find a right bulge tends to develop for a compliant LES. The bulge is most pronounced with a highest stiffness of the gastric wall. This implies that the competition between the LES stiffness and gastric wall stiffness might be another factor related to the esophageal lower diverticulum.

¹The support of grant R01 DK56033 and R01 DK079902 from NIH is gratefully acknowledged.

1:56PM R15.00003 Pore Water Pumping by Upside-Down Jellyfish, MANIKANTAM GADDAM, ARVIND SANTHANAKRISHNAN, Oklahoma State University — Patchy aggregations of Cassiopea medusae, commonly called upside-down jellyfish, are found in sheltered marine environments with low-speed ambient flows. These medusae exhibit a sessile, non-swimming lifestyle, and are oriented such that their bells are attached to the substrate and oral arms point towards sunlight. Pulsations of their bells are used to generate currents for suspension feeding. Their pulsations have also been proposed to generate forces that can release sediment locked nutrients into the surrounding water. The goal of this study is to examine pore water pumping by Cassiopea individuals in laboratory aquaria, as a model for understanding pore water pumping in unsteady flows. Planar laser-induced fluorescence (PLIF) measurements were conducted to visualize the release of pore water via bell motion, using fluorescent dye introduced underneath the substrate. 2D particle image velocimetry (PIV) measurements were conducted on the same individuals to correlate PLIF-based concentration profiles with the jets generated by pulsing of medusae. The effects of varying bell diameter on pore water release and pumping currents will be discussed.

2:09PM R15.00004 Flow Dynamics of a smart pump: Mytilus Galloprovincialis, KAREM PEKKAN, FAZIL USLU, Koc University, BIOFLUIDS AND CARDIOVASCULAR FLUID MECHANICS LABORATORY TEAM — Hydrodynamic performance of marine mussel, *Mytilus Galloprovincialis*, is studied by the time-resolved particle image velocimetry (PIV). We evaluated inhalant flow, exhalant jet flow, pumping performances, and flow control capabilities of the mussels quantitatively. Inhalant flow structures of mussels are measured at the coronal plane first-time in literature. Nutrient fluid is convected into the mussel by three-dimensional sink type flow that is different than exhalant jet flow. Inhalant velocity reaches its highest magnitude inside of the mussel mantle while accelerating outward the mussel. We calculated pressure gradient at the coronal plane where three-dimensional sink type inhalant flow is observed. As inhalant flow approaches mussel shell tip, suction force generated by the inhalant flow increases. Likewise, unique exhalant jet flow regimes are studied for 17 mussels. Mussels can control their exhalant jet flow structure from single potential core region to double one or vice versa. Peak exhalant jet velocity generated by the mussels changes between 2.77 cm/s and 11.1 cm/s as a function of mussel cavity volume. Hydrodynamic dissipation at sagittal plane is calculated to evaluate whether there is any interference between inhalant sink flow and exhalant jet flow or not. Results showed an efficient synchronized pumping mechanism. This pumping mechanism can feature flow-turning angle, the angle between inhalant and exhalant jet flow, 90° with standard deviation of 16°.

2:22PM R15.00005 The effect of resonant driving and damping on dynamic suction pumping, NICHOLAS BATTISTA, LAURA MILLER, University of North Carolina at Chapel Hill — Impedance pumping (or dynamic suction pumping) drives flow through a flexible valveless tube with a single region of actuation. It is a profoundly complex pumping mechanism given that the flow velocities and directions generated depend nonlinearly upon the driving frequency, material properties, duty factor, and location of the actuation point. Given the simplicity of its actuation, it is used in biomedical devices and is thought to generate flow in a number of biological systems. In this study, we numerically simulate an elastic tube with mass using the immersed boundary method and explore the performance when it is driven over a range of frequencies and damping factors. Flow is maximized during resonance, and bulk transport is minimal when the tube is over-damped.

2:35PM R15.00006 The effect of lymphatic valve morphology on fluid transport¹, ALEXANDER ALEXEEV, MATTHEW BALLARD, ZHANNA NEPIYUSHCHIKH, BRANDON DIXON, Georgia Institute of Technology — The lymphatic vasculature is present in nearly all invertebrate tissue, and is essential in the transport of fluid and particles such as immune cells, antigens, proteins and lipids from the tissue to lymph nodes and to the venous circulation. Lymphatic vessels are made up of a series of contractile units that work together in harmony as "micro hearts" to pump fluid against a pressure gradient. Lymphatic valves are critical to this functionality, as they open and close with the oscillating pressure gradients from contractions, thus allowing flow in only one direction and leading to a net pumping effect. We use a hybrid lattice-Boltzmann lattice spring model which captures fluid-solid interactions through two-way coupling between a viscous fluid and lymphatic valves in a section of a lymphatic vessel to study the dynamics of lymphatic valves and their effect on fluid transport. Further, we investigate the effect of variations in valve geometry and material properties on fluid pumping. This work helps to increase our understanding of the mechanisms of lymphatic fluid transport, which has implications in a variety of pathologies, including cancer metastasis, autoimmunity, atherosclerosis and obesity.

¹Support from NSF CMMI 1635133 is gratefully acknowledged.

2:48PM R15.00007 Experimental Fluidic Investigation of Degradation of Pico-liter Oil Droplets by Physical and Biological Processes, MARYAM JALALI, JIAN SHENG, Texas A & M Univ — This study used laboratory experiments to assess degradation of crude oil by physical and biological processes including dissolution and consumption. To perform this study, we have developed a bioassay that consists of a flow chamber with a bottom glass substrate printed with an array of pico-liter oil droplets using micro-Transfer Printing. The technique allows the printing of highly homogeneous pico-liter droplet array with different dimensions and shapes that can be maintained for weeks. Since the droplets are pinned and stationary on the bottom substrate, the key processes can be evaluated by measuring the change of shape and volume using Atomic Force Microscopy. Parallel microfluidic bioassays are established at the beginning, exposed to abiotic/biotic solutions, and scarified for characterization at given time intervals for each experiment. Two processes, dissolution and consumption, are investigated. In addition, the effects of dispersant on these processes are also studied. The results show that the amount of oil degraded by bacteria accounts for almost 50% of the total volume in comparison to 25% via dissolution. Although dispersant has a subtle effect on dissolution, the effect on rates of consumption and its asymptotic behavior are substantial. Experiments involving different bacterial strains, dispersant concentration, and flow shear rate are on-going.

3:01PM R15.00008 Biodegradation of crude oil dispersions by marine bacteria , GABRIEL

JUAREZ, UIUC, VICENTE FERNANDEZ, ROMAN STOCKER, ETH Zurich — Dispersants are used to break up marine oil slicks and increase the available surface area for bacteria to degrade oil hydrocarbons. However, this common view neglects key elements of the microscale interactions between bacteria and oil droplets, namely encounters and growth. Utilizing experimental observations of bacteria colonizing oil droplets, we model the interactions affecting hydrocarbon consumption between a collection of oil droplets with varying sizes and a single bacterial pool. The results show that degradation time is minimized for intermediate droplet sizes and that reducing droplet size too much can lead to years in increased degradation time. This mechanical model provides a baseline for understanding oil biodegradation and mitigation strategies in open marine systems.

3:14PM R15.00009 Quantifying oil degradation processes by flow, microbes and dispersant using digital holographic interferometry and micro-bioassay , JIAN SHENG, Tesas A & M Univ, Johns Hopkins Univ, MARYAM JALALI, Tesas A & M Univ, LARRY BROCK , Johns Hopkins Univ — The unceasing demand of hydrocarbons has led and will lead to the future events of releasing crude into marine environment like Deep Horizon oil spill. The burning question to scientific community after the spill was the fate of oil spill especially with high concentration of dispersant. It is found that various physical processes such as wind, wave, turbulence, compounded with dispersants, break oil into suspension of micro-droplets. It is widely accepted that dispersant reduces interfacial tension and results in increased surface to volume ratio and subsequently improve biodegradation. Due to complexity of oil composition, key mechanisms differ substantially from well-studied laboratory system, especially in the presence of other environmental factors such as flow shear and microbes. To investigate these mechanisms at oil water interface qualitatively, we have developed a micro-bioassay consisting of microfluidics with a substrate printed with oil droplet array and a digital holographic interferometer (DHI). The degradation of micro-droplets is evaluated with the change of shape and volume measured in real time by DHI at a 2-minute interval over 100 hours. Time resolved experiments are performed to study effects of droplet size, dispersant concentrations, flow shear, and different bacteria species on the rate of degradation. The details on the rate and mechanisms will be provided in the talk.

Tuesday, November 22, 2016 1:30PM - 3:40PM –

Session R16 Bio: Respiratory System D133/134 - Luciano Castillo, Texas Tech University

1:30PM R16.00001 Experimenatal analysis of the effect of cartilaginous rings on human tracheobronchial flow , JOSE MONTOYA SEGNINI, University of Turabo, HUMBERTO BOCANEGRAS EVANS, LUCIANO CASTILLO, Texas Tech University — We present a set of high-resolution PIV experiments carried out in a refractive index-matched model of a trachea with cartilage rings at $Re \approx 2800$. Results show a higher vorticity along the walls of the trachea in the model with cartilaginous rings as well as small recirculation areas on the upstream side of the wall cavities created by the rings. Furthermore, the ringed model experiences higher shear stress in the trachea due to the sudden change in the wall position created by the rings. Additionally, small recirculation areas are identified in the cavities between rings. For the smooth model, a stronger separation bubble is observed at the bronchi entrance, generating a stronger shear layer and increasing the wall shear stress on the bottom bronchi wall. The differences observed go against the notion that the main airway, i.e. trachea and main bronchi, may be modeled as smooth. Our results suggest that cartilage rings will have an impact on the wall shear stress and may affect particle deposition, which is of importance in inhaled drug delivery and pollutant deposition in the airway. Additionally, the effects introduced by the rings may change the flow characteristics in further generations.

1:43PM R16.00002 ABSTRACT WITHDRAWN –

1:56PM R16.00003 Vortical Structures in CT-based Breathing Lung Models , JIWOONG CHOI, CHANGHYUN LEE, Seoul Natl Univ, ERIC HOFFMAN, CHING-LONG LIN, University of Iowa — The 1D-3D coupled computational fluid dynamics (CFD) lung model is applied to study vortical structures in the human airways during normal breathing cycles. During inhalation, small vortical structures form around the turbulent laryngeal jet and Taylor-Görtler-like vortices form near the curved walls in the supraglottal region and at airway bifurcations. On exhalation elongated vortical tubes are formed in the left main bronchus, whereas a relatively slower stream is observed in the right main bronchus. These structures result in helical motions in the trachea, producing long lasting high wall shear stress on the wall. The current study elucidates that the correct employment of image-based airway deformation and lung deflation information is crucial for capturing the physiologically consistent regional airflow structures. The pathophysiological implications of these structures in destruction of tracheal wall will be discussed.

2:09PM R16.00004 ABSTRACT WITHDRAWN –

2:22PM R16.00005 Relationship between Pulmonary Airflow and Resistance in Patients with Airway Narrowing Using An 1-D Network Resistance and Compliance Model¹ , SANGHUN CHOI, JIWOONG CHOI, ERIC HOFFMAN, CHING-LONG LIN, The University of Iowa — To predict the proper relationship between airway resistance and regional airflow, we proposed a novel 1-D network model for airway resistance and acinar compliance. First, we extracted 1-D skeletons at inspiration images, and generated 1-D trees of CT unresolved airways with a volume filling method. We used Horsfield order with random heterogeneity to create diameters of the generated 1-D trees. We employed a resistance model that accounts for kinetic energy and viscous dissipation (Model A). The resistance model is further coupled with a regional compliance model estimated from two static images (Model B). For validation, we applied both models to a healthy subject. The results showed that Model A failed to provide airflows consistent with air volume change, whereas Model B provided airflows consistent with air volume change. Since airflows shall be regionally consistent with air volume change in patients with normal airways, Model B was validated. Then, we applied Model B to severe asthmatic subjects. The results showed that regional airflows were significantly deviated from air volume change due to airway narrowing. This implies that airway resistance plays a major role in determining regional airflows of patients with airway narrowing.

¹Support for this study was provided, in part, by NIH Grants U01 HL114494, R01 HL094315, R01 HL112986, and S10 RR022421.

2:35PM R16.00006 Effects of lung disease on the three-dimensional structure and air flow pattern in the human airway tree, TRISTAN VAN DE MOORTELE, Univ of Minnesota , ANDRAS NEMES, CHRISTINE WENDT, FILIPPO COLETTI, University of Minnesota — The morphological features of the airway tree directly affect the air flow features during breathing, which determines the gas exchange and inhaled particle transport. Lung disease, Chronic Obstructive Pulmonary Disease (COPD) in this study, affects the structural features of the lungs, which in turn negatively affects the air flow through the airways. Here bronchial tree air volume geometries are segmented from Computed Tomography (CT) scans of healthy and diseased subjects. Geometrical analysis of the airway centerlines and corresponding cross-sectional areas provide insight into the specific effects of COPD on the airway structure. These geometries are also used to 3D print anatomically accurate, patient specific flow models. Three-component, three-dimensional velocity fields within these models are acquired using Magnetic Resonance Imaging (MRI). The three-dimensional flow fields provide insight into the change in flow patterns and features. Additionally, particle trajectories are determined using the velocity fields, to identify the fate of therapeutic and harmful inhaled aerosols. Correlation between disease-specific and patient-specific anatomical features with dysfunctional airflow patterns can be achieved by combining geometrical and flow analysis.

2:48PM R16.00007 Flow characteristics in the airways of a COPD patient with a saber-sheath trachea, DOHYUN JIN, HAECHEON CHOI, Seoul National University, CHANGHYUN LEE, JIWOONG CHOI, Seoul National University College of Medicine, KWANGGI KIM, National Cancer Center — The chronic obstructive pulmonary disease (COPD) is a lung disease characterized by the irreversible airflow limitation caused by the damaged small airways and air sacs. Although COPD is not a disease of the trachea, many patients with COPD have saber-sheath tracheas. The effects of this morphological change in the trachea geometry on airflow are investigated in the present study. An unstructured finite volume method is used for the simulations during tidal breathing in normal and COPD airways, respectively. During inspiration, local large pressure drop is observed in the saber-sheath region of the COPD patient. During expiration, vortical structures are observed at the right main bronchus of the COPD airway, while the flow in the normal airway remains nearly laminar. High wall shear stress exists at convex regions of both airways during inspiration and expiration. However, due to the morphological changes in the COPD airway, relatively higher wall shear stress is observed in the patient airways.

3:01PM R16.00008 The Effects of High Frequency Oscillatory Flow on Particles' Deposition in Upper Human Lung Airways, JEREMY BONIFACIO¹, HAMID RAHAI², SHAHAB TAHERIAN³, CEERS/COE/California State University, Long Beach — The effects of oscillatory inspiration on particles' deposition in upper airways of a human lung during inhalation/exhalation have been numerically investigated and results of flow characteristics, and particles' deposition pattern have been compared with the corresponding results without oscillation. The objective of the investigation was to develop an improved method for drug delivery for Asthma and COPD patients. Previous clinical investigations of using oral airway oscillations have shown enhanced expectoration in cystic fibrosis (CF) patients, when the frequency of oscillation was at 8 Hz with 9:1 inspiratory/expiratory (I:E) ratio. Other investigations on oscillatory ventilation had frequency range of 0.5 Hz to 2.5 Hz. In the present investigations, the frequency of oscillation was changed between 2 Hz to 10 Hz. The particles were injected at the inlet and particle velocity was equal to the inlet air velocity. One-way coupling of air and particles was assumed. Lagrangian phase model was used for transport and depositions of solid 2.5 micron diameter round particles with 1200 kg/m³ density. Preliminary results have shown enhanced PM deposition with oscillatory flow with lower frequency having a higher deposition rate

¹Graduate Assistant

²Professor and Director

³Research Associate

3:14PM R16.00009 Surfactants and the Mechanics of Respiration., ABDULRAHMAN JBAILY, ANDREW J. SZERI, University of California, Berkeley — Alveoli are small sacs found at the end of terminal bronchioles in human lungs with a mean diameter of 200 μm . A thin layer of fluid (hypophase) coats the inner face of an alveolus and is in contact with the air in the lungs. The thickness of this layer varies among alveoli, but is in the range of 0.1 to 0.5 μm for many portions of the alveolar network. The interfacial tension σ at the air-hypophase interface tends to favor collapse of the alveolus, and resists its expansion during inhalation. Type II alveolar cells synthesize and secrete a mixture of phospholipids and proteins called pulmonary surfactant. These surfactant molecules adsorb to the interface causing σ to decrease. For example, σ of water at body temperature is $\approx 70 \text{ mN/m}$ and falls to an equilibrium value of $\approx 25 \text{ mN/m}$ when surfactants are present. Also, in a dynamic sense, it is known that σ is reduced to near 0 during exhalation when the surfactant film compresses. In this work, the authors develop a mechanical and transport model of the alveolus to study the effect of surfactants on various aspects of respiration. The model is composed of three principal parts: (i) air movement into and out of the alveolus; (ii) a balance of linear momentum across the two-layered membrane of the alveolus (hypophase and elastic wall); and (iii) a pulmonary surfactant transport problem in the hypophase. The goal is to evaluate the influence of pulmonary surfactant on respiratory mechanics.

3:27PM R16.00010 Circular flow patterns induced by ciliary activity in reconstituted human bronchial epithelium.¹, ANNIE VIALLAT, CNRS, KAMEL KHELLOUFI, PlatOd, DELPHINE GRAS, PASCAL CHANEZ, Aix Marseille univ, AIX MARSEILLE UNIV., CNRS, CINAM, MARSEILLE, FRANCE TEAM, AIX MARSEILLE UNIV., CNRS, INSERM, LAI, MARSEILLE, FRANCE TEAM — Mucociliary clearance is the transport at the surface of airways of a complex fluid layer, the mucus, moved by the beats of microscopic cilia present on epithelial ciliated cells. We explored the coupling between the spatial organisation and the activity of cilia and the transport of surface fluids on reconstituted cultures of human bronchial epithelium at air-liquid interface, obtained by human biopsies. We reveal the existence of stable local circular surface flow patterns of mucus or Newtonian fluid at the epithelium surface. We find a power law over more than 3 orders of magnitude showing that the average ciliated cell density controls the size of these flow patterns, and, therefore the distance over which mucus can be transported. We show that these circular flow patterns result from the radial linear increase of the local propelling forces (due to ciliary beats) on each flow domain. This linear increase of local forces is induced by a fine self-regulation of both cilia density and orientation of ciliary beats. Local flow domains grow and merge during cilogenesis to provide macroscopic mucus transport. This is possible only when the viscoelastic mucus continuously exerts a shear stress on beating cilia, revealing a mechanosensitive function of cilia.

¹M. K. Khelloufi thanks the society MedBioMed for financial support. This work was supported by the ANR MUCOCIL project, grant ANR-13-BSV5-0015 of the French Agence Nationale de la Recherche.

Tuesday, November 22, 2016 1:30PM - 3:40PM –
Session R17 Reacting Flows: Experiments II D131 - Chenning Tong, Clemson University

1:30PM R17.00001 Fire Whirls, Vortex Breakdown(?), and Blue Whirls¹ , ELAINE ORAN,

HUAHUA XIAO, MICHAEL GOLLNER, University of Maryland — As we were investigating the efficiency of fire-whirl burning on water, we observed the usual transformation of a pool fire to a fire whirl, and then suddenly, we saw the fire undergo a third transition. A blue cup appeared around the base of the fire whirl, surrounding the yellow flame, the yellow flame receded into the cup and finally disappeared. What remained was a small, rapidly spinning blue flame that burned until the fuel on the water was consumed. The blue whirl was shaped like a spinning cup, closed at the bottom near the water surface, and spreading in radius moving upwards towards the rim. Above the blue cup lip, there was a purple cone-shaped mist. The fuel was usually n-heptane, but at one point it was crude oil, and still the blue whirl formed naturally. The height of the fire whirl on the laboratory pan was larger than a half meter, and this evolved into a blue whirl about 48 cm high. Occasionally the blue whirl would become “unstable and revert to a transitional state of blue cup holding a yellow flame. When the blue whirl formed, turbulence seemed to disappear, and the flame became quiet. We will show videos of how this happened and discuss the evolution of the fire whirl to the blue whirl in vortex-breakdown concepts.

¹This work was supported by and EAGER award from NSF and Minta Martin Endowment Funds in the Department of Aerospace Engineering at the University of Maryland.

1:43PM R17.00002 Time-resolved PIV investigation of flashback in stratified swirl flames of hydrogen-rich fuel¹ , RAKESH RANJAN, NOEL CLEMENS, The University of Texas at Austin — Hydrogen is

one of the promising alternative fuels to achieve greener power generation. However, susceptibility of flashback in swirl flames of hydrogen-rich fuels acts as a major barrier to its adoption in gas turbine combustors. The current study seeks to understand the flow-flame interaction during the flashback of the hydrogen-rich flame in stratified conditions. Flashback experiments are conducted with a model combustor equipped with an axial swirler and a center-body. Fuel is injected in the main swirl flow via the fuel ports on the swirler vanes. To achieve mean radial stratification, these fuel ports are located at a radial location closer to the outer wall of the mixing tube. Stratification in the flow is assessed by employing Anisole PLIF imaging. Flashback is triggered by a rapid increase in the global equivalence ratio. The upstream propagation of the flame is investigated by employing time-resolved stereoscopic PIV and chemiluminescence imaging. Stratification leads to substantially different flame propagation behavior as well as increased flame surface wrinkling.

¹We gratefully acknowledge the sponsorship by the DOE NETL under grant DEFC2611-FE0007107.

1:56PM R17.00003 The effect of stratification on premixed swirl-flame flashback by using porous center-body injection¹ , ANDREW MCCASLIN, RAKESH RANJAN, NOEL CLEMENS, Univ of Texas,

Austin — Boundary layer flashback must be prevented in order to stably operate stationary gas turbines. One strategy to avoid flashback is to create equivalence-ratio stratification, such as by reducing the fuel/air ratio in the boundary layer below the flammability limit. Typically, stratification is achieved by using radially non-uniform fuel injection. The goal of the current study is to reduce the propensity of flashback in a premixed annular swirl combustor that uses a premix section with center-body. A porous metal center-body (10 micron pore size) is used to bleed air directly into the boundary layer and thus locally reduce the equivalence ratio. Planar laser-induced fluorescence imaging of anisole-seeded flow is carried out to assess the stratification in the flow. Time-resolved PIV and chemiluminescence imaging are used to investigate flashback at atmospheric pressure conditions. A comparative study between fully premixed and stratified flame flashback is conducted to determine how stratification influences flashback physics.

¹This work was sponsored by the DOE NETL under grant DEFC2611-FE0007107. This source of funding is gratefully acknowledged.

2:09PM R17.00004 Investigation of lean combustion stability and pressure drop in porous media burners , SADAF SOBHANI, Stanford University, BRET HALEY, DAVID BARTZ, ALZETA Corporation, JARED DUNNMON, Stanford University, JOHN SULLIVAN, ALZETA Corporation, MATTHIAS IHME, Stanford University — The stability and thermal durability of combustion in porous media burners (PMBs) is examined experimentally and computationally. For this, two burner

concepts are considered, which consist of different pore topologies, porous materials, and matrix arrangements. Long-term material durability tests at constant and cycled on-off conditions are performed, along with a characterization of combustion stability, pressure drop and pollutant emissions for a range of equivalence ratios and mass flow rates. Experimental thermocouple temperature measurements and pressure drop data are presented and compared to results obtained from one-dimensional volume-averaged simulations. Experimental and model results show reasonable agreement for temperature profiles and pressure drop evaluated using Ergun's equations. Enhanced flame stability is illustrated for burners with Yttria-stabilized Zirconia Alumina upstream and Silicon Carbide in the downstream combustion zone. Results reinforce concepts in PMB design and optimization, and demonstrate the potential of PMBs to overcome technological barriers associated with conventional free-flame combustion technologies.

2:22PM R17.00005 ABSTRACT WITHDRAWN —

2:35PM R17.00006 Influence of Aerodynamic Strain Rate on Local Extinction in Turbulent Non-premixed Jet Flames¹ , ARAVIND RAMACHANDRAN, VENKATESWARAN NARAYANASWAMY,

KEVIN LYONS, North Carolina State University — 2-D velocity field measurements obtained from Particle Image Velocimetry (PIV) are used to obtain aerodynamic strain rate information in regions of local extinction in lifted turbulent non-premixed methane jet flames in coflow. Diluting the coflow to reduce the oxygen molefraction results in increased occurrences of local extinction. Statistical analysis is performed to correlate regions of high local strain rate with local extinctions in both air coflow and diluted coflow cases to study the influence of strain rate against vortical structures in extinguishing the flame front. A comparison is also made with heated and vitiated coflow cases, where autoignition is a flame stabilization mechanism and influenced by local strain rate. At high jet exit velocities ($U_x \gg U_r$), the out-of-plane strain rate component can be neglected but the convection of extinguished pockets into the measurement plane needs to be resolved by stereoscopic (3-D) measurements which will be done in a future work.

¹This work has been supported by the U.S. Army Research Office (Contracts W911NF1210140 and W911NF1610087) Dr. Ralph Anthenien, Technical Monitor, ARO.

2:48PM R17.00007 Shear Layer Interactions in the Helical Hydrodynamic Structures of Swirling, Reacting Jets, TRAVIS SMITH, Georgia Inst of Tech, KIRAN MANOHARAN, Indian Institute of Science, BENJAMIN EMERSON, Georgia Inst of Tech, SANTOSH HEMCHANDRA, Indian Institute of Science, TIM LIEUWEN, Georgia Inst of Tech — Swirling jets with density stratification are a canonical combustor flow field. This work consisted of coupled experimental and theoretical analysis of the spatial structure of the most amplified modes in an annular jet, with a specific focus on the radial mode shapes of the shear layer disturbances, which we characterize as inner shear layer (ISL) motion relative to outer shear layer (OSL) motion. The stability analysis identifies spatial structures dominated by ISL motion, modes dominated by OSL motion, and modes with mixed ISL and OSL motion. These mixed modes are further classified as sinuous or varicose radial structures, depending on the relative motions of the two shear layers. The presence and spatial dependencies of these spatial modes are demonstrated experimentally with a 5 kHz stereo PIV measurement of a reacting swirling jet. In the experiment, we demonstrate that external excitations of various spatial configurations can be used to elicit hydrodynamic responses of axisymmetric and helical motions in either the ISL, the OSL, or the sinuous or varicose radial modes.

3:01PM R17.00008 Effects of mean shear and scalar initial length scale on three-scalar mixing in turbulent coaxial jets¹, CHENNING TONG, WEI LI, MENGYUAN YUAN, Clemson Univ, CAMPBELL CARTER, AFRL — We investigate three-scalar mixing in a turbulent coaxial jet, in which a center jet and an annular flow, consisting of acetone-doped air and ethylene respectively, are mixed with the co-flow air. We investigate the effects of the velocity and length scale ratios of the annular flow to the center jet. Planar laser-induced fluorescence and Rayleigh scattering are employed to image the scalars. The results show that the velocity ratio alters the relative mean shear rates in the mixing layers between the center jet and the annular flow and between the annular flow and the co-flow, modifying the scalar fields through mean-flow advection, turbulent transport, and small-scale mixing. The length scale ratio determines the degree of separation between the center jet and the co-flow. The results show that while varying the velocity ratio can alter the mixing characteristics qualitatively, varying the annulus width only has quantitative effects. The evolution of the mean scalar profiles are dominated by the mean-flow advection, while the shape of the joint probability density function is largely determined by the turbulent transport and molecular diffusion. The results in the present study have implications for understanding and modeling multiscalar mixing in turbulent reactive flows.

¹Supported by NSF

3:14PM R17.00009 Characterizing Laminar Flame Interactions with Turbulent Fluidic Jets and Solid Obstacles for Turbulence Induction.¹, JESSICA CHAMBERS, STEPHEN GERDTS, KAREEM AHMED, Propulsion and Energy Laboratory of Central Florida — A detonation engine's fundamental design concept focuses on enhancing the Deflagration to Detonation Transition (DDT), the process through which subsonic flames accelerate to form a spontaneous detonation wave. Flame acceleration is driven by turbulent interactions that expand the reaction zone and induce mixing of products and reactants. Turbulence in a duct can be generated using solid obstructions, fluidic obstacles, duct angle changes, and wall skin friction. Solid obstacles have been previously explored and offer repeatable turbulence induction at the cost of pressure losses and additional system weight. Fluidic jet obstacles are a novel technique that provide advantages such as the ability to be throttled, allowing for active control of combustion modes. The scope of the present work is to expand the experimental database of varying parameters such as main flow and jet equivalence ratios, fluidic momentum ratios, and solid obstacle blockage ratios. Schlieren flow visualization and particle image velocimetry (PIV) are employed to investigate turbulent flame dynamics throughout the interaction. Optimum conditions that lead to flame acceleration for both solid and fluidic obstacles will be determined.

¹American Chemical Society

3:27PM R17.00010 Experimental testing of a liquid bipropellant rocket engine using nitrous oxide and ethanol diluted with water, JEFF PHILLIP, RUDY MORALES, STEWART YOUNGBLOOD, New Mexico Tech, W. VENNER SAUL, MARK GRUBELICH, Sandia National Laboratories, MICHAEL HARGATHER, New Mexico Tech — A research scale liquid bipropellant rocket engine testing facility was constructed at New Mexico Tech to perform research with various propellants. The facility uses a modular engine design that allows for variation of nozzle geometry and injector configurations. Initial testing focused on pure nitrous oxide and ethanol propellants, operating in the range of 5.5-6.9 MPa (800-1000 psi) chamber pressure with approximately 667 N (150 lbf) thrust. The system is instrumented with sensors for temperature, pressure, and thrust. Experimentally found values for specific impulse are in the range of 250-260 s which match computational predictions. Exhaust flow visualization is performed using high speed schlieren imaging. The engine startup and steady state exhaust flow features are studied through these videos. Computational and experimental data are presented for a study of dilution of the ethanol-nitrous oxide propellants with water. The study has shown a significant drop in chamber temperature compared to a small drop in specific impulse with increasing water dilution.

Tuesday, November 22, 2016 1:30PM - 3:27PM –
Session R18 Flow Instability: Richtmyer-Meshkov II D135 - Niranjan Ghaisas, Center for Turbulence Research, Stanford University

1:30PM R18.00001 Richtmyer-Meshkov flow of elastic-plastic solids using a high-order Eulerian framework, NIRANJAN S. GHASAS, Center for Turbulence Research, Stanford University, AKSHAY SUBRAMANIAM, Dept. of Aeronautics and Astronautics, Stanford University, SANJIVA K. LELE, Dept. of Aeronautics and Astronautics and Center for Turbulence Research, Stanford University — A high-order, fully Eulerian numerical framework is developed for tracking large, elastic-plastic deformations of solids coupled to fluids. Material interfaces are treated numerically using a diffuse-interface approximation. The numerical method is based on a 10th-order compact finite difference scheme for spatial discretization, a 4th-order Runge-Kutta time stepping method and a localized artificial diffusivity (LAD) method for regularizing shocks and material interfaces. This numerical framework was previously established for ideal gases and is extended in this study to liquids (stiffened gases) and solids. We establish the accuracy of our method by comparing to analytical results and demonstrate the superior resolution properties of our method by comparing to results of previous numerical studies that employed lower order methods. The effects of different equations of state and material stiffness parameters on the characteristics of the Richtmyer-Meshkov flow are investigated.

1:43PM R18.00002 ABSTRACT WITHDRAWN –

1:56PM R18.00003 Probabilistic events in shock driven multiphase hydrodynamic instabilities¹, WOLFGANG BLACK, University of Missouri - Columbia, NICK DENISSEN, Los Alamos National Laboratory, JACOB MCFARLAND, University of Missouri - Columbia — Multiphase flows are an important and complex topic of research with a rich parameter space. Historically many simplifications and assumptions have been made to allow simulation techniques to be applied to these systems. Some common assumptions include no particle-particle effects, evenly distributed particle fields, no phase change, or even constant particle radii. For some flows, these assumptions may be applicable but as the systems undergo complex accelerations and eventually become turbulent these multiphase parameters can create significant effects. Through the use of FLAG, a multiphysics hydrodynamics code developed at Los Alamos national laboratory, these assumptions can be relaxed or eliminated to increase fidelity and guide the development of experiments. This talk will build on our previous work utilizing simulations on the shock driven multiphase instability with a new investigation into a greater parameter space provided by additional multiphase effects; including a probabilistic particle field, various particle radii, and particle-particle effects on the evolution of commonly studied interfaces.

¹Los Alamos National Laboratory LA-UR-16-25652

2:09PM R18.00004 Growth-rate of Richtmyer-Meshkov instability for small and large amplitude initial perturbation¹, NORA SWISHER, ARUN PANDIAN, ZACHARY DELL, Carnegie Mellon University, ROBERT STELLINGWERF, Stellingwerf Consulting, SNEZHANA ABARZHI, Carnegie Mellon University — We study the effect of the amplitude of the initial perturbation on Richtmyer-Meshkov instability (RMI) by means of Smooth Particle Hydrodynamics simulations and by the rigorous theory and the newly developed empirical model. A broad parameter regime is analyzed. Initially, the interface has a single-mode sinusoidal perturbation with the amplitude varying from 0% to 100% of its wavelength. An empirical model is developed to describe the non-monotone dependence of the RMI growth-rate on the initial amplitude. The initial growth rate of the interface has a peak value. The position of the peak depends only weakly on the Mach and Atwood numbers, whereas the peak value depends strongly on Atwood number and weakly on Mach number. The ratio of initial growth rate to background velocity is related to the energy partitioning between the interface and the bulk. We find an upper bound of the ratio of the interfacial energy to the bulk energy, and identified its scaling with the Atwood number. This peak value of the energy ratio indicates that RM interfacial growth can be controlled by initial conditions.

¹The work is supported by the US National Science Foundation

2:22PM R18.00005 Dimensional crossover in Richtmyer-Meshkov flows¹, KATSUNOBU NISHIHARA, Osaka University, AKLANT K. BHOWMICK, SNEZHANA ABARZHI, Carnegie Mellon University — We analyze nonlinear dynamics of large scale coherent structures in Richtmyer-Meshkov flows. Group theory based analysis is applied with a detailed consideration of RM dynamics invariant with respect to p2mm (3D rectangular), p4mm (3D square) and pm1 (2D) groups. Symmetry dictates that asymptotic solutions form a 2 parameter family for rectangular flows and a 1 parameter family for 3D square and 2D flows. For 3D square and 2D symmetry, asymptotic solutions are obtained for the 1st and 2nd order of approximation and the fastest growth rate occurs at zero bubble curvatures. Fourier amplitudes exponentially decay with increase in order showing that solutions are convergent. Both 2D and 3D square solutions are stable with respect to symmetry conserving perturbations. Isotropic 3D square solutions are universally stable, while 2D solutions are unstable to anisotropic perturbations. Furthermore, the 3D and 2D solutions cannot be continuously transformed from one to another, and the dimensional crossover is discontinuous.

¹The work is supported by the US National Science Foundation

2:35PM R18.00006 Wave interference in Richtmyer-Meshkov flows¹, ROBERT F. STELLINGWERF, Stellingwerf Consulting, ARUN PANDIAN, SNEZHANA ABARZHI, Carnegie Mellon University — While it is a conventional wisdom that the initial conditions determine the linear and nonlinear dynamics of the Richtmyer-Meshkov (RM) flows, the research in this area is focused primarily on the effects of the wavelength and amplitude of the interface perturbation. The information is hitherto largely ignored about the influences on the evolution of Richtmyer-Meshkov instability (RMI) of the relative phase of a multi-wave perturbation and the interference of the perturbation waves. In this work we report a detailed study of confluence of effects of the relative phase as well as amplitudes of the interfacial waves on the structure of bubbles and spikes that is formed at the RM unstable interface after the shock passage. We show that the phase and the wave interference are important factors of the dynamics, because they influence the RM flow qualitatively and quantitatively, including the symmetry of the interface, the morphology of spikes and bubbles, and the RMI growth.

¹The work is supported by the US National Science Foundation

2:48PM R18.00007 Shock Driven Multiphase Instabilities in Scramjet Applications, JACOB MCFARLAND, The University of Missouri — Shock driven multiphase instabilities (SDMI) arise in many applications from dust production in supernovae to ejecta distribution in explosions. At the limit of small, fast reacting particles the instability evolves similar to the Richtmyer-Meshkov (RM) instability. However, as additional particle effects such as lag, phase change, and collisions become significant the required parameter space becomes much larger and the instability deviates significantly from the RM instability. In scramjet engines the SDMI arises during a cold start where liquid fuel droplets are injected and processed by shock and expansion waves. In this case the particle evaporation and mixing is important to starting and sustaining combustion, but the particles are large and slow to react, creating significant multiphase effects. This talk will examine multiphase mixing in scramjet relevant conditions in 3D multiphase hydrodynamic simulations using the FLASH code from the University of Chicago FLASH center.

3:01PM R18.00008 Two-fluid plasma Richtmyer-Meshkov instability¹, VINCENT WHEATELEY, DARYL BOND, School of Mechanical and Mining Engineering, The University of Queensland, DALE PULLIN, Graduate Aerospace Laboratories, California Institute of Technology, RAVI SAMTANEY, Mechanical Engineering, King Abdullah University of Science and Technology — The Richtmyer-Meshkov instability of a shock accelerated perturbed density interface is computationally investigated in the context of ideal two-fluid plasmas. This is accomplished by numerically solving separate sets of conservation equations for the ions and electrons, coupled to the full Maxwell's equations. We focus on cases without an imposed magnetic field and with Debye lengths ranging from a thousandth to a tenth of the interface perturbation wavelength. For all cases investigated, the behavior of the flow is substantially different from that predicted by the Euler or ideal magnetohydrodynamics equations. Electric fields generated by charge separation cause interface oscillations, particularly in the electrons, that drive a secondary high-wavenumber instability. Consequently, the density interface is substantially more unstable than predicted by the Euler equations for all cases investigated. Self-generated magnetic fields are predicted within our simulations, but their orientation is such that they do not dampen the Richtmyer-Meshkov instability.

¹This work was partially supported by the KAUST Office of Sponsored Research under Award URF/1/2162-01.

3:14PM R18.00009 Effect of a seed magnetic field on two-fluid plasma Richtmyer-Meshkov instability¹, DARYL BOND, VINCENT WHEATLEY, School of Mechanical and Mining Engineering, The University of Queensland, RAVI SAMTANEY, Mechanical Engineering, King Abdullah University of Science and Technology, DALE PULLIN, Graduate Aerospace Laboratories, California Institute of Technology — We investigate the effect of a uniform seed magnetic field on the plasma Richtmyer-Meshkov instability (RMI) using two-fluid simulations. These couple sets of conservation equations for the ions and electrons to the full Maxwell's equations. We consider cases where the seed magnetic field is normal to the interface and where the reference Debye length and Larmor radius range from a tenth to a thousandth of the interface perturbation wavelength. In ideal magnetohydrodynamics (MHD), it has been shown that in the presence of such a seed magnetic field, the growth of the RMI is suppressed by the transport of vorticity from the interface by MHD shocks. Our two-fluid plasma simulations reveal that while the RMI is suppressed in the presence of the seed field, the suppression mechanism varies depending on the plasma length-scales. Two-fluid plasma RMI simulations also reveal a secondary, high-wavenumber, electron-driven interface instability. This is not suppressed by the presence of the seed field.

¹This work was partially supported by the KAUST Office of Sponsored Research under Award URF/1/2162-01

Tuesday, November 22, 2016 1:30PM - 3:40PM – Session R19 Bio: Flapping D136 - Matthew Melius, Portland State University

1:30PM R19.00001 Giant larvaceans: biologically equivalent flapping flexible foils exhibit bending modes that enhance fluid transport¹, KAKANI KATIJA, ALANA SHERMAN, BRUCE ROBISON, 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 societally relevant challenges. A group of midwater organisms, known as giant larvaceans (genus *Bathochordaeus*), beat their tails to drive food and particle-laden water through complex, mucus filtering structures to feed. Giant larvaceans, whose motion and kinematics resemble flapping flexible foils, range in size from 1 to 10 cm in length, and can be found between the surface and 400 m. Using remotely-operated vehicles and DeepPIV, an instrument that enables in situ particle image velocimetry (PIV) measurements, the filtration rates and kinematics of giant larvaceans were investigated. These measurements yielded filtration rates for giant larvaceans as high as 80 L/hr, which exceeds expected filtration rates by a factor of 2 when compared with other larvacean groups. Comparing tail kinematics between *Bathochordaeus* and smaller larvaceans reveals differences in tail bending modes, where a hinge is present throughout the tail beat in giant larvaceans. Using laboratory PIV measurements with swimming animals and soft-bodied mechanical mimics, we reveal how these differences in tail kinematics can lead to enhanced fluid transport.

¹This work has been supported by the Packard Foundation.

1:43PM R19.00002 Mathematical modeling of flipping flaps and flinging fins in fluids¹, JEFF ELDREDGE, Univ of California - Los Angeles, XUANHONG AN, Illinois Institute of Technology, DARWIN DARAKANANDA, Univ of California - Los Angeles — Inviscid vortex models have served for decades as tools for distilling the physics of lifting and propulsive systems. In large-amplitude motions or massively separated flows, they lose some of their appeal due to the large number of vortex elements required to capture such flows with reasonable fidelity. However, in recent work (and in another talk at this conference by Darakananda et al.), we have shown that computational economy and physical fidelity can both be retained in a vortex model by using a heterogeneous set of vortex elements: vortex sheets of limited extent to capture the early formation of a new vortex structure, and a set of discrete vortices that represent developing and full-formed coherent structures. In this talk, we focus on the use of this hybrid vortex model for predicting interactions with flexible structures. By utilizing structures composed from linked rigid bodies, we can readily distinguish local added mass and vortex contributions along the body. We will demonstrate the overall model on two problems: the self-propulsion of a flexible plate due to rapid rotation about a pivot at the leading edge, and the enhancement of lift by the controlled pivot of a trailing-edge flap. We also discuss the use of such a model as a component in a dynamical observer.

¹AFOSR Grant FA9550-14-1-0328

1:56PM R19.00003 Force and Power Measurements of a Functionally-Graded Chordwise-Flexible Flapping Wing¹, DURLAV MUDBHARI, MALCOLM ERDOGAN, KEITH MOORED, Lehigh University — Flyers and swimmers flap their wings and fins to propel themselves efficiently over long distances. A key element to achieve their high performance is the flexibility of their appendages. While numerous studies have shown that homogeneously flexible wings can enhance force production and efficiency, animals actually have wings with varying flexural rigidity along their chord and span. The goal of this study is to understand and characterize the force production and energetics of functionally-graded, chordwise flexible wings. A flapping wing composed of a rigid and a flexible region, that define a chordwise gradient in flexural rigidity, is used to model functionally-graded materials. By varying the ratio of the lengths of the rigid to flexible regions, the flexural rigidity of the flexible region, and the flapping frequency, the thrust production of a functionally-graded wing is directly measured in a wind tunnel. A novel vacuum chamber apparatus is used in conjunction with the wind tunnel measurements to reliably measure the aerodynamic power input and the propulsive efficiency. Limited flow visualization is performed with particle image velocimetry in order to connect the force production and energetics of the partially-flexible wing with its generated flow structures.

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

2:09PM R19.00004 Flapping Wings of an Inclined Stroke Angle: Experiments and Reduced-Order Models in Dual Aerial/Aquatic Flight¹, JACOB IZRAELEVITZ, MICHAEL TRIANTAFYLLOU, MIRANDA KOTIDIS, Massachusetts Inst of Tech-MIT — Flapping wings in nature demonstrate a large force actuation envelope, with capabilities beyond the limits of static airfoil section coefficients. Puffins, guillemots, and other auks particularly showcase this mechanism, as they are able to both generate both enough thrust to swim and lift to fly, using the same wing, by changing the wing motion trajectory. The wing trajectory is therefore an additional design criterion to be optimized along with traditional aircraft parameters, and could possibly enable dual aerial/aquatic flight. We showcase finite aspect-ratio flapping wing experiments, dynamic similarity arguments, and reduced-order models for predicting the performance of flapping wings that carry out complex motion trajectories.

2:22PM R19.00005 Flapping dynamics of a flexible propulsor near ground.¹ , JAEHA

RYU, SUNG GOON PARK, BOYOUNG KIM, HYUNG JIN SUNG, KAIST, FLOW CONTROL LAB TEAM — The flapping motion of a flexible propulsor near the ground was simulated using the immersed boundary method. The hydrodynamic benefits of the propulsor near the ground were explored by varying the heaving frequency (St) of the leading edge of the flexible propulsor. The propulsion near the ground had some advantages to generate thrust and propel faster than that away from the ground. The mode analysis and the flapping amplitude along the Lagrangian coordinate were examined to analyze the kinematics as a function of the ground proximity (d) and the heaving frequency (St). The trailing edge amplitude (a_{tail}) and the net thrust (\bar{F}_x) were influenced by the heaving frequency (St) of the flexible propulsor. The hydrodynamic benefits of the flexible propulsor by the ground effect were discussed within the framework of dynamics and kinematics. The trailing edge amplitude presented the high-peak and the low-peak. Each peak showed distinct difference in terms of both the dynamics and the kinematics. The vortical structures in the wake were analyzed for different flapping conditions.

¹This work was supported by the Creative Research Initiatives (No. 2016-004749) program of the National Research Foundation of Korea (MSIP).

2:35PM R19.00006 Testing Momentum Enhancement of Ribbon Fin Based Propulsion Using a Robotic Model With an Adjustable Body , IAN ENGLISH, OSCAR CURET, None — Lighthill and Blake's 1990 momentum enhancement theory suggests there is a multiplicative propulsive effect linked to the ratio of body and fin heights in Gymnotiform and Balistiform swimmers, which propel themselves using multi-rayed undulating fins while keeping their bodies mostly rigid. Proof of such a momentum enhancement could have a profound effect on unmanned underwater vehicle design and shed light on the evolutionary advantage to body-fin ratios found in nature, shown as optimal for momentum enhancement in Lighthill and Blake's theory. A robotic ribbon fin with twelve independent fin rays, elastic fin membrane, and a body of adjustable height was developed specifically to experimentally test momentum enhancement. Thrust tests for various body heights were conducted in a recirculating flow tank at different flow speeds and fin flapping frequencies. When comparing thrust at different body heights, flow speeds, and frequencies to a 'no-body' thrust test case at each frequency and flow speed, data indicate there is no momentum enhancement factor due to the presence of a body on top of an undulating fin. This suggests that if there is a benefit to a specific ratio between body and fin height, it is not due to momentum enhancement.**2:48PM R19.00007 The effect of traveling wave shapes in the maneuver control and efficiency of an underwater robot propelled by an undulating fin¹ , HANLIN LIU, OSCAR CURET, Florida Atlantic Univ — Effective control of propulsive undulating fins has the potential to enhance the maneuverability and efficiency of underwater vehicles allowing them to navigate in more complex environments. Aquatic animals using this type of propulsion are able to perform complex maneuvers by sending different traveling waves along one or multiple elongated fins. Recent work has investigated the propulsive forces, the hydrodynamics and the efficiency of an undulating ribbon fin. However, it is still not understood how different traveling wave shapes along the fin can be used to control the hydrodynamic forces and torques to perform different maneuvers. In this work, we study the effect of traveling wave shapes on the hydrodynamic forces and torques, swimming speed, maneuver control and propulsive performance of an underwater vehicle propelled by an undulating fin. The underwater robot propels by actuating a fin that is composed of sixteen independent rays interconnected with a flexible membrane. The hull contains all the electronics, batteries, motors and sensors. The underwater vehicle was tested in a water tank-flume facility. In a series of experiments, we measured the motion of the vessel and the power consumption for different traveling wave patterns. In addition, we measured the flow around the fin using Particle Image Velocimetry. We present the results concerning the power distribution along the fin, propulsive efficiency, free-swimming speed and pitch control based on different fin kinematics.**

¹National Science Foundation under Grant No. 1420774

3:01PM R19.00008 A bioinspired modular aquatic robot , PHANINDRA TALLAPRAGADA, BEAU POLLARD, Mechanical Engineering, Clemson University — Several bio inspired swimming robots exist which seek to emulate the morphology of fish and the flapping motion of the tail and fins or other appendages and body of aquatic creatures. The locomotion of such robots and the aquatic animals that they seek to emulate is determined to a large degree by the changes in the shape of the body, which produce periodic changes in the momentum of the body and the creation and interaction of the vorticity field in the fluid with the body. We demonstrate an underactuated robot which swims due to the periodic changes in the angular momentum of the robot effected by the motion of an internal rotor. The robot is modular, unactuated tail like segments can be easily added to the robot. These segments modulate the interaction of the body with the fluid to produce a variety of passive shape changes that can allow the robot to swim in different modes.**3:14PM R19.00009 Unsteady Performance of Finite-Span Pitching Propulsors in Mixtures of Side-by-Side and In-Line Arrangements , MELIKE KURT, KEITH MOORED, Lehigh Univ — Birds, insects, and fish propel themselves by flapping their wings or oscillating their fins in unsteady motions. Many of these animals fly or swim in groups or collectives, typically described as flocks, swarms and schools. The three-dimensional steady flow interactions and the two dimensional unsteady flow interactions that occur in collectives are well characterized. However, the interactions that occur among three-dimensional unsteady propulsors remain relatively unexplored. The aim of the current study is to measure the forces acting on and the energetics of two finite-span pitching wings. The wings are arranged in mixtures of canonical in-line and side-by-side configurations while the phase delay between the pitching wings is varied. The thrust force, fluid-mediated interaction force between the wings and the propulsive efficiency are quantified. The three-dimensional interaction mechanisms are compared and contrasted with previously examined two-dimensional mechanisms. Stereoscopic particle image velocimetry is employed to characterize the three-dimensional flow structures along the span of the pitching wings.****3:27PM R19.00010 An autonomous sperm-like propulsor in a quiescent flow , BOYOUNG KIM, SUNG GOON PARK, HYUNG JIN SUNG, KAIST — Flapping motions of flexible fins 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. It is important to understand the physics of the flapping motions to utilize them for the biomimetic machines. In the present study, we introduce a sperm-like propulsor that consists of a rigid head containing genetic information and a flapping flexible tail for propulsion. The head gives a sinusoidal torque to the leading edge of the tail, and the flexible tail flaps along the leading edge. In other words, the sperm-like propulsor is moved by an oscillating relative angle between the head and the leading edge of the tail. Unlike self-propelled heaving and pitching fins, the 'autonomous' sperm-like propulsor has no prescribed motion or constraint referenced from outside coordinates. The penalty method and the immersed boundary method are used to solve the autonomous sperm-like propulsor in a quiescent flow. The cruising speed and the propulsive efficiency of the propulsor are explored as a function of the head size (D/L), the pitching angle (θ_0), the pitching frequency (f), and the distance from the wall (G/L).**

Tuesday, November 22, 2016 1:30PM - 3:01PM –

Session R20 Bio: General Topics III D137-138 - Laura Miller, University of North Carolina, Chapel Hill

1:30PM R20.00001 Passive aerial dispersal of insects and other arthropods, LAURA

MILLER, University of North Carolina Chapel Hill — One of the defining features of the aerial dispersal of tiny organisms is the ability to overcome negative buoyancy. This can be accomplished by dispersing in the right wind conditions (e.g. an updraft) or by active flight or active release. Once in the air, draggy structures, such as the draglines of spiders or bristled wings of tiny insects, can reduce the settling velocity and extend the time of transport. Purely passive mechanisms allow spiders and other arthropods to drift on strands of silk to heights of 14,000 m and distances of hundreds of miles. Similarly, tiny insects like thrips and parasitoid wasps can travel distances of thousands to tens of thousands of meters, possibly using a combination of periods of active and passive flight. In this presentation, we used the immersed boundary method to quantify settling velocities and transport dynamics of parachuting insects and other arthropods within a quiescent fluid, a uniform updraft, and eddies.

1:43PM R20.00002 ABSTRACT WITHDRAWN —**1:56PM R20.00003 Radial fingering at an active interface**, AMARENDER NAGILLA, IITB Monash

Research Academy, RANGANATHAN PRABHAKAR, Department of Mechanical and Aerospace Engineering, Monash University, Australia, SAMEER JADHAV, Chemical Engineering Department, Indian Institute of technology Bombay, India — It has been suggested that the shapes of single cells crawling on surfaces [Callan-Jones et al., Phys. Rev. Lett., 100:258106, 2008] and those of the fronts of thin layers of cells collectively expanding to close a wound [Mark et al., Biophys. J., 98:361-370, 2010] are the results of fingering instabilities. Motivated by these studies, we investigate the conditions under which an actively forced interface between a pair of immiscible viscous fluids will destabilize under Hele-Shaw confinement. The case of a circular active interface with surface tension and bending resistance is considered. Active forces exerted by the inner fluid at the interfacial region can be either completely internal or due to interactions with the confining substrate. In addition, the effects of cell growth or actin depolymerization or external injection of cell suspensions are modeled by including a distributed source and a point source of arbitrary strengths. Linear stability analysis reveals that at any given mean radius of the interface, its stability is dictated by two key dimensionless parameters. We discuss the different regions in a state space of these parameters.

2:09PM R20.00004 The Importance of Seed Characteristics in the Dispersal of Splash-Cup Plants, JOEL EKLOF, RACHEL PEPPER PEPPER, JULIANA ECHTERNACH, Univ of Puget Sound — Splash-

cup plants disperse their seeds by exploiting the kinetic energy of raindrops. When raindrops impact the splash-cup, a 3-5 mm vessel that holds seeds, the seeds are projected up to 1 m away from the parent plant. It has been established, using 3D printed models, that a 40cone angle maximizes dispersal distance when seeds are not present in the cup. We therefore use 40cups with the addition of different types of seeds to determine the effect that seeds of varying characteristics have on the dispersal and splash dynamics of splash-cup plants. Splash characteristics and dispersal distances of seeds with differing characteristics such as size, shape, texture, density, and hydrophobicity were compared to one another, as well as to the case of having no seeds present. We found that the presence of seeds dramatically decreased dispersal distance and changed splash characteristics (are measured by the angle and velocity of the resulting splash). In addition, different types of seeds yielded splashes with differing dispersal distance and splash characteristics. Splash characteristics and dispersal distances of glass beads of differing hydrophobicity were compared to determine the effect hydrophobicity has on dispersal and splash dynamics. These beads yielded some differences in dispersal distance, but no notable difference in splash dynamics. Models of the conical fruit bodies of the splash-cups were 3D printed and high-speed video was used to find splash characteristics, and dispersal distance was calculated by measuring the distance from the model to the final resting position of the seeds and droplets.

2:22PM R20.00005 Dynamic self-organization of confined autophoretic particles¹,

ANTHONY MEDRANO, University of Southern California, SBASTIEN MICHELIN, Ecole Polytechnique, EVA KANSO, University of Southern California — We study the behavior of chemically-active Janus particles in microfluidic Hele-Shaw-type confinement. These micron-scale chemical motors, when immersed in a fuel-laden fluid, produce an ionic chemical field which leads to motility and consequently a local fluid flow. In unconfined settings, experimental and computational studies have shown these particles to spontaneously self-organize into crystal structures, and form into asters of two or more particles. Here, we show that geometric confinement alters both the chemical and hydrodynamic signature of the particles in such a way that their far-field effects can be modeled as source dipoles. Each particle moves according to its own self-propelled motion and in response to the chemical and hydrodynamic field created by other particles. Two interaction modes are observed: self-assembly into quasi-static crystals and into dynamically-evolving chains. We discuss the conditions that lead to these modes of interactions and the phase transitions between them for various Janus particle concentrations.

¹The National GEM Consortium

2:35PM R20.00006 Feedback between intracellular flow, signaling and active stresses in Physarum plasmodial fragments, SHUN ZHANG, University of California at San Diego, ROBERT GUY, University of

California at Davis, JUAN CARLOS DEL ALAMO, University of California at San Diego — Physarum polycephalum is a multinucleated slime mold whose endoplasm flows periodically driven by the contraction of its ectoplasm, a dense shell of F-actin cross-linked by myosin molecular motors and attached to the cell membrane. Ectoplasm contractions are regulated by calcium ions whose propagation is in turn governed by the flow. We study experimentally how this feedback leads to auto-oscillation by simultaneously measuring endoplasmic flow speed and rheological properties, the traction stresses between the ectoplasm and its substratum and the distribution of endoplasmic free calcium ions. We find that physarum fragments smaller than 100 microns remain round and stay in place. However, larger fragments break symmetry leading to sustained forward locomotion, in process that is reminiscent of an interfacial instability that seems to settle around two different limit cycles (traveling waves and standing waves). By using different adhesive coatings in the substratum we investigate the role of substratum friction in the emergence of coherent endoplasmic flow patterns and overall physarum fragment locomotion.

2:48PM R20.00007 ABSTRACT WITHDRAWN —**Tuesday, November 22, 2016 1:30PM - 3:40PM —**

Session R21 Bio: Microswimmers D139-140 - Eva Kanso, University of Southern California

1:30PM R21.00001 Precession in Stokes flow: spin and revolution of a bacterial flagellum¹, TAKUJI ISHIKAWA, YOICHIRO SAWANO, HIROMICHI WAKEBE, YUICHI INOUE, Tohoku University, AKIHIKO ISHIIJIMA, Osaka University, YUJI SHIMOGONYA, Tohoku University — The bacterial flagellar motor is an ion-driven rotary machine in the cell envelope of bacteria. When we performed a bead assay, in which the cell body was affixed to a glass surface to observe the rotation of a truncated flagellum via the positioning of a 250 nm-diameter gold nanoparticle, we often observed that the filament motion consisted of two types of rotation: spin and revolution, which resulted in precession. Since the mechanism of flagella precession was unknown, we investigated it using numerical simulations. The results show that the precession occurred due to hydrodynamic interactions between the flagellum and the wall in the Stokes flow regime. We also developed a simple theory of the precession, which validity was confirmed by comparing with the simulation. The theory could be utilized to predict both the filament tilt angle and motor torque from experimental flagellar precession data. The knowledge obtained is important in understanding mechanical properties of the bacterial motor and hook.

¹This work was supported in part by a Japan Society Promotion of Science Grants-in-Aid for Scientific Research (JSPS KAKENHI) (Grant Nos. 25000008 and 26242039).

1:43PM R21.00002 Hydrodynamics of freely swimming flagellates¹, JULIA DOLGER, Department of Physics and Centre for Ocean Life, Technical University of Denmark, LASSE TOR NIELSEN, THOMAS KIORBOE, National Institute of Aquatic Resources and Centre for Ocean Life, Technical University of Denmark, TOMAS BOHR, ANDERS ANDERSEN, Department of Physics and Centre for Ocean Life, Technical University of Denmark — Flagellates are a diverse group of unicellular organisms forming an important part of the marine ecosystem. The arrangement of flagella around the cell serves as a key trait optimizing and compromising essential functions. With micro-particle image velocimetry we observed time-resolved near-cell flows around freely swimming flagellates, and we developed an analytical model based on the Stokes flow around a solid sphere propelled by a variable number of differently placed, temporally varying point forces, each representing one flagellum. The model allows us to reproduce the observed flow patterns and swimming dynamics, and to extract quantities such as swimming velocities and prey clearance rates as well as flow disturbances revealing the organism to flow-sensing predators. Our results point to optimal flagellar arrangements and beat patterns, and essential trade-offs. For biflagellates with two symmetrically arranged flagella we contrasted two species using undulatory and ciliary beat patterns, respectively, and found breast-stroke type beat patterns with equatorial power strokes to be favorable for fast as well as quiet swimming.

¹The Centre for Ocean Life is a VCKR Centre of Excellence supported by the Villum Foundation.

1:56PM R21.00003 Emergence of multiple synchronization modes in hydrodynamically-coupled cilia, HANLIANG GUO, EVA KANSO, University of Southern California — Motile cilia and flagella exhibit different phase coordinations. For example, closely swimming spermatozoa are observed to synchronize together; bi-flagellates *Chlamydomonas* regulate the flagella in a “breast-stroke” fashion; cilia on the surface of *Paramecium* beat in a fixed phase lag in an orchestrated wave like fashion. Experimental evidence suggests that phase coordinations can be achieved solely via hydrodynamical interactions. However, the exact mechanisms behind it remain illusive. Here, adapting a “geometric switch” model, we observe different synchronization modes in pairs of hydrodynamically-coupled cilia by changing physical parameters such as the strength of the cilia internal motor and the separation distance between cilia. Interestingly, we find regions in the parameter space where the coupled cilia reach stable phase coordinations and regions where the phase coordinations are sensitive to perturbations. We also find that leaning into the fluid reduces the sensitivity to perturbations, and produces stable phase coordination that is neither in-phase nor anti-phase, which could explain the origin of metachronal waves in large cilia populations.

2:09PM R21.00004 Hydrodynamic interaction between two helical swimmers, ALEJANDRO RUIZ ESPARZA, FRANCISCO GODINEZ, Universidad Nacional Autonoma de Mexico, ERIC LAUGA, University of Cambridge, ROBERTO ZENIT, Universidad Nacional Autonoma de Mexico — Many motile bacteria, such as *E. coli*, possess several helical flagellar filaments that bundle together to form a coherent helical element for propulsion. In order to understand the process of bundling, we study the interaction between two identical helical magnetic swimmers that self propel in a highly viscous Newtonian fluid due to the rotation of an external magnetic field. Our experiments reveal that hydrodynamic interactions lead to nontrivial collective and relative effects, both in translation and rotation. We will present our experimental results and discuss the physical mechanisms responsible for our observations.

2:22PM R21.00005 Stokesian swimming of a helical swimmer across an interface, FRANCISCO GODINEZ, ARMANDO RAMOS, ROBERTO ZENIT, Universidad Nacional Autonoma de Mexico — Microorganisms swim in flows dominated by viscous effects but in many instances the motion occurs across heterogeneous environments where the fluid properties may vary. To our knowledge, the effect of such in-homogeneity has not been addressed in depth. We conduct experiments in which a magnetic self-propelled helical swimmer displaces across the interface between two immiscible density stratified fluids. As the swimmer crosses the interface, at a fixed rotation rate, its speed is reduced and a certain volume of the lower fluid is dragged across. We quantify the drift volume and the change of swimming speed for different swimming speeds and different fluid combinations. We relate the reduction of the swimming speed with the interfacial tension of the interface. We also compare the measurements of the drift volume with some recent calculations found in the literature.

2:35PM R21.00006 Instabilities of a rotating helical rod, YUNYOUNG PARK, Chung-Ang University, WILLIAM KO, University of Cincinnati, YONGSAM KIM, Chung-Ang University, SOOKKYUNG LIM, University of Cincinnati — Bacteria such as *Escherichia coli* and *Vibrio alginolyticus* have helical flagellar filament. By rotating a motor, which is located at the bottom end of the flagellar filament embedded in the cell body, CCW or CW, they swim forward or backward. We model a left-handed helix by the Kirchhoff rod theory and use regularized Stokes formulation to study an interaction between the surrounding fluid and the flagellar filament. We perform numerical studies focusing on relations between physical parameters and critical angular frequency of the motor, which separates overwinding from twirling. We are also interested in the buckling instability of the hook, which is very flexible elastic rod. By measuring buckling angle, which is an angle between rotational axis and helical axis, we observe the effects of physical parameters on buckling of the hook.

2:48PM R21.00007 The swimming speed of a confined rotating helix in creeping flow, VERONICA ANGELES, ROBERTO ZENIT, Universidad Nacional Autonoma de Mexico — Recent theoretical and numerical studies have shown that the swimming speed of a rotating helix confined in a tube or between walls is higher than the unconfined case, for the same helix properties (helix geometry and rotation speed). We conduct experiments using a magnetic self-propelled force-free robot placed in between two walls or inside a cylinder. We vary the degree of confinement and measure the translation speed for different helix geometries and rotation speeds. We do find an increase of the swimming speeds, which is in good agreement with the predictions of a wall-corrected resistive-force theory. However, since the torque also increases as a result of confinement, the experiments are restricted by the available magnetic torque. Therefore, the increase in swimming speed is only observed for low confinement levels.

3:01PM R21.00008 A numerical study of the effects of fluid rheology and stroke kinematics on flagellar swimming in complex fluids, CHUANBIN LI, ROBERT GUY, BECCA THOMASES, UC Davis — It is observed in experiments that as the fluid rheology is changed, *Chlamydomonas reinhardtii* exhibits changes in both flagellar kinematics and the swimming speed. To understand this phenomenon, we develop a computational model of the swimmer, using flagellar strokes fit from experimental data. We conduct numerical simulations by changing strokes and fluid rheology independently to dissect the effects of these two factors. We discover that stroke patterns extracted from viscoelastic fluids generate much lower stress and have higher efficiency at the cost of lower swimming speed. We also discover that higher fluid elasticity hinders swimming for a fixed stroke pattern.

3:14PM R21.00009 Propulsion by Helical Strips in Circular Channels, SERHAT YESILYURT, EBRU DEMIR, Sabanci University, Istanbul — Progress in manufacturing techniques avails the production of artificial micro swimmers (AMS) in various shapes and sizes. There are numerous studies on the generation of efficient locomotion by means of helical tails with circular cross-sections. This work focuses on locomotion with helical strips in circular channels. A CFD model is used to analyze the effects of geometric parameters and the radius of the channel on swimming velocity of infinite helical-strips in circular channels. Results show that there is an optimum wavelength that depends on thickness to channel radius ratio, suggesting that these parameters need to be optimized simultaneously. With constant torque, thinner strips swim faster, whereas under constant angular velocity application, thicker strips (in radial direction) prevail. As width approaches the wavelength, velocity decreases under both conditions, unless a magnetically coated tail is simulated, for which width has an optimum value. Increasing channel radius to helix amplitude ratio increases the velocity up to a maximum and after a slight drop, saturation occurs as bulk swimming conditions are approached.

3:27PM R21.00010 Trajectories of Artificial Microswimmers with Helical Tails Inside Circular Channels, SERHAT YESILYURT, HAKAN CALDAG, Sabanci University — Trajectories are obtained for millimeter-scale artificial microswimmers inside circular channels filled with glycerol. Rotating magnetic field is applied to propel 3D-printed swimmers with helical tails and permanent magnetic heads. Experiments are recorded with a high-speed camera and processed with contrast-based image processing tools to extract 3D trajectories and orientations of the swimmers. Swimmers pushed by the tail exhibit a helical trajectory at all times while straight trajectories are observed when the length to diameter ratio is very high for pulled ones. Long tails are pointed towards the channel's centerline and short ones are pointed towards the wall. Weak Poiseuille flow is found to alter the swimming speed and suppress the step-out behavior. Flow from tail side increases the instability of swimmers. Experimental observations are validated with snapshot and dynamic models that use CFD to obtain average and time-dependent velocities and trajectories of the swimmer. Lastly, modulation of the rotating magnetic field tilts the swimmer in desired directions or halts the swimmer propulsion without stopping the rotation of the swimmer.

Tuesday, November 22, 2016 1:30PM - 3:40PM –
Session R22 Microscale Flows: Electrokinetics E141/142 - Prashanta Dutta, Washington State University

1:30PM R22.00001 Stable Rotation of Microparticles using a Combination of Dielectrophoresis and Electroosmosis¹, PRASHANTA DUTTA, WALID REZANOOR, Washington State University — Electric field induced microparticle rotation has become a powerful technique to evaluate cell membrane dielectric properties and cell morphology. In this study, stable rotations of microparticles are demonstrated in a stationary AC electric field created from a set of coplanar interdigitated microelectrodes. The medium, particle size, and material are carefully chosen so that particle can be controlled by dielectrophoretic force, while a sufficiently high AC electroosmotic flow is produced for continuous particle rotation. Stable rotation up to 218 rpm is observed at 30 V_{p-p} applied sinusoidal potential in the frequency range of 80 – 1000 Hz. The particle spin rate observed from the experimental study is then validated with a numerical model. The model is formulated around complex charge conservation equation to determine the electric potential distribution in the domain. Stokes equation is employed to solve for AC electroosmotic fluid flow in the domain. Complexity arising from nonlinear potential drop across the electric double layer due to the application of a very large electric potential is also addressed by introducing modified capacitance equation which considers steric effect.

¹This work was supported in part by the U.S. National Science Foundation under grant no. DMS 1317671

1:43PM R22.00002 Influence of Electrolyte Concentration on the Aggregation Of Colloidal Particles Near Electrodes in Oscillatory Fields, SCOTT BUKOSKY, SUKHEEEN SAINI, WILLIAM RISTENPART, Dept. Chemical Engineering, University of California Davis — Micron-scale particles suspended in various aqueous electrolytes have been widely observed to aggregate near electrodes in response to oscillatory electric fields, a phenomenon believed to result from electrically induced flows around the particles. Most work has focused on a narrow range of ionic strengths. Here we demonstrate that an applied field causes micron-scale particles in aqueous NaCl to rapidly aggregate over a wide range of ionic strengths, but with significant differences in aggregation morphology. Optical microscopy observations reveal that at higher ionic strengths (~1 mM) particles arrange as hexagonally closed-packed (HCP) crystals, but at lower ionic strengths (~0.05 mM) the particles arrange in randomly closed-packed (RCP) structures. We interpret this behavior in terms of two complementary effects: an increased particle diffusivity at lower ionic strengths due to increased particle height over the electrode and the existence of a deep secondary minimum in the particle pair interaction potential at higher ionic strength that traps particles in close proximity to one another. The results suggest that electrically induced crystallization will readily occur only over a narrow range of ionic strengths.

1:56PM R22.00003 ABSTRACT WITHDRAWN –

2:09PM R22.00004 Mathematical analysis of electromigration dispersion fronts, IVAN C. CHRISTOV, Purdue University — It is of interest to understand traveling electromigration wave phenomena, such as *isotachophoretic boundaries*, because of their applications in electrophoretic separation methods. To this end, we construct exact solutions to an unusual nonlinear advection–diffusion equation arising in the study of Taylor–Aris (also known as shear) dispersion due to electroosmotic flow during electromigration in a capillary. An exact reduction to a Darboux equation is found under a traveling-wave anzats. The equilibria of this ordinary differential equation are analyzed, showing that their stability is determined solely by the (dimensionless) wave speed without regard to any (dimensionless) physical parameters. Integral curves, connecting the appropriate equilibria of the Darboux equation that governs traveling waves, are constructed, which in turn are shown to be asymmetric kink solutions (*i.e.*, non-Taylor shocks). Furthermore, it is shown that the governing Darboux equation exhibits *bistability*, which leads to two coexisting non-negative kink solutions for (dimensionless) wave speeds greater than unity.

2:22PM R22.00005 Reconfigurable microfluidic nanoparticle trapping using dielectrophoresis for chemical detection, REZA SALEMILANI¹, BRIAN PIOREK², MARTIN MOSKOVITS³, CARL MEINHART⁴, Institute for Collaborative Biotechnologies, University of California, Santa Barbara, California 93106, United States — We report a microfluidic particle manipulation platform based on dielectrophoresis (DEP) to capture and release nanoscale particles cyclically via reconfigurable traps. DEP is routinely used in microfluidic devices for capturing and trapping cells and particles of various sizes, however the trapping of small nanoparticles by DEP is challenging due to the inverse relationship of the DEP force with particle size. The architecture we describe uses electrically insulating silica beads of micron scale in conjunction with DEP electrodes configured to manipulate nanoscale particles for microfluidic applications such as filtration and chemical detection.

¹Department of Mechanical Engineering, University of California, Santa Barbara, California 93106, United States

²Department of Mechanical Engineering, University of California, Santa Barbara, California 93106, United States

³Department of Chemistry, University of California Santa Barbara, Santa Barbara, California 93106, United States

⁴Department of Mechanical Engineering, University of California, Santa Barbara, California 93106, United States

2:35PM R22.00006 Fluctuating Hydrodynamics of Electrolytes Solutions¹, JEAN-PHILIPPE PERAUD, ANDY NONAKA, Lawrence Berkeley National Laboratory, ANUJ CHAUDHRI, Jawaharlal Nehru Centre for Advanced Scientific Research, JOHN B. BELL, Lawrence Berkeley National Laboratory, ALEKSANDAR DONEV, Courant Institute of Mathematical Sciences, New York University, ALEJANDRO L. GARCIA, Department of Physics and Astronomy, San Jose State University — In this work, we develop a numerical method for multicomponent solutions featuring electrolytes, in the context of fluctuating hydrodynamics as modeled by the Landau-Lifshitz Navier Stokes equations. Starting from a previously developed numerical scheme for multicomponent low Mach number fluctuating hydrodynamics [1], we study the effect of the additional forcing terms induced by charged species. We validate our numerical approach with additional theoretical considerations and with examples involving sodium-chloride solutions, with length scales close to Debye length. In particular, we show how charged species modify the structure factors of the fluctuations, both in equilibrium and non-equilibrium (giant fluctuations) systems, and show that the former is consistent with Debye-Hückel theory. We also discuss the consistency of this approach with the electroneutral approximation in regimes where characteristic length scales are significantly larger than the Debye length. Finally, we use this method to explore a type of electrokinetic instability. [1] A. Donev & al., "Low Mach Number Fluctuating Hydrodynamics of Multispecies Liquid Mixtures", *Phys. Fluids*, 27, 3, 2015

¹This work was supported by the U.S. Department of Energy, Office of Science, Office of Advanced Scientific Computing Research,

2:48PM R22.00007 Time-dependent electrokinetic flows of non-Newtonian fluids in microchannel-array for energy conversion¹, MYUNG-SUK CHUN², BYOUNGJIN CHUN³, JI-YOUNG LEE, Korea Institute of Science and Technology (KIST), COMPLEX FLUIDS TEAM⁴ — We investigate the externally time-dependent pulsatile electrokinetic viscous flows by extending the previous simulations concerning the electrokinetic microfluidics for different geometries. The external body force originated from between the nonlinear Poisson–Boltzmann field and the flow-induced electric field is employed in the Cauchy momentum equation, and then the Nernst–Planck equation in connection with the net current conservation is coupled. Our explicit model allows one to quantify the effects of the oscillating frequency and conductance of the Stern layer, considering the shear thinning effect and the strong electric double layer interaction. This presentation reports the new results regarding the implication of optimum frequency pressure pulsations toward realizing mechanical to electrical energy transfer with high conversion efficiencies. These combined factors for different channel dimension are examined in depth to obtain possible enhancements of streaming current, with taking advantage of pulsating pressure field. From experimental verifications by using electrokinetic power chip, it is concluded that our theoretical framework can serve as a useful basis for micro/nanofluidics design and potential applications to the enhanced energy conversion.

¹NRF of Korea (No.2015R1A2A1A15052979) and KIST (No.2E26490)

²Prof. Dr.

³Dr.

⁴National Agenda Research Division

3:01PM R22.00008 Deformations of a pre-stretched elastic membrane driven by non-uniform electroosmotic flow¹, MORAN BERCOVICI, EVGENIY BOYKO, AMIR GAT, Technion - Israel Institute of Technology — We study viscous-elastic dynamics of fluid confined between a rigid plate and a pre-stretched elastic membrane subjected to non-uniform electroosmotic flow, and focus on the case of a finite-size membrane clamped at its boundaries. Considering small deformations of a strongly pre-stretched membrane, and applying the lubrication approximation for the flow, we derive a linearized leading-order non-homogenous 4th order diffusion equation governing the deformation and pressure fields. We derive a time-dependent Green's function for a rectangular domain, and use it to obtain several basic solutions for the cases of constant and time varying electric fields. In addition, defining an asymptotic expansion where the small parameter is the ratio of the induced to prescribed tension, we obtain a set of four one-way coupled equations providing a first order correction for the deformation field.

¹Funded by the European Research Council (ERC) under the Horizon 2020 Research and Innovation Programme, grant agreement No. 678734 (MetamorphChip)

3:14PM R22.00009 Flow of Power-Law Liquids in a Hele-Shaw Cell Driven by Non-Uniform Electroosmotic Slip in the Case of Strong Depletion, EVGENIY BOYKO, MORAN BERCOVICI, AMIR GAT, Technion-Israel Institute of Technology — We analyze flow of a non-Newtonian fluid in a Hele-Shaw cell, subjected to spatially non-uniform electroosmotic flow. We specifically focus on power-law fluids with wall depletion properties and derive a p-Poisson equation governing the pressure field, as well as a set of linearized equations representing its asymptotic approximation for weakly non-Newtonian behavior. To investigate the effect of non-Newtonian properties on the resulting fluidic pressure and velocity, we consider several configurations in one and two dimensions, and calculate both exact and approximate solutions. We show that the asymptotic approximation is in good agreement with exact solutions even for fluids with significant non-Newtonian behavior. The asymptotic model thus enables prediction of the flow and pressure fields for non-Newtonian fluids, and may be particularly useful for the analysis and design of microfluidic systems involving electro-kinetic transport of such fluids.

3:27PM R22.00010 Novel propulsion of active colloids by self-induced field gradients with potential for cargo transport¹, ALICIA BOYMELGREEN, GILAD YOSSIFON, Technion, TOUVIA MILOH, Tel Aviv University — Localized electric field gradients, induced by the dual symmetry-breaking of an asymmetric particle adjacent to a wall are shown to potentially drive particle motion, even in a uniform field. Since the driving gradient is induced by the particle itself, we have termed this propulsion mechanism “self-dielectrophoresis” (sDEP), to distinguish from traditional DEP where the driving non-uniform field is externally fixed and particle direction is restricted. It is also shown that sDEP driven particles are natural cargo carriers, since the localized gradients can also trap and release targets selectively and on demand. This phenomenon is specifically characterized for Gold-Polystyrene Janus spheres, including the establishment of a non-dimensional parameter marking the critical frequency at which sDEP dominates low-frequency ICEP—evidenced by a reversal in particle direction. Additionally we demonstrate that localized gradients can transform the translating Janus particles into an externally controlled, mobile floating electrode with the ability to collect, transport and release a target sample a target 1/50 of its size. It is also shown that calculated control of the frequency enables selective sorting and transport – if the driving frequency is aligned with the positive-DEP (pDEP) response of a specific “target” and negative-DEP (nDEP) of any other contaminants, only the former will be transported with the Janus sphere.

¹ISF,BSF,RBNI

Tuesday, November 22, 2016 1:30PM - 3:14PM –
Session R25 Surface Tension: General; Superfluids E145 - Yongkang Chen, Portland State University

1:30PM R25.00001 Revisiting the first fluid interface experiment in space¹, YONGKANG CHEN, MARK WEISLOGEL, Portland State University, WILLIAM MASICA, FRED KOHL, NASA Glenn Research Center retired, ROBERT GREEN, NASA Glenn Research Center — This year marks the 54th anniversary of the first fluid physics experiment performed aboard a spacecraft during the Mercury-Atlas 7 mission (MA7). The MA7 experiment test cell served as an early model for a spacecraft liquid fuel tank consisting of a circular standpipe mounted within a spherical container. The low-g free surface configuration was dependent on contact angle, fluid fill fraction, standpipe dimensions, and initial conditions. Well-behaved symmetric equilibrium interfaces in the symmetric tank were expected and observed during the historic flight. We revisit the problem here employing a modern numerical tool and discover a rich variety of asymmetric fluid interface configurations that were not observed during the experiment. Interestingly, experimental support for these newly-computed outcomes may be found in 54 year old drop tower data collected by the original NASA investigator team. In short, rotationally symmetric nodoidal surfaces are unstable in a certain domain giving rise to highly asymmetric surfaces with significant shifts in the mass center of the liquid. The NASA team selected a fluid fill level for MA7 that ‘fortunately’ fell outside this domain.

¹NASA NNX12A047A

1:43PM R25.00002 Marangoni effects on a thin liquid film coating a sphere with axial or radial thermal gradients, DI KANG, ALI NADIM, MARINA CHUGUNOVA, Claremont Graduate University — We study the time evolution of a thin liquid film coating the outer surface of a sphere in the presence of gravity, surface tension and thermal gradients. We derive the fourth-order nonlinear partial differential equation that models the thin film dynamics, including Marangoni terms arising from the dependence of surface tension on temperature. We consider two different heating regimes with axial or radial thermal gradients. We analyze the stability of a uniform coating under small perturbations and carry out numerical simulations in COMSOL for a range of parameter values. In the case of an axial temperature gradient, we find steady states with either uniform film thickness, or with drops forming at the top or bottom of the sphere, depending on the total volume of liquid in the film, dictating whether gravity or Marangoni effects dominate. In the case of a radial temperature gradient, a stability analysis reveals the most unstable non-axisymmetric modes on an initially uniform coating film.

1:56PM R25.00003 Electrostatic Assist of Liquid Transfer in Printing Processes, CHUNG-HSUAN HUANG, SATISH KUMAR, University of Minnesota — Transfer of liquid from one surface to another plays an important role in many printing processes. Incomplete liquid transfer can produce defects that are detrimental to the operation of printed electronic devices, and one strategy for minimizing these defects is to apply an electric field, a technique known as electrostatic assist (ESA). However, the underlying physical mechanisms of ESA remain a mystery. To better understand these mechanisms, slender-jet models for both perfect dielectric and leaky dielectric Newtonian liquid bridges with moving contact lines are developed. Nonlinear partial differential equations describing the time- and axial-evolution of the bridge radius and interfacial charge are derived, and then solved using finite-element methods. For perfect dielectrics, it is found that application of an electric field enhances transfer of liquid to the more wettable surface. For leaky dielectrics, application of an electric field can augment or oppose the influence of wettability differences, depending on the direction of the electric field and the sign of the interfacial charge. The physical mechanisms underlying these observations will be discussed.

2:09PM R25.00004 Experiments on chemically enhanced immiscible fluid displacements¹, TEJASWI SOORI, THOMAS WARD, Iowa State Univ — This talk focuses on experiments conducted by displacing a vegetable oil within a capillary tube (diameter < 1 mm) using an aqueous alkali solution. Estimates of the residual film were measured as a function of Reynolds (Re), viscous Atwood (At) and capillary (Ca) numbers. The pendant drop method was used to measure surface tension of the aqueous alkali solutions. We observed a decrease in surface tension for an increase in alkali concentration, which beyond a critical concentration forms a stable micro-emulsion. We estimate the shear viscosity of the emulsion as a function of alkali and aqueous/oil concentrations. Separately we attempt to measure the average bulk diffusion coefficient of the emulsion in both phases which is necessary to estimate the Péclet number (P_e) and subsequent mass transport phenomena.

¹American Chemical Society Petroleum Research Fund

2:22PM R25.00005 Grooves drain dew, HENRI LHUISSIER, IUSTI, Aix-Marseille Univ. & CNRS, France, PIERRE-BRICE BINTEIN, LAURENT ROYON, MSC, Univ. Paris-Diderot, France, ANNE MONGRUET, DANIEL BEYSENS, ESPCI, Paris, France — The yield of natural dew harvesting is often limited by the amount of water remaining on the collector plate at sunrise. By cooling inclined and partially wetting plates in a controlled atmosphere, we show that this immobilized amount is significantly reduced when the plate is dug with vertical sub-millimeter-sized grooves. As condensation proceeds, the grooves rapidly fill up with water and hasten drop shedding by two mechanisms. First, they connect and provoke the coalescence of distant drops, which accelerates the emergence of large drops. Second, they reduce the drop pinning to the plate, which decreases the drop size at the onset of shedding. We will discuss how these mechanisms depend on the rate of condensation, the plate inclination and the grooves dimensions, as well as the consequences for dew harvesting.

2:35PM R25.00006 Three Dimensional Particle Tracking in Superfluid Helium¹, PETER

MEGSON, DANIEL LATHROP, ITAMAR SHANI, University of Maryland, College Park — Superfluid helium is a macroscopic quantum state which exhibits exotic physical properties, such as flow without friction and ballistic heat transport. Superfluid flow is irrotational except about line-like topological phase defects with quantized circulation, known as quantized vortices. The presence of these vortices and their dynamics is the dominating factor of turbulence in superfluid flows. One commonly studied regime of superfluid turbulence is thermal counterflow, where a local heat flux drives the formation and growth of a tangle of vortices. This talk will present experimental studies of counterflow turbulence performed using a multi-camera three-dimensional imaging apparatus with micron-sized ice tracer particles as well as fluorescent nanoparticles. In particular, we will discuss the measurement of three-dimensional velocities and their autocorrelations. Additionally, we are developing new techniques for optical studies of bulk superfluid helium, with particular focus on characterizing tracer particles and particle dispersal mechanisms.

¹Funding from NSF DMR-1407472

2:48PM R25.00007 Vortex reconnections and rebounds in trapped atomic BoseEinstein condensates, LUCA GALANTUCCI, School of Mathematics and Statistics, Newcastle Univ., SIMONE SERAFINI, ELENA ISENI, TOM BIENAIME', RUSSELL BISSET, FRANCO DALFOVO, GIACOMO LAMPORESI, GABRIELE FERRARI, INO-CNR BEC Center and Dipartimento di Fisica, Univ. di Trento, CARLO F. BARENGHI, School of Mathematics and Statistics, Newcastle Univ. — Reconstructions and interactions of filamentary coherent structures play a fundamental role in the dynamics of classical and quantum fluids, plasmas and nematic liquid crystals. In quantum fluids vorticity is concentrated into discrete (quantised) vortex lines (unlike ordinary fluids where vorticity is a continuous field), turning vortex reconstructions into isolated events, conceptually easier to study. In order to investigate the impact of non-homogeneous density fields on the dynamics of quantum reconstructions, we perform a numerical study of two-vortex interactions in magnetically trapped elongated Bose-Einstein condensates in the T=0 limit. We observe different vortex interactions regimes depending on the vortex orientations and their relative velocity: unperturbed orbiting, bounce dynamics, single and double reconnection events. The key ingredients driving the dynamics are the anti-parallel preferred alignment of the vortices and the impact of density gradients arising from the inhomogeneity of the trapping potential. The results are confirmed by ongoing experiments in Trento performed employing an innovative non-destructive real-time imaging technique capable of determining the axial dynamics and the orientation of the vortices.**3:01PM R25.00008 The latent heat of vaporization of supercritical fluids**, DANIEL BANUTI,

MURALIKRISHNA RAJU, Stanford University, JEAN-PIERRE HICKEY, University of Waterloo, MATTHIAS IHME, Stanford University — The enthalpy of vaporization is the energy required to overcome intermolecular attractive forces and to expand the fluid volume against the ambient pressure when transforming a liquid into a gas. It diminishes for rising pressure until it vanishes at the critical point. Counterintuitively, we show that a latent heat is in fact also required to heat a supercritical fluid from a liquid to a gaseous state. Unlike its subcritical counterpart, the supercritical pseudoboiling transition is spread over a finite temperature range. Thus, in addition to overcoming intermolecular attractive forces, added energy simultaneously heats the fluid. Then, considering a transition from a liquid to an ideal gas state, we demonstrate that the required enthalpy is invariant to changes in pressure for $0 < p < 3p_{cr}$. This means that the classical pressure-dependent latent heat is merely the equilibrium part of the phase transition. The reduction at higher pressures is compensated by an increase in a nonequilibrium latent heat required to overcome residual intermolecular forces in the real fluid vapor during heating. At supercritical pressures, all of the transition occurs at non-equilibrium; for $p \rightarrow 0$, all of the transition occurs at equilibrium.

Tuesday, November 22, 2016 1:30PM - 2:35PM –**Session R26 Suspensions: Confined Flows** E146 - Amanda Howard, Brown University

1:30PM R26.00001 Development of wall layering in non-homogenous suspension shear flows, AMANDA HOWARD, MARTIN MAXEY, Brown University — Fully developed homogeneous suspensions of particles in a pressure driven flow show a net increase in particle volume fraction in the center of the channel, representing a densely packed core region, as well as particle layering along the channel walls. Using numerical simulations, we examine the early development of the wall layers in a suspension of neutrally buoyant, non-Brownian particles in steady and unsteady low Reynolds number flows. Because of the no-slip boundary condition, the particles have no tangential velocity at the wall, but can roll in the streamwise direction. When the particle roughness is monodispersed the particles form a layer a uniform distance from the channel wall. Using a bidispersed particle roughness breaks up the coherence of the wall layer resulting in a greater exchange of particles between the wall and the core region. We will report on a series of simulations varying the particle roughness and sizes to examine the development of the wall layer and near-wall particle fluxes.

1:43PM R26.00002 ABSTRACT WITHDRAWN –

1:56PM R26.00003 Are hydrodynamic interactions screened in spherically confined micro-compartments?, CHRISTIAN APONTE-RIVERA, ROSEANNA ZIA, Robert Frederick Smith School of Chemical and Biomolecular Engineering — We study diffusion of hydrodynamically interacting particles confined by a spherical cavity via dynamic simulation, as a model for intracellular transport. Previous models of 3D confined transport typically assume that hydrodynamic interactions are screened and thus can be neglected, but such assumptions lead to qualitative errors in predictive models. Recent studies show that crowding does not screen hydrodynamic entrainment of freely diffusing particles in unbound suspensions, and that diffusing near a planar wall can weaken (but does not screen) hydrodynamic entrainment. Biophysical and other confined suspensions are crowded, watery compartments, suggesting a role of both crowding and confinement in hydrodynamic entrainment. In the present work, we utilize our new computational framework to study the effect of 3D micro-confinement on particle entrainment, and whether such entrainment is algebraically screened. We measure the hydrodynamic entrainment of one particle in the flow induced by another, in suspensions of arbitrary concentration. We find that the strength of entrainment varies spatially in the cavity, changes qualitatively with the size of the confined particles relative to the enclosure, but varies only quantitatively with the concentration of particles.

2:09PM R26.00004 Stokesian Dynamics Simulation for Particulate Suspensions Flowing through Y-Shaped Bifurcating Channel, MALLIKARJUNA MALLELA, PANKAJ TIWARI, ANUGRAH SINGH, Indian Inst of Tech-Guwahati — Suspension of particles dispersed in a fluid medium are encountered in various situations. Many methods, both analytical and numerical, have been developed for understanding and predicting the macroscopic equilibrium and transport properties of these multiphase systems. Stokesian dynamics(Brady and Bossis, 1988) is also one such simulation method which has been fairly successful towards this end. Many interesting phenomena that are often observed in the suspension flow, such as shear induced migration, size segregation, wall slip, etc. are not encountered in simple Newtonian homogeneous liquids. The shear-induced migration phenomena exhibited by concentrated suspensions lead to many interesting observations such as inhomogeneous distribution of the particles and blunting of velocity profiles. Moreover, this phenomenon causes the fractionation of fluid and particle when the suspension flows through the specially designed bifurcating geometries and have been attracting wide attention. In the present work, we are studying the distribution of the particles in the 2D Y-shaped geometry. We have also carried out simulations with varying relative flow rate in daughter branches by varying the width of the daughter branches to study the partitioning of the particles in the daughter branches

2:22PM R26.00005 Stokes-flow simulations of a rising particle suspension in a horizontal rotating cylinder, SUDARSHAN KONIDENA, ANKIREDDY KATHA, ANUGRAH SINGH, Indian Inst of Tech-Guwahati — Experiments performed by Makrand et al. [1] with a suspension of rising particles in a horizontal rotating cylinder exhibit a wide spectrum of non-equilibrium patterns. The axial banding patterns also show a tendency to travel along the rotational axis. We performed Stokes-flow simulations to investigate the aforementioned axial banding along with providing a mechanism for the traveling bands. An order parameter is established basing on either the centrifugal velocity or rising velocity of an isolated particle and the average angular velocity of all the particles. Basing on the rotational frequency, the low and high frequency bands could be characterized into either gravitationally dominant or centrifugally dominant phases with respect to the order parameter. This analysis could aid in providing with a unifying mechanism for axial banding in horizontal rotating cylinders.

Tuesday, November 22, 2016 1:30PM - 3:14PM –
Session R27 CFD: Discontinuous Galerkin and Higher Order Schemes E147-148 - Shervin Sammak, University of Pittsburgh

1:30PM R27.00001 A New Discontinuous Galerkin Method for Convection-Diffusion Problems: The Gradient-Recovery DG Method, PHILIP JOHNSON, ERIC JOHNSEN, University of Michigan, Ann Arbor — The Discontinuous Galerkin (DG) numerical method, while well-suited for hyperbolic PDE systems such as the Euler equations, is not naturally competitive for convection-diffusion systems, such as the Navier-Stokes equations. Where the DG weak form of the Euler equations depends only on the field variables for calculation of numerical fluxes, the traditional form of the Navier-Stokes equations requires calculation of the gradients of field variables for flux calculations. It is this latter task for which the standard DG discretization is ill-suited, and several approaches have been proposed to treat the issue. The most popular strategy for handling diffusion is the “mixed” approach, where the solution gradient is constructed from the primal as an auxiliary. We designed a new mixed approach, called Gradient-Recovery DG; it uses the Recovery concept of Van Leer & Nomura with the mixed approach to produce a scheme with excellent stability, high accuracy, and unambiguous implementation when compared to typical mixed approach concepts. In addition to describing the scheme, we will perform analysis with comparison to other DG approaches for diffusion. Gas dynamics examples will be presented to demonstrate the scheme’s capabilities.

1:43PM R27.00002 High-order Hybridized Discontinuous Galerkin methods for Large-Eddy Simulation, PABLO FERNANDEZ, NGOC-CUONG NGUYEN, JAIME PERAIRE, Massachusetts Inst of Tech-MIT — With the increase in computing power, Large-Eddy Simulation emerges as a promising technique to improve both knowledge of complex flow physics and reliability of flow predictions. Most LES works, however, are limited to simple geometries and low Reynolds numbers due to high computational cost. While most existing LES codes are based on 2nd-order finite volume schemes, the efficient and accurate prediction of complex turbulent flows may require a paradigm shift in computational approach. This drives a growing interest in the development of Discontinuous Galerkin (DG) methods for LES. DG methods allow for high-order, conservative implementations on complex geometries, and offer opportunities for improved sub-grid scale modeling. Also, high-order DG methods are better-suited to exploit modern HPC systems. In the spirit of making them more competitive, researchers have recently developed the hybridized DG methods that result in reduced computational cost and memory footprint. In this talk we present an overview of high-order hybridized DG methods for LES. Numerical accuracy, computational efficiency, and SGS modeling issues are discussed. Numerical results up to Re=460k show rapid grid convergence and excellent agreement with experimental data at moderate computational cost.

1:56PM R27.00003 Extending the dynamic slip-wall model to a compressible discontinuous-Galerkin method, CORENTIN CARTON DE WIART, SCOTT MURMAN, NASA Ames Research Center — Standard equilibrium wall models suffer from both a strong dependence upon mesh resolution and the equilibrium turbulence assumption. Non-equilibrium wall models similarly have limitations for complex geometry due to the need for an auxiliary semi-structured mesh solver, and coupling between the LES and wall-model regions. Bose and Moin’s dynamic slip-wall model¹ offers a new modeling paradigm that does not rely upon assumptions about the local flow physics and uses a dynamic procedure so that the results are independent of resolution. Despite this, the model has not gained significant traction and few independent implementations have been tested. The current work implements the dynamic slip-wall model in an entropy-stable Discontinuous-Galerkin spectral-element solver with a dynamic variational multiscale sub-grid model². This involves both extending the model to a compressible formulation and to a different numerical method. The compressible model is outlined and tested on both attached and separated flows of aerodynamic interest.

¹Bose and Moin, Phys. Fluids **26**(1), (2014)

²Murman *et al.*, AIAA 2016-1059

2:09PM R27.00004 DG-FDF solver for large eddy simulation of compressible flows, SHERVIN SAMMAK, University of Pittsburgh, MICHAEL BRAZELL, DIMITRI MAVRIPLIS, University of Wyoming, PEYMAN GIVI, University of Pittsburgh — A new computational scheme is developed for large eddy simulation (LES) of compressible turbulent flows with the filtered density function (FDF) subgrid scale closure. This is a hybrid scheme, combining the discontinuous Galerkin (DG) Eulerian solver with a Lagrangian Monte Carlo FDF simulator. The methodology is shown to be suitable for LES, as a larger portion of the resolved energy is captured as the order of spectral approximation increases. Simulations are conducted of both subsonic and supersonic flows. The consistency and the overall performance of the DG-FDF solver are demonstrated, together with its shock capturing capabilities.

2:22PM R27.00005 A fully-coupled discontinuous Galerkin spectral element method for two-phase flow in petroleum reservoirs¹, ANKUR TANEJA, JONATHAN HIGDON, University of Illinois at Urbana-Champaign — A spectral element method (SEM) is presented to simulate two-phase fluid flow (oil and water phase) in petroleum reservoirs. Petroleum reservoirs are porous media with heterogeneous geologic features, and the flow of two immiscible phases involves sharp, moving interfaces. The governing equations of motion are time-dependent, non-linear PDEs with strong hyperbolic nature. A fully-coupled numerical scheme using discontinuous Galerkin (DG) method with nodal spectral element basis functions for spatial discretization, and an implicit Runge-Kutta type time-stepping is developed to solve the PDEs in a robust, stable manner. Isoparametric mapping is used to generate grids for reservoir and well geometry. We present the performance capabilities of the DG scheme with high-order basis functions to accurately resolve sharp fluid interfaces and a variety of heterogeneous geologic features. High-order convergence of SEM is demonstrated. Numerical results are presented for reservoir flows with various injection-production patterns. Typical reservoir heterogeneities like low-permeable regions, impermeable shale barriers, etc. are included in the numerical tests. Comparisons with commonly used finite volume methods and linear and quadratic finite element methods are presented.

¹ExxonMobil Upstream Research Co.

2:35PM R27.00006 Discontinuous Galerkin method for predicting heat transfer in hypersonic environments¹, ERIC CHING, YU LV, MATTHIAS IHME, Department of Mechanical Engineering, Stanford University — This study is concerned with predicting surface heat transfer in hypersonic flows using high-order discontinuous Galerkin methods. A robust and accurate shock capturing method designed for steady calculations that uses smooth artificial viscosity for shock stabilization is developed. To eliminate parametric dependence, an optimization method is formulated that results in the least amount of artificial viscosity necessary to sufficiently suppress nonlinear instabilities and achieve steady-state convergence. Performance is evaluated in two canonical hypersonic tests, namely a flow over a circular half-cylinder and flow over a double cone. Results show this methodology to be significantly less sensitive than conventional finite-volume techniques to mesh topology and inviscid flux function. The method is benchmarked against state-of-the-art finite-volume solvers to quantify computational cost and accuracy.

¹Financial support from a Stanford Graduate Fellowship and the NASA Early Career Faculty program are gratefully acknowledged

2:48PM R27.00007 Dynamic mesh adaptation for front evolution using discontinuous Galerkin based weighted condition number relaxation¹, PATRICK GREENE, SAM SCHOFIELD, ROBERT NOURGALIEV, Lawrence Livermore National Laboratory — A new mesh smoothing method designed to cluster cells near a dynamically evolving interface is presented. The method is based on weighted condition number mesh relaxation with the weight function being computed from a level set representation of the interface. The weight function is expressed as a Taylor series based discontinuous Galerkin (DG) projection, which makes the computation of the derivatives of the weight function needed during the condition number optimization process a trivial matter. For cases when a level set is not available, a fast method for generating a low-order level set from discrete cell-centered fields, such as a volume fraction or index function, is provided. Results show that the low-order level set works equally well for the weight function as the actual level set. The method retains the excellent smoothing capabilities of condition number relaxation, while providing a method for clustering mesh cells near regions of interest. Dynamic cases for moving interfaces are presented to demonstrate the method's potential usefulness as a mesh relaxer for arbitrary Lagrangian Eulerian (ALE) methods.

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

3:01PM R27.00008 A three dimensional Dirichlet-to-Neumann map for surface waves over topography¹, ANDRE NACHBIN, DAVID ANDRADE, IMPA/Brazil — We consider three dimensional surface water waves in the potential theory regime. The bottom topography can have a quite general profile. In the case of linear waves the Dirichlet-to-Neumann operator is formulated in a matrix decomposition form. Computational simulations illustrate the performance of the method. Two dimensional periodic bottom variations are considered in both the Bragg resonance regime as well as the rapidly varying (homogenized) regime. In the three-dimensional case we use the Luneburg lens-shaped submerged mound, which promotes the focusing of the underlying rays.

¹FAPERJ Cientistas do Nosso Estado grant 102917/2011 and ANP/PRH-32.

Tuesday, November 22, 2016 1:30PM - 3:40PM –
Session R28 Particle-laden Flows: Particle-Turbulence Interaction II F149 - Laura Villafane,
Stanford University

1:30PM R28.00001 Radiative heating of inertial particles in a turbulent square duct flow, ANDREW BANKO, LAURA VILLAFANE, CHRISTOPHER ELKINS, JOHN EATON, Stanford University — The coupled dynamics of small inertial particles, turbulence, and radiative heating is examined experimentally. A vertically downward airflow with Reynolds number of order 10,000 is laden with disperse Nickel particles which are smaller than all flow length scales. The particles have Stokes numbers of order 10 and the thermal time constant is similar to the aerodynamic time constant. This particle-air mixture is exposed to monochromatic near infrared radiation through one wall of the duct. While the gas and walls are nearly transparent to the incident radiation, the particles absorb energy and heat the gas with a spatial distribution dependent on the particle concentrations. The mass loading ratio of particles is varied in order to study the effect of increasing optical depth on the gas temperature rise. A fine wire thermocouple is used to measure the mean gas temperature variation along the full width of the duct, including the near wall region where particle concentrations mildly increase. Total energy absorption is inferred from measurements of transmitted light intensity. Comparisons are made to a 1-D model which assumes homogeneity of all flow quantities, low optical depth, and ignores preferential concentration.

1:43PM R28.00002 Sensitivity of inertial particle response on turbulent duct flows to mass loading ratio and Reynolds number.

¹, LAURA VILLAFANE, ANDREW BANKO, CHRIS ELKINS, JOHN EATON, Stanford University — The momentum coupled dynamics of particles and turbulence are experimentally investigated in a vertical fully developed turbulent square duct flow of air laden with Nickel particles. Significant preferential concentration is present for the Stokes numbers investigated, which vary from 3 to 30 based on the Kolmogorov time scale. Higher order measures of preferential concentration, such as the sizes and shapes of clusters and voids, are analyzed for increasing mass loading ratios. The mass loadings chosen span the one-way and two-way coupled regimes, while the volume loading is kept low. The effect of Stokes number and mass loading is also evaluated for particle velocity statistics and compared to the unladen gas statistics. Planar laser scattering is used to record instantaneous particle images in the center of the duct. Preferential concentration statistics are computed from box counting and Voronoi tessellation algorithms. PIV and PTV techniques are used to calculate particle velocity statistics. The analysis is extended to the near wall region in the logarithmic layer for the case of low mass loading. These results are compared to those from the duct center to assess the effects of strong carrier phase inhomogeneity on the particle distributions.

¹This Material is based upon work supported by the Department of Energy National Nuclear Security Administration under Award Number DE-NA0002373-1

1:56PM R28.00003 Snowflakes as inertial particles in turbulence

, FILIPPO COLETTI, ANDRAS NEMES, TEJA DASARI, JIARONG HONG, MICHELE GUALA, University of Minnesota — We report on the first direct measurements of trajectories and settling velocity of snow particles in the atmospheric surface layer. During a nocturnal snowfall we deploy an imaging system consisting of a searchlight and high speed cameras to illuminate and track thousands of snowflakes over a 7 m by 4 m vertical plane. We simultaneously characterize their shape and size using digital holography, while recording the air turbulence properties via sonic anemometry. We show that, in the meteorological conditions in object, the snowflake motion exhibits hallmark features identified by fundamental studies of particle-laden turbulence in both the Lagrangian and the Eulerian framework. The acceleration distribution displays stretched exponential tails, and by comparing with previous laboratory and computational studies we infer the Stokes number and aerodynamic response time of the snowflakes. The fall speed is found to be much greater than the expected value in still air, indicating that turbulence enhances settling according to the preferential sweeping mechanism. These observations demonstrate the major role of turbulence in determining the snow fall speed, and create the basis for leveraging results from particle-laden turbulence research towards improved snow precipitation models.

2:09PM R28.00004 Marine particle aggregate breakup in turbulent flows

, MATTHEW RAU, NRC Postdoctoral Fellow, Naval Research Lab, STEVEN ACKLESON, GEOFFREY SMITH, Naval Research Lab — The dynamics of marine particle aggregate formation and breakup due to turbulence is studied experimentally. Aggregates of clay particles, initially in a quiescent aggregation tank, are subjected to fully developed turbulent pipe flow at Reynolds numbers of up to 25,000. This flow arrangement simulates the exposure of marine aggregates in coastal waters to a sudden turbulent event. Particle size distributions are measured by in-situ sampling of the small-angle forward volume scattering function and the volume concentration of the suspended particulate matter is quantified through light attenuation measurements. Results are compared to measurements conducted under laminar and turbulent flow conditions. At low shear rates, larger sized particles indicate that aggregation initially governs the particle dynamics. Breakup is observed when large aggregates are exposed to the highest levels of shear in the experiment. Models describing the aggregation and breakup rates of marine particles due to turbulence are evaluated with the population balance equation and results from the simulation and experiment are compared. Additional model development will more accurately describe aggregation dynamics for remote sensing applications in turbulent marine environments.

2:22PM R28.00005 Turbulence Modulation and Particle Segregation in a Turbulent

Channel Flow

, KEE ONN FONG, MOSTAFA TOLOUI, OMID AMILI, JIARONG HONG, FILIPPO COLETTI, Univ of Minnesota - Twin Cities — Particle-laden flows are ubiquitous in biological, environmental, and engineering flows, but our understanding of the mechanism by which particles modulate turbulence is incomplete. Simulations involve a wide range of scales, and shall be corroborated by measurements that reconstruct the motion of both the continuous and dispersed phases. We present experimental observations on the interaction between inertial particles and turbulent flow through a vertical channel in two-way coupled regime. The working fluid is air laden with size-selected glass particles, which we investigate by planar particle image velocimetry and digital inline holography. Unlike most previous experiments, we focus on a regime in which particle segregation and turbulence modulation are both strong. PIV shows that turbulence modulation is especially pronounced near the wall, where particles accumulate by turbophoresis. The segregation, however, is much weaker than what suggested by one-way coupled simulations. Results from digital holography confirm the trends in particle concentration and velocities, and additionally provide information on the three-dimensional clustering. The findings are compared to previous investigations and discussed in the context of modeling strategies.

2:35PM R28.00006 Effects of solid inertial particles on the velocity and temperature statistics of wall bounded turbulent flow

, BAMDAD LESSANI, South Dakota School of Mines and Technology, HADI NAKHAEI, Technical University of Denmark — The effect of solid inertial particles on the velocity and temperature statistics of a non-isothermal turbulent channel flow is studied using direct numerical simulation. A two-way coupled Eulerian-Lagrangian approach is adopted. Three different particle Stokes numbers of $St = 25, 60, 200$, at a constant particle mass loading of $\phi_m = 0.57$, are considered. The variations of different budget terms for the turbulent kinetic energy equation and fluctuating temperature variance equation in the presence of particles are reported. It is shown that the near wall dissipation and viscous transport terms are larger for $St = 25$ particles compared to the ones of higher inertia particles ($St = 60, 200$). The same behavior is observed for the dissipation and viscous transport terms of the fluctuating temperature variance equation. The fluid turbulent heat flux is also reduced by the presence of particles, but as a result of fluid-particle heat exchange, the total heat transfer rate stays always higher for particle-laden flow even for the largest particles considered. The total Nusselt number is split into a turbulence contribution and a particle contribution, and the effects of particles inertia on fluid turbulent heat flux and fluid-particle heat transfer are examined.

2:48PM R28.00007 Momentum transfer and particle stress in polydisperse, particle-laden flow

, DAVID RICHTER, OMAR GARCIA, CHRISTOPHER ASTEPHEN, University of Notre Dame — Direct numerical simulations are performed in combination with two-way coupled Lagrangian point particles to study the effects of polydispersity on particle-induced modifications to momentum transfer in turbulent wall-bounded flow. Turbulent Couette flow is chosen as an idealized testbed for this purpose since total momentum flux is uniform in the wall-normal direction. Monodisperse simulations are first used to characterize momentum flux modification and particle stress as a function of particle Stokes number, and from this understanding bidisperse and continuously polydisperse mixtures of particle Stokes number are simulated. A simple model is then constructed to predict the total particle stress of these particle mixtures. While in the dilute limit particle stresses are nearly linearly additive, the entire mixture cannot simply be modeled by a single monodisperse particle with an effective Stokes number.

3:01PM R28.00008 Cell structures caused by settling particles in turbulent Rayleigh-Bnard convection, CHANGHOON LEE, SANGRO PARK, Yonsei University — Turbulent thermal convection is an important phenomenon frequently found in nature and industrial processes, often with laden particles. In the last several decades, the vast majority of studies have addressed single phase convective flow with focus on the scaling relation of flow parameters associated with heat transfer. Particle-laden Rayleigh-Bnard convection, however, has not been sufficiently studied. In this study, modulation of cell structures by settling particles in turbulent Rayleigh-Bnard convection in a doubly periodic square channel is investigated using direct numerical simulation with a point particle approach. Flow parameters are fixed at Rayleigh number= 10^6 , Prandtl number=0.7, the aspect ratio=6, and Froude number=0.19. We report from the simulations that settling heavy particles modulate irregular large-scale thermal plume structures into organized polygonal cell structures. Different shapes of flow structures are obtained for different particle diameters and mass loadings. We found that polygonal cell structures arise due to asymmetric feedback force exerted by particles onto hot and cold plumes. Increasing the number of particles augments the asymmetry and the polygonal cell structures become smaller, eventually going to the hexagonal structures.

3:14PM R28.00009 The effect of collision, Stokes and Reynolds numbers on turbophoresis¹, MAHDI ESMAILY-MOGHADAM, ALI MANI, Stanford University — Migration of inertial particles toward solid boundaries in turbulent flows is known as turbophoresis. In this study, we investigate the effect of various parameters on turbophoresis through direct numerical simulations of turbulent flow laden with Lagrangian point-particles. We consider a flow of air in a square duct at a bulk Reynolds number of 5,000 to 20,000 dispersed with nickel particles ranging in size from 4 to 16 micron in diameter. We examine the effect of the Stokes and Reynolds numbers on the near-wall particle concentration and its relationship to the turbophoretic velocity. Our results are consistent with the previously published results pertaining to the saturation of the turbophoretic velocity for Stokes numbers larger than 10. Adopting a hard sphere collision model, we examine the role of collisions on the near wall concentration and demonstrate the sensitivity of the results to the restitution coefficient. Our findings show that while reducing the restitution coefficient leads to a higher degree of turbophoresis; collision can decrease the near wall concentration by orders of magnitude for a global particle volume fraction of $O(10^{-5})$.

¹This work was supported by the United States Department of Energy under the Predictive Science Academic Alliance Program 2 (PSAAP2) at Stanford University.

3:27PM R28.00010 Direct numerical simulation of powder electrification in a turbulent channel flow¹, HOLGER GROSSHANS, Universite Catholique de Louvain, MILTIADIS PAPALEXANDRIS, Universit Catholique de Louvain — Particle electrification is often encountered in process industries. Sometimes it has useful applications, such as the control of particle trajectories through an electric field. In other situations it has negative effects. For example, during pneumatic transport it can cause particle deposition or, even worse, spark discharges and subsequent fires and explosions. Despite its frequent occurrence, due to the complexity of the underlying physical mechanisms, there are still many open questions regarding particle electrification and inconsistent theoretical predictions have been reported. The objective of our work is to gain a better understanding and physical insight of this phenomenon. To this end, we performed Direct Numerical Simulations to analyze the turbulent flow of a carrier fluid with immersed particles in a channel. Moreover, the motion of the particles was computed in a Lagrangian framework and dynamic models accounting for the particle-wall and particle-particle charge exchange were implemented. In our talk, we discuss in detail the effect of the fluid turbulence to the build-up of the electrostatic charge of the particles. Furthermore, we elaborate on the influence of the particle Stokes number and gravitational forces to the process of powder charging.

¹Supported by the National Research Fund of Belgium (FNRS) under the GRANMIX Projet de Recherche grant.

Tuesday, November 22, 2016 1:30PM - 3:40PM – Session R29 CFD: Algorithms II F150 - Joseph Oefelein, Sandia National Labs, California

1:30PM R29.00001 Comparison of Quasi-Conservative Pressure-Based and Fully-Conservative Formulations for the Simulation of Transcritical Flows¹, GUILHEM LACAZE, JOSEPH OEFELIN, Sandia Natl Labs — High-pressure flows are known to be challenging to simulate due to thermodynamic non-linearities occurring in the vicinity of the pseudo-boiling line. This study investigates the origin of this issue by analyzing the behavior of thermodynamic processes at elevated pressure and low temperature. We show that under transcritical conditions, non-linearities significantly amplify numerical errors associated with construction of fluxes. These errors affect the local density and energy balances, which in turn creates pressure oscillations. For that reason, solvers based on a conservative system of equations that transport density and total energy are subject to unphysical pressure variations in gradient regions. These perturbations hinder numerical stability and degrade the accuracy of predictions. To circumvent this problem, the governing system can be reformulated to a pressure-based treatment of energy. We present comparisons between the pressure-based and fully conservative formulations using a progressive set of canonical cases, including a cryogenic turbulent mixing layer at rocket engine conditions.

¹Department of Energy, Office of Science, Basic Energy Sciences Program

1:43PM R29.00002 A GPU-based High-order Multi-resolution Framework for Compressible Flows at All Mach Numbers, CHRISTOPHER J. FORSTER, Georgia Institute of Technology, Sandia National Laboratories, MARC K. SMITH, Georgia Institute of Technology — The Wavelet Adaptive Multiresolution Representation (WAMR) method is a general and robust technique for providing grid adaptivity around the evolution of features in the solutions of partial differential equations and is capable of resolving length scales spanning 6 orders of magnitude. A new flow solver based on the WAMR method and specifically parallelized for the GPU computing architecture has been developed. The compressible formulation of the Navier-Stokes equations is solved using a preconditioned dual-time stepping method that provides accurate solutions for flows at all Mach numbers. The dual-time stepping method allows for control over the residuals of the governing equations and is used to complement the spatial error control provided by the WAMR method. An analytical inverse preconditioning matrix has been derived for an arbitrary number of species that allows preconditioning to be efficiently implemented on the GPU architecture. Additional modifications required for the combination of wavelet-adaptive grids and preconditioned dual-time stepping on the GPU architecture will be discussed. Verification using the Taylor-Green vortex to demonstrate the accuracy of the method will be presented.

1:56PM R29.00003 High order accurate finite difference schemes based on symmetry preservation, ERSIN OZBENLI, PRAKASH VEDULA, U. Oklahoma, Norman — A new algorithm for development of high order accurate finite difference schemes for numerical solution of partial differential equations using Lie symmetries is presented. Considering applicable symmetry groups (such as those relevant to space/time translations, Galilean transformation, scaling, rotation and projection) of a partial differential equation, invariant numerical schemes are constructed based on the notions of moving frames and modified equations. Several strategies for construction of invariant numerical schemes with a desired order of accuracy are analyzed. Performance of the proposed algorithm is demonstrated using analysis of one-dimensional partial differential equations, such as linear advection diffusion equations inviscid Burgers equation and viscous Burgers equation, as our test cases. Through numerical simulations based on these examples, the expected improvement in accuracy of invariant numerical schemes (up to fourth order) is demonstrated. Advantages due to implementation and enhanced computational efficiency inherent in our proposed algorithm are presented. Extension of the basic framework to multidimensional partial differential equations is also discussed.

2:09PM R29.00004 The Space-Time CESE Method Applied to Viscous Flow Computations with High-Aspect Ratio Triangular or Tetrahedral Meshes, CHAU-LYAN CHANG, NASA Langley Research Center, BALAJI VENKATACHARI, National Institute of Aerospace — Flow physics near the viscous wall is intrinsically anisotropic in nature, namely, the gradient along the wall normal direction is much larger than that along the other two orthogonal directions parallel to the surface. Accordingly, high aspect ratio meshes are employed near the viscous wall to capture the physics and maintain low grid count. While such arrangement works fine for structured-grid based methods with dimensional splitting that handles derivatives in each direction separately, similar treatments often lead to numerical instability for unstructured-mesh based methods when triangular or tetrahedral meshes are used. The non-splitting treatment of near-wall gradients for high-aspect ratio triangular or tetrahedral elements results in an ill-conditioned linear system of equations that is closely related to the numerical instability. Altering the side lengths of the near wall tetrahedrons in the gradient calculations would make the system less unstable but more dissipative. This research presents recent progress in applying numerical dissipation control in the space-time conservation element solution element (CESE) method to reduce or alleviate the above-mentioned instability while maintaining reasonable solution accuracy.

2:22PM R29.00005 Framework for a Robust General Purpose Navier-Stokes Solver on Unstructured Meshes¹, CHENG-NIAN XIAO, FABIAN DENNER, BEREND G. M. VAN WACHEM, Imperial College — A numerical framework for a pressure-based all-speeds flow solver operating on unstructured meshes, which is robust for a broad range of flow configurations, is proposed. The distinct features of our framework are the full coupling of the momentum and continuity equations as well as the use of an energy equation in conservation form to relate the thermal quantities with the flow field. In order to overcome the well-documented instability occurring while coupling the thermal energy to the remaining flow variables, a multistage iteration cycle has been devised which exhibits excellent convergence behavior without requiring any numerical relaxation parameters. Different spatial schemes for accurate shock resolution as well as complex thermodynamic gas models are also seamlessly incorporated into the framework. The solver is directly applicable to stationary and transient flows in all Mach number regimes (sub-, trans-, supersonic), exhibits strong robustness and accurately predicts flow and thermal variables at all speeds across shocks of different strengths. We present a wide range of results for both steady and transient compressible flows with vastly different Mach numbers and thermodynamic conditions in complex geometries represented by different types of unstructured meshes.

¹The authors are grateful for the financial support provided by Shell

2:35PM R29.00006 Compressible turbulence and shock-capturing using a variational multiscale method, ANIRBAN GARAI, NICHOLAS BURGESS, SCOTT MURMAN, LASLO DIOSADY, NASA Ames Research Center — We have previously developed a dynamic extension of Hughes' variational multiscale method which is implemented in an entropy-stable Discontinuous-Galerkin spectral-element solver¹. This solver and sub-grid model have been examined on standard low-speed benchmark flows, e.g. homogeneous turbulence, channel flow, etc. Here we extend the approach to higher speeds where compressibility effects are no longer insignificant, and the flowfields develop unsteady shocklets and shock waves. Homogeneous isotropic turbulence at high turbulent Mach number is tested for two cases - decaying and passing through a normal shock. Numerical simulations using the multiscale sub-grid model, no sub-grid model, and a variation of Barter and Darmofal's shock-capturing scheme² are examined in isolation and combination. The computed results are compared against theoretical observations and previous computational results.

¹Murman *et al.*, AIAA 2016-1059

²Barter and Darmofal, J. Comp. Physics, 229(5)

2:48PM R29.00007 ABSTRACT WITHDRAWN —

3:01PM R29.00008 Cost and accuracy comparison between the diffuse interface method and the geometric volume of fluid method for simulating two-phase flows¹, SHAHAB MIRJALILI, Stanford University, CHRISTOPHER BLAKE IVEY, Lawrence Livermore National Laboratory, ALI MANI, Stanford University — The diffuse interface(DI) and volume of fluid(VOF) methods are mass conserving front capturing schemes which can handle large interfacial topology changes in realistic two phase flows. The DI method is a conservative phase field method that tracks an interface with finite thickness spread over a few cells and does not require reinitialization. In addition to having the desirable properties of level set methods for naturally capturing curvature and surface tension forces, the model conserves mass continuously and discretely. The VOF method, which tracks the fractional tagged volume in a cell, is discretely conservative by requiring costly geometric reconstructions of the interface and the fluxes. Both methods however, suffer from inaccuracies in calculation of curvature and surface tension forces. We present a quantitative comparison of these methods in terms of their accuracy, convergence rate, memory, and computational cost using canonical 2D two-phase test cases: damped surface wave, oscillating drop, equilibrium static drop, and dense moving drop. We further compared the models in their ability to handle thin films by looking at the impact of a water drop onto a deep water pool. Considering these results, we suggest qualitative guidelines for using the DI and VOF methods.

¹Supported by ONR

3:14PM R29.00009 Impact Detection for Characterization of Complex Multiphase Flows¹

WAI HONG RONALD CHAN, JAVIER URZAY, ALI MANI, PARVIZ MOIN, Stanford University — Multiphase flows often involve a wide range of impact events, such as liquid droplets impinging on a liquid pool or gas bubbles coalescing in a liquid medium. These events contribute to a myriad of large-scale phenomena, including breaking waves on ocean surfaces. As impacts between surfaces necessarily occur at isolated points, numerical simulations of impact events will require the resolution of molecular scales near the impact points for accurate modeling. This can be prohibitively expensive unless subgrid impact and breakup models are formulated to capture the effects of the interactions. The first step in a large-eddy simulation (LES) based computational methodology for complex multiphase flows like air-sea interactions requires effective detection of these impact events. The starting point of this work is a collision detection algorithm for structured grids on a coupled level set / volume of fluid (CLSVOF) solver (Mortazavi *et al.*, JFM 2016) adapted from an earlier algorithm for cloth animations (Bridson *et al.*, 2002) that triangulates the interface with the marching cubes method (Lorensen and Cline, Comp. Graphics 1987). We explore the extension of collision detection to a geometric VOF solver and to unstructured grids (Ivey and Moin, JCP 2015). Supported by ONR/A*STAR.

¹Agency of Science, Technology and Research, Singapore; Office of Naval Research, USA

3:27PM R29.00010 An implicit control-volume finite element method for well-reservoir modelling¹

DIMITRIOS PAVLIDIS, PABLO SALINAS, ZHIHUA XIE, CHRISTOPHER PAIN, OMAR MATAR, Imperial College London — Here a novel implicit approach (embodied within the IC-Ferst) is presented for modelling wells with potentially a large number of laterals within reservoirs. IC-Ferst is a conservative and consistent, control-volume finite element method (CV-FEM) model and fully unstructured/geology conforming meshes with anisotropic mesh adaptivity. As far as the wells are concerned, a multi-phase/multi-well approach, where well systems are represented as phases, is taken here. Phase volume fraction conservation equations are solved for in both the reservoir and the wells, in addition, the field within wells is also solved for. A second novel aspect of the work is the combination of modelling and resolving of the motherbore and laterals. In this case wells do not have to be explicitly discretised in space. This combination proves to be accurate (in many situations) as well as computationally efficient. The method is applied to a number of multi-phase reservoir problems in order to gain an insight into the effectiveness, in terms of production rate, of perforated laterals. Model results are compared with semi-analytical solutions for simple cases and industry-standard codes for more complicated cases.

¹EPSRC UK Programme Grant MEMPHIS (EP/K003976/1)

Tuesday, November 22, 2016 1:30PM - 3:01PM – Session R31 Experimental Techniques - Scalar Measurements F152 - Hirotaka Sakaue, University of Notre Dame

1:30PM R31.00001 Temperature Distribution Measurement of The Wing Surface under Icing Conditions

HIROSHI ISOKAWA, TAKESHI MIYAZAKI, Univ of Electro-Comm (UEC), SHIGEO KIMURA, Kanagawa Institute of Technology, HIROTAKA SAKAUE, Univ of Notre Dame, KATSUAKI MORITA, Japan Aerospace Exploration Agency, JAPAN AEROSPACE EXPLORATION AGENCY COLLABORATION, UNIV OF NOTRE DAME COLLABORATION, KANAGAWA INSTITUTE OF TECHNOLOGY COLLABORATION, UNIV OF ELECTRO-COMM (UEC) TEAM — De- or anti-icing system of an aircraft is necessary for a safe flight operation. Icing is a phenomenon which is caused by a collision of supercooled water frozen to an object. For the in-flight icing, it may cause a change in the wing cross section that causes stall, and in the worst case, the aircraft would fall. Therefore it is important to know the surface temperature of the wing for de- or anti-icing system. In aerospace field, temperature-sensitive paint (TSP) has been widely used for obtaining the surface temperature distribution on a testing article. The luminescent image from the TSP can be related to the temperature distribution. (TSP measurement system) In icing wind tunnel, we measured the surface temperature distribution of the wing model using the TSP measurement system. The effect of icing conditions on the TSP measurement system is discussed.

1:43PM R31.00002 High Speed Pressure Sensitive Paint for Dynamic Testing¹

CAROLINA PENA, Harvard University, KYLE CHISM, PAUL HUBNER, The University Of Alabama — Pressure sensitive paint (PSP) allows engineers to obtain accurate, high-spatial-resolution measurements of pressure fields over a structure. The pressure is directly related to the luminescence emitted by the paint due to oxygen quenching. Fast PSP has a higher surface area due to its porosity compared to conventional PSP, which enables faster diffusion and measurements to be acquired three orders of magnitude faster than with conventional PSP. A fast time response is needed when testing vibrating structures due to fluid-structure interaction. The goal of this summer project was to set-up, test and analyze the pressure field of an impinging air jet on a vibrating cantilever beam using Fast PSP. Software routines were developed for the processing of the emission images, videos of a static beam coated with Fast PSP were acquired with the air jet on and off, and the intensities of these two cases were ratioed and calibrated to pressure. Going forward, unsteady pressures on a vibrating beam will be measured and presented. Eventually, the long-term goal is to integrate luminescent pressure and strain measurement techniques, simultaneously using Fast PSP and a luminescent photoelastic coating on vibrating structures.

¹ Funding from NSF REU site grant EEC 1358991 is greatly appreciated.

1:56PM R31.00003 Simultaneous measurement of temperature and velocity of air flow over 1000C using two color phosphor thermometry

MASATOSHI FUKUTA, SATOSHI SOMEYA, TETSUO MUNAKATA, AIST, LCS TEAM — Thermal barrier coatings were applied to the gas turbines and the internal combustion engines for the high thermal efficiency. The evaluation and the improvement of coatings require to measure transient gaseous flow near the wall with coatings. An aim of this study is to combine a two color phosphor thermometry with the PIV to measure simultaneously temperature and velocity of the gas over 1000C. The temperature and velocity distribution of an impinging jet of high temperature air was simultaneously visualized in experiments. The temperature was estimated from an intensity ratio of luminescent in different ranges of wavelength, 500 ~600 nm and 400 ~480 nm. Uncertainty of measured temperature was less than 10C. Temperatures measured by the developed method and by thermocouples were agreed well. The measured velocity by the PIV with phosphor particles were also agreed well with the velocity measured by a Laser Doppler Velocimeter.

2:09PM R31.00004 Development of molecular based optical techniques for thermometry and velocimetry for fluorocarbon media¹, SHAHRAM POUYA, GARY BLANCHARD, MANOOCHEHRI KOOCHESFAHANI, Michigan State University — Fluorocarbon solvents are very stable inert fluids with unique physical properties that make them attractive compounds as refrigerant and several medical applications such as contrast enhanced ultrasound imaging. Since they do not mix with typical organic solvents or water, most luminescent (fluorescent or phosphorescent) probes cannot be used as tracers for optical diagnostic techniques. Perfluoropentane, a compound from this family, is used as a simulant fluid by NASA for two-phase heat transfer/mixing experiments under micro-gravity condition due to its low boiling temperature. Here we study the feasibility of employing non-intrusive optical methods for measurements of temperature and/or velocity within Perfluoropentane as the working fluid. Preliminary results of temperature and velocity measurement using Laser Induced Fluorescence and Molecular Tagging Velocimetry are presented.

¹This work was supported by NASA Grant number NNX16AD52A.

2:22PM R31.00005 ABSTRACT WITHDRAWN —

2:35PM R31.00006 Experimental Testing of the T-NSTAP in Supersonic Flow¹, KATHERINE KOKMANIAN, MARCUS HULTMARK, Princeton University — A fast response nanoscale temperature sensor (T-NSTAP) was developed at Princeton University. This novel sensor has been shown to increase both the spatial and the temporal resolutions compared to conventional cold-wire probes, due to its large aspect ratio yet small overall size ($100\text{ nm} \times 2\text{ }\mu\text{m} \times 200\text{ }\mu\text{m}$). The T-NSTAP has been tested in various subsonic facilities, however it has not yet been tested under supersonic conditions. Here we will present the first measurements from supersonic flows using the T-NSTAP in Princeton's Low Turbulence Variable Geometry Facility at Mach 3 and later in Princeton's Hypersonic Boundary Layer Facility (HBLF) at Mach 8 in order to enable unfiltered data of the temperature field in high speed flows. Since the HBLF can generate more challenging conditions than these probes have previously been tested in, our attention will be focused on ensuring that the T-NSTAP can withstand these conditions. Assuming that a shock will form at the front edge of the sensor, the total force on the T-NSTAP was calculated to be on the order of μN , which is less than when it was tested in subsonic pressurized conditions. Investigations will be undertaken to ensure that the structural and electrical properties of the sensors are maintained during the tests.

¹Air Force Office of Scientific Research (AFOSR)

2:48PM R31.00007 Dual Luminescence Imaging for Two Phase Flow, HIROTAKA SAKAUE, Univ of Notre Dame, KATSUAKI MORITA, JAXA — Dual luminescence imaging gives time-resolved information of fluid dynamic phenomena. It uses two luminescent probes; one is sensitive to the detecting medium and the other as a reference. It is a non-intrusive technique, and both luminescent outputs are captured by a high-speed color camera as color-filtered images. By taking a ratio of the two images at the same time frame, this imaging technique can give time-resolved information. It is suitable for a moving and free surface objects. It is also suitable for a measurement where a target is small to mount a conventional thermocouple and pressure probes. Some of the applications of this imaging are described in the presentation, such as icing and boiling phenomena.

Tuesday, November 22, 2016 1:30PM - 2:48PM —
Session R32 Excitation of Turbulent Boundary Layers Oregon Ballroom 201 - Philippe Lavoie,
University of Toronto

1:30PM R32.00001 Measurements of turbulent flow overlying impermeable and permeable walls¹, TAEHOON KIM, University of Illinois, Urbana Champaign, GIANLUCA BLOIS, University of Notre Dame, JAMES BEST, University of Illinois, Urbana Champaign, KENNETH CHRISTENSEN, University of Notre Dame — There exist an array of natural and industrial flow systems wherein the flow is bounded by a surface that is both permeable and rough (e.g. river beds, bed reactors). In such scenarios, the wall boundary condition is complex as it involves both slip and penetration which together significantly modify the statistical and structural modifications the overlying flow owing to momentum exchange across the wall. The current investigation explores the individual roles of topography and permeability in such flows by systematically decoupling one from the other with a number of wall models having the same porous structure (i.e. cubically arranged spheres; two and five layers, respectively, to highlight the effect of turbulence penetration depth) but with different surface topography (smooth versus cubically arranged hemispheres). High resolution particle-image velocimetry measurements were conducted in the streamwise-wall-normal ($x - y$) plane and refractive-index matching was employed to optically access the flow within the permeable wall. First- and second-order velocity statistics are used to assess the flow modifications associated with the different wall models and thus ascertain the individual impacts of permeability and topography.

¹NSF

1:43PM R32.00002 Flows induced by power-law stretching surface motion modulated by arbitrary transverse surface shear, PATRICK WEIDMAN, University of Colorado — Boundary-layer solutions for the flow induced by power-law stretching of a plate are obtained for two generalizations that include arbitrary transverse plate shearing. In one extension the power-law motion is a product of the arbitrary transverse shearing motion. In the other extension the streamwise coordinate is added to the transverse shearing motion and together are raised to the power of stretching. In both cases the original boundary-value problem of Banks is obtained, irrespective of the arbitrary transverse shearing motion.

1:56PM R32.00003 Experimental investigation of compliant wall surface deformation in a turbulent channel flow.¹, CAO ZHANG, JIN WANG, JOSEPH KATZ, Johns Hopkins University — The dynamic response of a compliant wall under a turbulent channel flow is investigated by simultaneously measuring the time-resolved, 3D flow field (using tomographic PIV) and the 2D surface deformation (using interferometry). The pressure distributions are calculated by spatially integrating the material acceleration field. The Reynolds number is $Re_\tau = 2300$, and the centerline velocity (U_0) is 15% of the material shear speed. The wavenumber-frequency spectra of the wall deformation contain a non-adverted low-frequency component and advected modes, some traveling downstream at U_0 and others at $0.72U_0$. Trends in the wall dynamics are elucidated by correlating the deformation with flow variables. The spatial pressure-deformation correlations peak at $y/h \approx 0.12$ (h is half channel height), the elevation of Reynolds shear stress maximum in the log-layer. Streamwise lagging of the deformation behind the pressure is caused in part by phase-lag of the pressure with decreasing distance from the wall, and in part by material damping. Positive deformations (bumps) are preferentially associated with ejections, which involve spanwise vortices located downstream and quasi-streamwise vortices with spanwise offset, consistent with hairpin-like structures. The negative deformations (dents) are preferentially associated with pressure maxima at the transition between an upstream sweep to a downstream ejection.

¹Sponsored by ONR.

2:09PM R32.00004 Temporal and Spatial Response of a Turbulent Boundary Layer to Forcing by Synthetic Jets¹, RONALD HANSON, University of Toronto, BHARATHRAM GANAPATHISUBRAMANI, University of Southampton, PHILIPPE LAVOIE, University of Toronto, TIM BERK, University of Southampton — In this experimental study we examine the spatial and temporal response of a turbulent boundary layer affected by a single, and pair of, synthetic jet actuator(s). The spatial signature of the boundary layer mean-flow has been previously shown to result from large vortical motions caused by the interaction between the synthetic jets and the cross flow. By means of hot-wire measurements, phase-locked to the synthetic jet input, the propagation of the unsteady disturbance can be quantified over the actuation cycle of a synthetic jet. Using long samples both the phase-locked variation of the periodic (actuation cycle) and turbulent fluctuations are examined. It is shown that both the mean flow and turbulence characteristics are markedly different across the span, owing to the three dimensionality of the upstream input. Further, the disturbance shape and phase of the phase-locked disturbance varies significantly with forcing level, in part owing to the disruption of the mean velocity. Particular focus is given to the interaction occurring between the near-wall and outer region scales, which vary across the span, with respect to various forcing conditions.

¹The financial support of Airbus is gratefully acknowledged.

2:22PM R32.00005 Turbulent boundary layer over flexible plates., PARAND ROSTAMI, TINDARO IOPPOLO, Southern Methodist University — This research describes the structure of a turbulent boundary layer flow with a zero pressure gradient over elastic plates. The elastic plates made of a thin aluminum sheets with thickness between 50 and 500 microns were placed on the floor of a subsonic wind tunnel and exposed to a turbulent boundary layer flow with a free stream velocity between 20m/s and 100m/s. The ceiling of the test section of the wind tunnel is adjustable so that a nearly zero pressure gradient is obtained in the test section. Hot-wire anemometry was used to measure the velocity components. Mean, fluctuating velocities and Reynolds stresses will be presented and compared with the values of a rigid plate.

2:35PM R32.00006 The effect of excitation on the plane wall jet¹, SHIBANI BHATT, SRAVAN ARTHAM, EBENEZER GNANAMANICKAM, Embry-Riddle Aeronautical University — The plane wall jet (PWJ) is a unique boundary layer flow in which the highly energetic large-scales of the outer free shear layer transition to turbulence through an inviscid process while, the wall-bounded layer becomes turbulent through a viscous mechanism. These large-scale structures of the PWJ amplitude and frequency modulate the finer scales of the flow much like in canonical boundary layers. However, the unique configuration of the PWJ allows for the independent excitation of the large-scales in the flow to study this interaction with the finer scales. An experimental study is carried out in a PWJ facility operating at friction Reynolds numbers $Re_\tau > 1000$. The PWJ is excited over three decades of Strouhal number. The changes to the turbulent statistics due to the excitation, across the boundary layer, are presented. It was seen that the excitation alters the energy spectra across the entire boundary layer. Certain scales were excited and others augmented and this modification was a function of the excitation frequency. In general, the energy of the large-scales were more significantly altered when compared to the finer scales. Certain excitation frequencies appear to more dramatically alter the energy of the large-scales with changes also to the wall shear stress.

¹Supported by the Air Force Office of Scientific Research (FA9550-16-1-0194)

Tuesday, November 22, 2016 1:30PM - 3:27PM –
Session R33 Turbulence: Strongly Anisotropic Flows Oregon Ballroom 202 - Fabien Godeferd,
Ecole Centrale de Lyon

1:30PM R33.00001 ABSTRACT WITHDRAWN –

1:43PM R33.00002 Vorticity alignment and spectral statistics in a variable-density turbulent flow¹, 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 — Turbulent flows with high density gradients subject to an externally imposed acceleration field, such as gravity, occur in many phenomena, ranging from geophysics to astrophysics. This study investigates turbulence in fluids over a range of density ratios, from small ($R=1.005$) to large ($R=10$). The investigation relies on direct numerical simulation using the incompressible variable-density Navier-Stokes equations, in a triply periodic domain. The flow is initialized with density gradients perpendicular to the acceleration field. This configuration induces baroclinic torques with shear and buoyancy contributing to the evolution of turbulence and turbulent mixing. Of interest in fluid modeling is vorticity alignment, which is presented for the broad density ratio range studied. Prominent variable-density contributions to the vorticity field such as baroclinic torques are discussed. Kinetic-energy spectra are compared to specific kinetic energy spectra to illustrate aspects of variable-density effects.

¹This material is based upon work supported by the DOE, AFOSR, NSF GRFP, and Caltech.

1:56PM R33.00003 Scale by scale energy flux in rotating homogeneous turbulence

, FABIEN GODEFERD, DONATO VALLEFUOCO, AURORE NASO, Fluid Mechanics and Acoustics Laboratory, CNRS, University of Lyon — Homogeneous rotating turbulence is strongly anisotropic and exhibits vortices elongated along the rotation axis, and reduced downscale energy cascade w.r.t. isotropic turbulence. We characterize its dynamics by the Karman-Howarth-Monin equation for 2nd-order 2-point velocity correlations $R(\mathbf{r}, t)$, where \mathbf{r} is the vector separation: $\partial_t R/2 = \nabla \cdot \mathbf{F}/4 + \nu \nabla^2 R + \phi_{inj}$ showing the balance between energy flux \mathbf{F} (3rd-order moment of velocity increment $\mathbf{u}(\mathbf{x} + \mathbf{r}) - \mathbf{u}(\mathbf{x})$), dissipation, and injected energy. From Direct Numerical Simulations of forced rotating turbulence, we get estimates of all terms in the KHM eq. at each scale \mathbf{r} . A map of \mathbf{F} components in the axisymmetric frame is obtained, and compared with experimental data (Lamrabet et al. 2011) for two components measured in an azimuthal plane. We evaluate the role of the unmeasured azimuthal component of energy flux, at different Rossby numbers. We also explain why experiments and inertial wave turbulence theory by Galtier (2013) predict opposing trends in the dependence of the radial energy flux with the direction of \mathbf{r} , by identifying two separate regimes in different scale ranges.

2:09PM R33.00004 The structure of MHD turbulence under an external magnetic field: results from simulations on elongated domains¹

, X.M. ZHAI, P.K. YEUNG, Georgia Tech — Turbulence in an electrically conducting fluid in the limit of low magnetic Reynolds number is, because of the Lorentz force due to an external magnetic field, very different from classical turbulence at both the large scales and the small scales. The importance of minimizing finite domain-size effects on the large scale development has often tended to limit the Reynolds number reached in the past. In this work we use periodic domains stretched along the magnetic field with aspect ratio up to 8 and beyond. The initial state is obtained from decaying isotropic turbulence with large-eddy length scales of order 1% of the length of the domain. After a transient period the kinetic energy returns to a power law decay while the integral length scales in the direction parallel to the magnetic field show preferential growth. At early times the parallel velocity component becomes stronger than the other two but this anisotropy is subsequently reversed under the combined effects of anisotropic Joule dissipation and viscous dissipation. The small scales show characteristics of quasi two-dimensional behavior in the transverse plane. Results over a range of magnetic interaction parameters and Reynolds numbers are compared with known theoretical predictions.

¹Supported by NSF Grant CBET-1510749 and supercomputer resources at TACC/XSEDE and ALCF

2:22PM R33.00005 Anisotropic grid adaptation in LES

, SIAVASH TOOSI, JOHAN LARSSON, University of Maryland — The modeling errors depend directly on the grid (or filter) spacing in turbulence-resolving simulations (LES, DNS, DES, etc), and are typically at least as significant as the numerical errors. This makes adaptive grid-refinement complicated, since it prevents the estimation of the local error sources through numerical analysis. The present work attempts to address this difficulty with a physics-based error-source indicator that accounts for the anisotropy in the smallest resolved scales, which can thus be used to drive an anisotropic grid-adaptation process. The proposed error indicator is assessed on a sequence of problems, including turbulent channel flow and flows in more complex geometries. The formulation is geometrically general and applicable to complex geometries.

2:35PM R33.00006 Crossover between two- and three-dimensional turbulence in spatial mixing layers

, LUCA BIANCOFIORE, Bilkent University — We investigate how the domain depth affects the turbulent behaviour in spatially developing mixing layers by means of large-eddy simulations (LES) based on a spectral vanishing viscosity technique.

Analyses of spectra of the vertical velocity, of Lumley's diagrams, of the turbulent kinetic energy and of the vortex stretching show that a two-dimensional behaviour of the turbulence is promoted in spatial mixing layers by constricting the fluid motion in one direction. This finding is in agreement with previous works on turbulent systems constrained by a geometric anisotropy, pioneered by Smith, Chasnov & Waleffe [*Phys. Rev. Lett.*, **77**, 2467-2470]. We observe that the growth of the momentum thickness along the streamwise direction is damped in a confined domain. A full two-dimensional turbulent behaviour is observed when the momentum thickness is of the same order of magnitude as the confining scale.

2:48PM R33.00007 Advection and the Efficiency of Spectral Energy Transfer in Two-Dimensional Turbulence

, NICHOLAS OUELLETTE, LEI FANG, Stanford University — We report measurements of the geometric alignment of the small-scale turbulent stress and the large-scale rate of strain that together lead to the net flux of energy from small scales to large scales in two-dimensional turbulence. We find that the instantaneous alignment between these two tensors is weak, and thus that the spectral transport of energy is inefficient. We show, however, that the strain rate is much better aligned with the stress at times in the past, suggesting that the differential advection of the two is responsible for the inefficient spectral transfer. We provide evidence for this conjecture by measuring the alignment statistics conditioned on weakly changing stress history. Our results give new insight into the relationship between scale-to-scale energy transfer, geometric alignment, and advection in turbulent flows.

3:01PM R33.00008 Janus spectra: cascades without local isotropy¹

, CHIEN-CHIA LIU, RORY CERBUS, PINAKI CHAKRABORTY, Okinawa Institute of Science and Technology — Two-dimensional turbulent flows host two disparate cascades: of enstrophy and of energy. The phenomenological theory of turbulence, which provides the theoretical underpinning of these cascades, assumes local isotropy. This assumption has been amply verified via computational, experimental and field data amassed to date. Local isotropy mandates that the streamwise (u) and transverse (v) velocity fluctuations partake in the same cascade; consequently, the attendant spectral exponents (α_u and α_v) of the turbulent energy spectra are the same, $\alpha_u = \alpha_v$. Here we report experiments in soap-film flows where α_u corresponds to the energy cascade, but concurrently α_v corresponds to the enstrophy cascade, as if two mutually independent turbulent fields of disparate dynamics were concurrently active within the flow. This species of turbulent energy spectra, which we term the Janus spectra, has never been observed or predicted theoretically. Remarkably, the tools of phenomenological theory can be invoked to elucidate this manifestly anisotropic flow.

¹Okinawa Institute of Science and Technology

3:14PM R33.00009 Direct and inverse energy cascades in strongly rotating turbulent flows¹

, GANAPATI SAHOO, IRENE MAZZITELLI, Department of Physics & INFN, University of Tor Vergata, Rome, Italy, PRASAD PERLEKAR, TIFR Centre for Interdisciplinary Sciences, Hyderabad, India, FABIO BONACCORSO, LUCA BIFERALE, Department of Physics & INFN, University of Tor Vergata, Rome, Italy —

Rotation plays a key role in many geophysical and astrophysical flows. Under a strong rotation rate (low Rossby numbers), three-dimensional turbulent flows show a tendency to develop fluctuations in a plane perpendicular to the rotation axis leading to a two-dimensional and three-components (2D3C) evolution. By using high resolution direct numerical simulations up to 4096^3 collocation points we present a systematic analysis of the 2D3C field and of the energy transport both concerning direct and inverse cascades using a decomposition in helical-Fourier modes.

¹Supported by ERC Advanced Grant (N. 339032) NewTURB

Tuesday, November 22, 2016 1:30PM - 3:14PM –

Session R34 Turbulence: DNS Simulations Oregon Ballroom 203 - Daniel Livescu, Los Alamos National Laboratory

1:30PM R34.00001 Time tracking and interaction of energy-eddies at different scales¹, JOSE I. CARDESA, ALBERTO VELA-MARTIN, JAVIER JIMENEZ, Technical University of Madrid — We study the energy cascade through coherent structures obtained in time-resolved simulations of incompressible, statistically steady isotropic turbulence. The structures are defined as geometrically connected regions of the flow with high kinetic energy. We compute the latter by band-pass filtering the velocity field around a scale r . We analyse the dynamics of structures extracted with different r , which are a proxy for eddies containing energy at those r . We find that the size of these “energy-eddies” scales with r , while their lifetime scales with the local eddy-turnover $r^{2/3}\epsilon^{-1/3}$, where ϵ is the energy dissipation averaged over all space and time. Furthermore, a statistical analysis over the lives of the eddies shows a slight predominance of the splitting over the merging process. When we isolate the eddies which do not interact with other eddies of the same scale, we observe a parent-child dependence by which, on average, structures are born at scale r during the decaying part of the life of a structure at scale $r' > r$. The energy-eddy at r' lives in the same region of space as that at r . Finally, we investigate how interactions between eddies at the same scale are echoed across other scales.

¹Funded by the ERC project Coturb

1:43PM R34.00002 Dynamic Mode Decomposition of Jet in Channel Crossflow¹, ZHAO WU, DOMINIQUE LAURENCE, Univ of Manchester — In this paper, the authors present a comparative analysis of Koopman modes computed from snap-shots of direct numerical simulations of a jet in channel crossflow (channel flow Re number =3333, jet-to-crossflow velocity ratio =1/6). The flow is complex due to interactions between the jet and the cross-flow, and contains geometry-dependent large-scale coherent structures; thus, the Koopman mode analysis provides a powerful tool for studying the spatial and spectral information of the flow. The Koopman modes are approximated by the DMD modes restricted to Krylov subspace, and the Koopman modes isolate structures associated with single frequency only. In this work, we address issues related to the physical interpretation of the DMD modes. The results show that the computed Koopman modes identify the relevant frequencies and the corresponding three-dimensional flow structures automatically. We present the selected DMD modes, which show big differences in the spatial structures and frequency. The shear layer vortices are separated from the horseshoe vortex. These modes have large amplitudes among all modes obtained.

¹Acknowledging the IT Services at The University of Manchester, the use of ARCHER HPC allocated via UK EPSRC Turbulence Consortium (EPSRC grant EP/L000261/1), the use of BlueGene/Q supercomputer sponsored by EDF RD centre Chatou and Cedric Flageul.

1:56PM R34.00003 Lagrangian statistics of turbulent dispersion from 8192³ direct numerical simulation of isotropic turbulence¹, DHAWAL BUARIA, Georgia Tech and ENS Lyon, France, P.K. YEUNG, Georgia Tech, B.L. SAWFORD, Monash University, Australia — An efficient massively parallel algorithm has allowed us to obtain the trajectories of 300 million fluid particles in an 8192³ simulation of isotropic turbulence at Taylor-scale Reynolds number 1300. Conditional single-particle statistics are used to investigate the effect of extreme events in dissipation and enstrophy on turbulent dispersion. The statistics of pairs and tetrads, both forward and backward in time, are obtained via post-processing of single-particle trajectories. For tetrads, since memory of shape is known to be short, we focus, for convenience, on samples which are initially regular, with all sides of comparable length. The statistics of tetrad size show similar behavior as the two-particle relative dispersion, i.e., stronger backward dispersion at intermediate times with larger backward Richardson constant. In contrast, the statistics of tetrad shape show more robust inertial range scaling, in both forward and backward frames. However, the distortion of shape is stronger for backward dispersion. Our results suggest that the Reynolds number reached in this work is sufficient to settle some long-standing questions concerning Lagrangian scale similarity.

¹Supported by NSF Grants CBET-1235906 and ACI-1036170

2:09PM R34.00004 DNS of turbulent Couette flow with transpiration - spectra and symmetry induced scaling laws.¹, SERGIO HOYAS, Instituto de Matematica Pura y Aplicada, UP Valencia, Spain, STEFANIE KRAHEBERGER, MARTIN OBERLACK, Chair of Fluid Dynamics, TU Darmstadt, Germany — We present DNS results of turbulent plane Couette flow with constant wall-normal transpiration for Reynolds numbers of $Re_\tau = 250, 500, 1000$ and several transpiration Reynolds numbers $Retr = V_0/U_w$. To obtain the DNS data, a pseudo-spectral code, which originally was developed at UP Madrid, see (Hoyas and Jiménez 2006), is used for the simulations. Due to the lack of experimental and DNS data, the convergence of every simulation has been validated using the total shear stress equation and the relation between the friction velocities at the lower and upper wall. Examining the spectra we found that the large and wide structures, which appear in pure Couette flow, see (Avsarkisov et al. 2014), are destroyed as soon as transpiration velocity is different from zero. This and the presence of anomalous spectra near the blowing wall indicates the strong influence of suction on the whole flow, which was observed in (Antonia et al. 1988) as well. As classical scaling laws are not valid due to transpiration, new scaling laws of the mean velocity are derived using Lie symmetry methods. Additionally, suction creates a comparably larger u_τ which, in turn, causes a flat and long region in the indicator function for the largest transpiration rate.

¹SH was partially funded by ENE2015-71333-R. SK was funded by DFG under grant no. OB96/39-1. Computer resources have been provided by LRZ Munich under grant pr92la.

2:22PM R34.00005 Multiscale modeling of turbulent channel flow over porous walls¹, SUDHAKAR YOGARAJ, UGIS LACIS, SHERVIN BAGHERI, KTH Royal Institute of Technology, Sweden — We perform direct numerical simulations of fully developed turbulent flow through a channel coated with a porous material. The Navier-Stokes equations governing the fluid domain and the Darcy equations of the porous medium are coupled using an iterative partitioned scheme. At the interface between the two media, boundary conditions derived using a multiscale homogenization approach are enforced. The main feature of this approach is that the anisotropic micro-structural pore features are directly taken into consideration to derive the constitutive coefficients of the porous media as well as of the interface. The focus of the present work is to study the influence of micro-structure pore geometry on the dynamics of turbulent flows. Detailed turbulence statistics and instantaneous flow field are presented. For comparison, flow through impermeable channel flows are included.

¹Supported by the European Unions Horizon 2020 research and innovation programme under the Marie Skłodowska-Curie grant agreement No 708281

2:35PM R34.00006 Lagrangian and Eulerian statistics in homogeneous, anisotropic flows¹, KARTIK IYER, FABIO BONACCORSO, Department of Physics and INFN, University of Rome, Tor Vergata, FEDERICO TOSCHI, Department of Applied Physics, University of Eindhoven, LUCA BIFERALE, Department of Physics and INFN, University of Rome, Tor Vergata — We report results from highly resolved direct numerical simulations of anisotropic homogeneous flows using up to 2048^3 collocations points. We examine a turbulent Kolmogorov flow with randomly correlated phases in order to recover space homogeneity on average. We present Eulerian and Lagrangian measurements concerning the universality of isotropic and anisotropic contributions using a systematic decomposition based on the eigenfunctions of the SO(3) group of rotations in three dimensions. Additionally, we discuss absolute dispersion statistics of particles in flows subjected to different large-scale anisotropies.

¹ERC ADG NewTURB 2013

2:48PM R34.00007 Lagrangian statistics in turbulent channel flow: implications for Lagrangian stochastic models¹, NICKOLAS STELZENMULLER, Universit de Grenoble Alpes, LEGI, JUAN IGANCIO POLANCO, IVANA VINKOVIC, Universit Lyon 1, INSA Lyon, Ecole Centrale Lyon, CNRS, LMFA UMR 5509, NICOLAS MORDANT, Universit de Grenoble Alpes, LEGI — Lagrangian acceleration and velocity correlations in statistically one-dimesional turbulence are presented in the context of the development of Lagrangian stochastic models of inhomogeneous turbulent flows. These correlations are measured experimentally by 3D PTV in a high aspect ratio water channel at $Re_\tau = 1450$, and numerically from DNS performed at the same Reynolds number. Lagrangian timescales, key components of Lagrangian stochastic models, are extracted from acceleration and velocity autocorrelations. The evolution of these timescales as a function of distance to the wall is presented, and compared to similar quantities measured in homogeneous isotropic turbulence. A strong dependance of all Lagrangian timescales on wall distance is present across the width of the channel. Significant cross-correlations are observed between the streamwise and wall-normal components of both acceleration and velocity. Lagrangian stochastic models of this flow must therefore retain dependance on the wall-normal coordinate and the components of acceleration and velocity, resulting in significantly more complex models than those used for homogeneous isotropic turbulence.

¹We gratefully acknowledge funding from the Agence Nationale de la Recherche, LabEx Tec 21, and CONICYT Becas Chile

3:01PM R34.00008 Statistics of Vortical Structures in Variable-Density Turbulent Mixing Layers, JON BALTZER, DANIEL LIVESCU, Los Alamos National Laboratory — Direct Numerical Simulations are performed of temporal incompressible shear-driven planar mixing layers between two miscible streams of fluids with different densities. The simulations begin from thin disturbed interfaces and develop into self-similar states. We use very large domain sizes, corresponding to grids of up to $6144 \times 2048 \times 1536$ points, to produce high-quality statistics and allow natural growth of turbulent structures. A wide range of Atwood numbers are explored, ranging from nearly constant density to $A=0.87$ (or a density ratio of 14). At high Atwood numbers, a variety of statistics show that variable-density effects produce significant asymmetries. Here we focus on the differences in vortical structure of the light and heavy fluid streams and the importance of non-Boussinesq effects as Atwood number increases. Detailed budgets of vorticity moments are examined in conjunction with the alignments of vorticity relative to other flow quantities. The results display the variable density effects due to compositional variations, a distinctly different mechanism from the density variations associated with compressibility in high-speed flows.

Tuesday, November 22, 2016 1:30PM - 3:40PM –
Session R35 Turbulence: Planetary Boundary Layer Oregon Ballroom 204 - William Anderson, UT Dallas

1:30PM R35.00001 Modeling turbulent flows in the atmospheric boundary layer of Mars: application to Gale crater, Mars, landing site of the Curiosity rover.¹, WILLIAM ANDERSON, UT Dallas, KENZIE DAY, GARY KOCUREK, UT Austin — Mars is a dry planet with a thin atmosphere. Aeolian processes – wind-driven mobilization of sediment and dust – are the exclusive mode of landscape variability on Mars. Craters are common topographic features on the surface of Mars, and many craters on Mars contain a prominent central mound (NASA's *Curiosity* rover was landed in Gale crater). Using density-normalized large-eddy simulations, we have modeled turbulent flows over crater-like topographies that feature a central mound. We have also run one simulation of flow over a digital elevation map of Gale crater. Resultant datasets suggest a deflationary mechanism wherein vortices shed from the upwind crater rim are realigned to conform to the crater profile via stretching and tilting. This was accomplished using three-dimensional datasets (momentum and vorticity) retrieved from LES. As a result, helical vortices occupy the inner region of the crater and, therefore, are primarily responsible for aeolian morphodynamics in the crater. We have also used the immersed-boundary method body force distribution to compute the aerodynamic surface stress on the crater. These results suggest that secondary flows – originating from flow separation at the crater – have played an important role in shaping landscape features observed in craters (including the dune fields observed on Mars, many of which are actively evolving).

¹None

1:43PM R35.00002 ABSTRACT WITHDRAWN –

1:56PM R35.00003 Turbulent transitions in the stable boundary layer: Couette and Poiseuille flow¹, AMBER M. HOLDSWORTH, ADAM H. MONAHAN, University of Victoria — The stable boundary layer (SBL) can be classified into two distinct regimes. The weakly stable regime (WSBL) which occurs in the presence of moderate to strong pressure gradients or cloudy skies and is characterized by continuous turbulent mixing, and the very stable regime (VSBL) which occurs in the presence of weak pressure gradients or clear skies and turbulence weakens to the point of collapse. Modelling and observational results indicate that transitions from the WSBL to the VSBL occur when the maximum sustainable heat flux (MSHF), or shear capacity, is exceeded. The collapse of turbulence in the SBL is investigated using a one dimensional model of Couette flow with a constant heat flux. We show that the MSHF framework for predicting turbulent collapse is qualitatively robust to the choice of turbulence parameterization and extend these earlier stability analyses by numerically determining the unstable modes along the unstable branch. To explore transitions between the VSBL and the WSBL we extend the model to include a horizontal pressure gradient and a surface radiation scheme. Analysis of the Poiseuille flow demonstrates how the idealized energy/momentun budget model with parameterized turbulence can reproduce the regime transitions present in atmospheric data.

¹We acknowledge support from NSERC and the computing facilities of Westgrid and Compute Canada.

2:09PM R35.00004 Reduced-Basis Determination of Planetary Boundary-Layer Flow Statistics for a Novel Turbulence Model¹, JOSEPH SKITKA, BRAD MARSTON, Brown University, Department of Physics, BAYLOR FOX-KEMPER, Brown University, Department of Earth, Environmental and Planetary Sciences — Uncertainty in climate modeling and weather forecasting can largely be attributed to the omission or inaccurate representation of oceanic and atmospheric subgrid processes. Existing subgrid turbulence models are built on assumptions of isotropy, homogeneity, and the locality of correlations. Direct statistical simulation (DSS) using expansion in equal-time cumulants is a novel approach to subgrid modeling that does not make these assumptions. In prior work, a second-order closure, CE2, was shown to capture important vertical turbulent transports in Langmuir turbulence and Rayleigh-Bénard convection, but to run efficiently, this approach to turbulence modeling requires a drastic reduction in dimensionality. The present work addresses how accurately these systems can be represented with a truncated principal orthogonal decomposition (POD). The representation of turbulent transports by truncated POD bases are studied by static projection of fully resolved statistics and dynamical evolution of a reduced model. Results indicate the projected truncated turbulent statistics in these flows are less sensitive to flow details, like mixed-layer depth, than the truncated basis itself. The question of whether POD is an optimal truncation technique for these purposes is considered.

¹NSF DMR 1306806, NSF GCE 1350795, The Institute at Brown for Environment and Society Graduate Student Fellowship

2:22PM R35.00005 Physics-based Enrichment of Planetary Boundary Layer LES, ADITYA GHATE, SANJIVA LELE, Stanford University — A new multiscale simulation methodology is introduced to facilitate efficient simulations of very high Reynolds number wall bounded flows such as the PBL. The two-simulation, one-way coupled, scale splitting methodology combining a) Non-linear wave space model using the Gabor Transform and spectral eddy-viscosity, b) Representation of the subfilter fields via a set of random modes, and c) Large Eddy Simulation using a robust subgrid scale model, is introduced. The viability of the methodology is investigated using 3 increasingly sophisticated idealizations for the PBL. In the first idealization, the surface layer is approximated using a uniform shear and a positive (stable) temperature gradient which makes the problem homogeneous. The second idealization models the PBL as a constant pressure gradient driven half channel thus introducing inhomogeneity in the vertical direction. The high latitude Stable PBL used in GABLs1 intercomparison study (Beare et. al. BLM 2006) serves as the third idealization for the PBL and it further introduces Coriolis and Stratification effects. These idealizations help validate the two-simulation methodology, where comparisons are made in terms of statistics such as space-time correlations, k-omega spectra and profiles of second order correlations.

2:35PM R35.00006 Flow within and above heterogeneous and homogeneous canopies., ALI M. HAMED, MATTHEW J. SADOWSKI, LEONARDO P. CHAMORRO, University of Illinois at Urbana-Champaign — The flow development above and within homogeneous and heterogeneous canopies was studied using planar and stereo PIV in a refractive-index-matching open channel. The homogeneous model is constituted of elements of height h arranged in staggered configuration; whereas the heterogeneous canopy consisted of elements of two heights $h_1 = h + 1/3 h$ and $h_2 = h - 1/3 h$ alternated every two rows. Both canopies had the same roughness density, element geometry, and mean height. The flow was studied under three submergences $H/h = 2, 3, 4$, where H denotes the flow depth. Turbulence statistics complemented with quadrant analysis and proper orthogonal decomposition reveal richer flow dynamics induced by height heterogeneity. Topography-induced spatially-periodic mean flows are observed for the heterogeneous canopy. In contrast to the homogeneous case, non-vanishing vertical velocity is maintained across the entire length of the heterogeneous canopy with increased levels at lower submergence depths. The results indicate that heterogeneous canopies exhibit greater vertical turbulent exchange at the canopy interface, suggesting a potential for greater scalar exchange and greater impact on channel hydraulic resistance.

2:48PM R35.00007 Flow over a model boreal forest canopy and its dependence on canopy density¹, EMILY MOORE, GRAHAM FREEDLAND, TAMARA DIB, RAL BAYON CAL, None — Deforestation occurs due to the rapidly growing population demand for more space and resources. Effects on the canopy density are observed via a wind tunnel experiment. The flow field of the boreal forest canopy is obtained via particle image velocimetry where mean velocities and Reynolds stresses are evaluated. A scaled model that includes the traits of vegetative structures is used to represent a boreal forest within the test section of the wind tunnel.

¹National Science Foundation

3:01PM R35.00008 A study of the role of convective stratification and rates of aeolian activity on arid landscapes.¹, CHINTHAKA JACOB, UT Dallas, JOHN STOUT, US Dept. Agriculture, WILLIAM ANDERSON, UT Dallas — Aeolian activity – wind-driven mobilization of sediment and dust – is driven by aerodynamic surface stress. Existing models for aeolian activity scale mass flux on shear velocity to an exponent that exceeds unity, which demonstrates the role of turbulence in mobilizing sediment and dust. Large-eddy simulation (LES) was used to model neutrally stratified atmospheric boundary layer flows; a computational domain with very long streamwise extent was used to capture large- and very-large-scale motions. A time-series of local surface stress was used to generate a probability density function of stress, which was used to guide the selection of conditional-sampling thresholds. Results show that high stress events are caused by the passage of large scale inclined coherent structures composed of uniform momentum excesses, which are flanked on either side by low-stress regions (the opposite is true when conditioned on low stress events). Since surface heating during the daytime induces buoyancy fluxes that result in additional turbulence production (this is, in addition to production via mechanical shear), we have repeated the aforementioned simulations with convective heating. Parameters of LES cases are set to mimic flat, arid landscape with different heat flux forcing. The variation of structural inclination angle displays good general agreement with previously reported results, varying systematically with the Monin-Obukhov stability parameter under different stability conditions.

¹National Science Foundation, Grant AGS-1500224

3:14PM R35.00009 Numerical and experimental study of flow over stages of an offset merger dune interaction.¹, CHAO WANG, UT Dallas, ZHANQI TANG, Hebei University of Technology, NATHANIEL BRISTOW, GIANLUCA BLOIS, KEN CHRISTENSEN, University of Notre Dame, WILLIAM ANDERSON, UT Dallas — Results of unidirectional turbulent flows over barchan dunes at high Reynolds number are presented. In order to capture the inertial-dominated dynamics typical of these environmental flows, complementary large-eddy simulations (LES) and experimental measurements have been used. A series of dune field topographies have been considered wherein a small dune is positioned at different positions upflow of a large dune, from a spanwise-offset position. The smaller dune is geometrically similar, but one-eighth the volume of the larger dune, thus replicating instantaneous realizations during actual dune interactions in laboratory or natural settings. Experimental measurement and LES are both used to study these configurations, with strong agreement reported between resultant datasets. We report that flow channeling in the interdune space induces a mean flow heterogeneity – termed “wake veering” – in which the location of maximum momentum deficit in the dune wake is spanwise-displaced. Elevated turbulent stresses are observed in the shear layers flanking the channeling flow. Finally, spatial distributions of surface stress from LES have been used to identify locations of elevated erosion, predicting bedform migration patterns. Results show that locations of minimal erosion – whether associated with upflow sheltering or with vanishing spatial gradients of dune height – constitute spatial “junctions” of coalescing, proximal dunes.

¹National Science Foundation, Grant CBET-1603254

3:27PM R35.00010 Refractive index matched PIV measurements of flow around interacting barchan dunes., NATHANIEL BRISTOW, GIANLUCA BLOIS, TAEHOON KIM, University of Notre Dame, JAMES BEST, University of Illinois at Urbana-Champaign, KENNETH CHRISTENSEN, University of Notre Dame — Barchan dunes are crescent shaped bedforms found in both Aeolian and subaqueous environments, including deserts, river beds, continental shelves, and even the craters of Mars. The evolution of and dynamics associated with these mobile bedforms involve a strong degree of coupling between sediment transport, morphological change, and flow, the last of which represents the weakest link in our current understanding of barchan morphodynamics. Their three-dimensional geometry presents experimental challenges for measuring the full flow field, particularly around the horns and in the leeside of the dunes. In this study we present measurements of the turbulent flow surrounding fixed barchan dune models in various configurations using particle image velocimetry in a refractive index matching flume environment. The refractive index matching technique opens the door to making measurements in wall-parallel planes surrounding the models, as well as wall-normal plane measurements in the leeside region between the horns. While fixed bed experiments are unable to directly measure sediment transport, they allow us to focus solely on the flow physics and full resolution of the turbulent flow field in ways that are otherwise not possible in mobile bed experiments.

Tuesday, November 22, 2016 1:30PM - 3:01PM –
Session R36 Drops: Impacts: Heat Transfer and Leidenfrost Portland Ballroom 251 - Pirouz Kavehpour, UCLA

1:30PM R36.00001 Heat transfer and phase change in an impinging droplet, AYSAN RANGCHIAN, University of California, Los Angeles, NIKKI L. SHIRAZI, Oakwood School, H. PIROUZ KAVEHPOUR, University of California, Los Angeles — Non isothermal droplet impact on solid surfaces has several industrial applications such as spray cooling and 3D printing. Impinging of a droplet on a surface involves an initial phase of spreading followed by a subsequent return to the equilibrium shape. Thermal energy exchanged within the droplet fluid as well as between liquid/solid during the impact has been studied using an ultra high speed infrared camera. Variable parameters in the experiment include droplet temperature and kinetic energy of the droplet during the impact. The evolution of droplet shape viewed by IR camera is similar to what previously observed by high speed photography. The thermal map of droplet over time in these experiments agrees with previously reported numerical simulation. In addition, spacial and temporal temperature variations of liquid droplets on a surface as they solidify are presented. IR camera provides an accurate temperature diagram as the phase change occurs, which is essential for understanding the physics of 3D printing.

1:43PM R36.00002 Theoretical and Experimental Analyses of Molten Droplet Impact on Cold Substrates, ELAHEH ALIZADEH-BIRJANDI, H. PIROUZ KAVEHPOUR, University of California, Los Angeles (UCLA) — Spreading of liquid drop on cold solid substrates is a complicated problem that involves heat transfer, fluid dynamics, and phase change physics with the combination of complex wetting behavior of contact line. Many researchers are trying to obtain the final shape of the droplet or in other words the contact angle and radius of the drop after the solidification is complete. Understanding the physics behind the non-isothermal spreading of droplet is of utmost importance owing to its broad applications in diverse areas of industry. This work mainly focuses on obtaining important physical parameters involved in the process of spreading of molten droplets as well as controlling the post-solidification geometry of droplets. A complete set of experimental study is performed that shows the final radius in the case of free fall of droplet under high impact velocity is independent of the initial condition of the impact including the impact velocity and temperature gradients. The analytical modeling of the problem also verifies the accuracy of these results.

1:56PM R36.00003 Impact of Metal Droplets: A Numerical Approach to Solidification¹, ROBIN KOLDEWEIJ, University of Twente, RAJESH MANDAMPARAMBIL, TNO, DETLEF LOHSE, University of Twente — Layer-wise deposition of material to produce complex products is a subject of increasing technological relevance. Subsequent deposition of droplets is one of the possible 3d printing technologies to accomplish this. The shape of the solidified droplet is crucial for product quality. We employ the volume-of-fluid method (in the form of the open-source code Gerris) to study liquid metal (in particular tin) droplet impact. Heat transfer has been implemented based on the enthalpy approach for the liquid-solid phase. Solidification is modeled by adding a sink term to the momentum equations, reducing Navier-Stokes to Darcy's law for high solid fraction. Good agreement is found when validating the results against experimental data ². We then map out a phase diagram in which we distinguish between solidification behavior based on Weber and Stefan number. In an intermediate impact regime impact, solidification due to a retracting phase occurs. In this regime the maximum spreading diameter almost exclusively depends on Weber number. Droplet shape oscillations lead to a broad variation of the morphology of the solidified droplet and determine the final droplet height.

¹TNO

²S.D.Aziz, S.Chandra, Int. J. of Heat and Mass Trans. 43 (2000)

2:09PM R36.00004 The impact of a Leidenfrost drop on a spoked surface texture ,

SAMIRA SHIRI, Boston University, COLIN PATTERSON, GE Aviation; Boston University, JAMES BIRD, Boston University — Liquid drops can bounce when they impact nonwetting surfaces. Recently, studies have demonstrated that the time that the bouncing drop contacts a superhydrophobic surface can be reduced by incorporating ridged macrotextures on the surface. Yet the existing models aimed at explaining this phenomenon offer incompatible predictions of the contact time when a drop impacts multiple intersecting macrotextures, or spokes. Furthermore, it is unclear whether the effects of the macrotexture on the drop hydrodynamics extend to non-wetting surfaces in which direct contact is avoided by a thin vapor layer. Here we demonstrate that the phenomenon observed for macrotextured, superhydrophobic surfaces extends to macrotextured, wettable surfaces above the Leidenfrost temperature. We show that the number of droplets and overall residence time both depend on the number of intersecting spokes. Finally, we compare and contrast our results with mechanistic models to rationalize various elements of the phenomenon.

2:22PM R36.00005 An experimental study of the dynamic Leidenfrost phenomenon ,

MOHAMMAD KHAVARI, School of Mechanical & Aerospace Engineering, Nanyang Technological University, 50 Nanyang Avenue, 639798, Singapore, MOHAMMAD S.M. SAIFULLAH, Institute of Materials Research and Engineering, A*STAR, 2 Fusionopolis Way, 138634 Singapore, TUAN TRAN, School of Mechanical & Aerospace Engineering, Nanyang Technological University, 50 Nanyang Avenue, 639798, Singapore — Complete separation between an impacting droplet and a superheated surface can be achieved if the surface temperature is sufficiently high causing spontaneous generation of a vapor layer under the droplet. The transition to such vapor-induced separation, or Leidenfrost regime, depends on numerous parameters such as materials properties and the impact conditions including the impact velocity and surface temperature. Here we provide detailed experimental observations of several distinct impact dynamics at the Leidenfrost transition in order to understand the physical mechanism of such transition. We focus on the liquid-solid interface to identify necessary conditions for Leidenfrost transition to occur. We show that detailed and quantitative measurements of the wetted area during impact may lead to a physical understanding of the Leidenfrost phenomenon.

2:35PM R36.00006 Animating Impacting Spheres with the Elastic Leidenfrost Effect¹ ,

SCOTT WAITUKAITIS, ANTON SOUSLOV, MARTIN VAN HECKE, Univ of Leiden — Liquid droplets impacting on hot surfaces above the Leidenfrost temperature can squeeze out the vapor layer and enter the contact boiling regime. What happens to soft but vaporizable solids, such as hydrogel spheres, under such conditions? I will show how this combination leads to sustained bouncing dynamics. The key physics is the coupling between the sphere's elastic deformations and vaporization. Beyond being a new facet of the Leidenfrost effect, this phenomenon promises to be useful in fields such as fluid dynamics, microfluidics, and active matter.

¹NWO Veni and Vici Programs

2:48PM R36.00007 Thermo-responsive droplet deposition and solidification , MAZIYAR

JALAAL, Department of Mechanical Engineering, University of British Columbia, CAROLA SEYFERT, Institute of Fluid Mechanics, Technical University of Dresden, BORIS STOEGER, Department of Mechanical Engineering, University of British Columbia, NEIL BALMFORTH, Department of Mathematics, University of British Columbia — The spreading of a thermo-responsive droplet on a heated surface is studied. The spatio-temporal pattern of gel formation within the droplet is visualized using a new experimental method based on spectral domain optical coherence tomography. The method relies on a collective motion of sub-micron buoyant particles inside the droplet. The mechanisms that lead to the arrest of the spreading droplet are explored. The importance of evaporation-induced gel formation and heat conduction through surrounding air are highlighted. The proposed experimental technique can potentially be used to analyze the solidification of different fluids such as molten waxes. Thermo-responsivity is demonstrated to provide an effective control over the final shape of the droplet.

Tuesday, November 22, 2016 1:30PM - 2:35PM –

Session R37 Drops: General Portland Ballroom 252 - William Ristenpart, University of California Davis

1:30PM R37.00001 A parametric study on the rise of a pair of bubbles using algebraic volume of fluid method: effect of diameter and viscosity ratio.¹ , AMARESH DALAL, AMOL C KULKARNI, JAI MANIK, GANESH NATARAJAN, Indian Inst of Tech-Guwahati — The effect of droplet diameter and viscosity ratio on the coalescence of two bubbles rising in a quiescent liquid has been studied numerically using algebraic volume of fluid (VOF) method. If the upper bubble diameter is 75 % of the lower bubble, the time taken for their coalescence increases in comparison with the case of equal bubble diameter. For the case, when the diameter of the upper bubble is reduced, this delay may be attributed to comparatively weaker jet formed behind the leading bubble, ultimately resulting in lesser acceleration of the trailing bubble. While for the other case, when the diameter of the lower bubble is reduced, it is because of a totally different scenario of liquid entrapment observed during coalescence. The effect of viscosity of the surrounding fluid is also noticed separately for the situation when the diameters of the bubbles are equal. It has been observed that, the increase in viscosity of the surrounding fluid will increase the form drag over the bubbles, eventually leading to the delay in their coalescence.

¹This study is funded by a grant from BRNS, DAE, Government of India.

1:43PM R37.00002 Charged Water Droplets can Melt Metallic Electrodes , ERIC ELTON,

ETHAN ROSENBERG¹, WILLIAM RISTENPART, Dept. Chemical Engineering, University of California Davis — A water drop, when immersed in an insulating fluid, acquires charge when it contacts an energized electrode. Provided the electric field is strong enough, the drop will move away to the opposite electrode, acquire the opposite charge, and repeat the process, effectively 'bouncing' back and forth between the electrodes. A key implicit assumption, dating back to Maxwell, has been that the electrode remains unaltered by the charging process. Here we demonstrate that the electrode is physically deformed during each charge transfer event with an individual water droplet or other conducting object. We used optical, electron, and atomic force microscopy to characterize a variety of different metallic electrodes before and after drops were electrically bounced on them. Although the electrodes appear unchanged to the naked eye, the microscopy reveals that each charge transfer event yielded a crater approximately 1 micron wide and 50 nm deep, with the exact dimensions proportional to the applied field strength. We present evidence that the craters are formed by localized melting of the electrodes via Joule heating in the metal and concurrent dielectric breakdown of the surrounding fluid, suggesting that the electrode locally achieves temperatures exceeding 3400°C.

¹Present address: Dept. Materials Sci. Engineering, MIT

1:56PM R37.00003 Direct cascade of droplet fragmentation in sparkling fireworks ,

CHIHIRO INOUE, The University of Tokyo, EMMANUEL VILLERMAUX, Aix Marseille Universite — Sparkling fireworks are popular in Japan for 400 years. They 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, ejecting droplets seen as light streaks. The droplets successively fragment up to eight times leading to ever-smaller size and their branched light streaks traces are similar to pine needles. The phenomenon involves chemical reactions, thermal decomposition of metastable compounds, gas bubble nucleation and bursting, the formation of liquid ligaments, and droplets. We depict the features of this unique direct fragmentation cascade in nature, where thermal diffusion inside each droplet determines its lifetime. Droplet radius getting exponentially smaller until thermal production and heat losses equilibrate. We compute and document experimentally the number of fragmentation steps, and the overall size of the light streaks as well.

2:09PM R37.00004 ABSTRACT WITHDRAWN —**2:22PM R37.00005 ABSTRACT WITHDRAWN —****Tuesday, November 22, 2016 1:30PM - 3:40PM —**

Session R38 Flow Instability: Computations and Modeling Portland Ballroom 255 - Fernando Grinstein, Los Alamos National Lab

1:30PM R38.00001 Initial Conditions and Modeling for Shock Driven Turbulence. ,

FERNANDO GRINSTEIN, Los Alamos Natl Lab — We focus on the simulation of shock-driven material mixing driven by flow instabilities and initial conditions. Beyond complex multi-scale resolution of shocks and variable density turbulence, we must address the equally difficult problem of predicting flow transition promoted by energy deposited at the material interfacial layer during the shock interface interactions. Transition involves unsteady large-scale coherent-structure dynamics which can be captured by LES, but not by URANS based on equilibrium turbulence assumptions and single-point-closure modeling. Such URANS is frequently preferred on the engineering end of computation capabilities for full-scale configurations – and with reduced 1D/2D dimensionality being also a common aspect. With suitable initialization around each transition – e.g., reshock, URANS can be used to simulate the subsequent near-equilibrium weakly turbulent flow. We demonstrate 3D state-of-the-art URANS performance in one such flow regime. We simulate the CEA planar shock-tube experiments by Poggi et al. (1998) with an ILES strategy. Laboratory turbulence and mixing data are used to benchmark ILES. In turn, the ILES generated data is used to initialize and as reference to assess state-of-the-art 3D URANS. We find that by prescribing physics-based 3D initial conditions and allowing for 3D flow convection with just enough resolution, the additionally computed dissipation in 3D URANS effectively blends with the modeled dissipation to yield significantly improved statistical predictions.

1:43PM R38.00002 Shock driven multiphase flow with particle evaporation , JEEVAN DAHAL, JACOB MCFARLAND, Univ of Missouri - Columbia —

The computational study of the shock driven instability of a multiphase system with particle evaporation is presented. The particle evaporation modifies the evolution of the interface due to the addition of the vapor phase to the gas. The effects can be quantitatively measured by studying various gas parameters like density, temperature, vorticity and particle properties like diameter and temperature. In addition, the size distribution of particles also modifies the development of instability as the larger size particles damp the evolution of interface in comparison to the smaller size particles. The simulation results are presented to study these effects using FLASH developed at the FLASH Center at the University of Chicago. The capabilities of FLASH for particle modeling were extended using the Particle in Cell (PIC) technique for coupling of mass, momentum, and energy between the particle and carrier gas. A seeded cylinder of gas with particles having either a single radius or a distribution of radii was studied. The enstrophy production and destruction mechanisms were explored to understand the reason for change in vorticity with particle size.

1:56PM R38.00003 Application of Self-Similarity Constrained Reynolds-Averaged Turbulence Models to Rayleigh-Taylor and Richtmyer-Meshkov Unstable Turbulent Mixing¹ , TUCKER A. HARTLAND, Emory University, OLEG SCHILLING, Lawrence Livermore National Laboratory — Analytical self-similar solutions corresponding to Rayleigh-Taylor, Richtmyer-Meshkov and Kelvin-Helmholtz instability are combined with observed values of the growth parameters in these instabilities to derive coefficient sets for $K-\epsilon$ and $K-L-a$ Reynolds-averaged turbulence models. It is shown that full numerical solutions of the model equations give mixing layer widths, fields, and budgets in good agreement with the corresponding self-similar quantities for small Atwood number. Both models are then applied to Rayleigh-Taylor instability with increasing density contrasts to estimate the Atwood number above which the self-similar solutions become invalid. The models are also applied to a reshocked Richtmyer-Meshkov instability, and the predictions are compared with data. The expressions for the growth parameters obtained from the similarity analysis are used to develop estimates for the sensitivity of their values to changes in important model coefficients. Numerical simulations using these modified coefficient values are then performed to provide bounds on the model predictions associated with uncertainties in these coefficient values.

¹This work was performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract DE-AC52-07NA27344. This work was supported by the 2016 LLNL High-Energy-Density Physics Summer Student Program.

2:09PM R38.00004 A Comparative Analysis of Reynolds-Averaged Navier-Stokes Model Predictions for Rayleigh-Taylor Instability and Mixing with Constant and Complex Accelerations¹ , OLEG SCHILLING, Lawrence Livermore National Laboratory —

Two-, three- and four-equation, single-velocity, multicomponent Reynolds-averaged Navier-Stokes (RANS) models, based on the turbulent kinetic energy dissipation rate or lengthscale, are used to simulate $At = 0.5$ Rayleigh-Taylor turbulent mixing with constant and complex accelerations. The constant acceleration case is inspired by the Cabot and Cook (2006) DNS, and the complex acceleration cases are inspired by the unstable/stable and unstable/neutral cases simulated using DNS (Livescu, Wei & Petersen 2011) and the unstable/stable/unstable case simulated using ILES (Ramaprabhu, Karkhanis & Lawrie 2013). The four-equation models couple equations for the mass flux a and negative density-specific volume correlation b to the $K-\epsilon$ or $K-L$ equations, while the three-equation models use a two-fluid algebraic closure for b . The lengthscale-based models are also applied with no buoyancy production in the L equation to explore the consequences of neglecting this term. Predicted mixing widths, turbulence statistics, fields, and turbulent transport equation budgets are compared among these models to identify similarities and differences in the turbulence production, dissipation and diffusion physics represented by the closures used in these models.

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

2:22PM R38.00005 Adjoint-based approach to Enhancing Mixing in Rayleigh-Taylor Turbulence, ALI KORD¹, JESSE CAPECELATRO², Univ of Michigan - Ann Arbor — A recently developed adjoint method for multi-component compressible flow is used to measure sensitivity of the mixing rate to initial perturbations in Rayleigh-Taylor (RT) turbulence. Direct numerical simulations (DNS) of RT instabilities are performed at moderate Reynolds numbers. The DNS are used to provide an initial prediction, and the corresponding space-time discrete-exact adjoint provides a sensitivity gradient for a specific quantity of interest (QoI). In this work, a QoI is defined based on the time-integrated scalar field to quantify the mixing rate. Therefore, the adjoint solution is used to measure sensitivity of this QoI to a set of initial perturbations, and inform a gradient-based line search to optimize mixing. We first demonstrate the adjoint approach in the linear regime and compare the optimized initial conditions to the expected values from linear stability analysis. The adjoint method is then used in the high Reynolds number limit where theory is no longer valid. Finally, chaos is known to contaminate the accuracy of the adjoint gradient in turbulent flows when integrated over long time horizons. We assess the influence of chaos on the accuracy of the adjoint gradient to guide the work of future studies on adjoint-based sensitivity of turbulent mixing.

¹PhD Student, Department of Mechanical Engineering, University of Michigan, Ann Arbor, MI

²Assistant Professor, Department of Mechanical Engineering, University of Michigan, Ann Arbor, MI

2:35PM R38.00006 Vortical Effects on the Compressible Rayleigh-Taylor Instability, SCOTT WIELAND, University of Colorado Boulder, DANIEL LIVESCU, Los Alamos National Laboratory, OLEG V. VASILYEV, University of Colorado Boulder, SCOTT J. RECKINGER, Montana State University — High fidelity wavelet based direct numerical simulations (WDNS) of compressible, miscible, and single mode Rayleigh Taylor instability (RTI) with a stratified background density have been completed in 2 and 3 dimensions. As the instability grows, vorticity dynamics are largely responsible for the self-propagation and growth of the bubble and spike. However, in the presence of a background stratification, the vortex interactions are significantly altered. In the case of low Atwood number RTI, this leads to previously unseen regimes, namely, the exaggeration of bubble and spike asymmetries for a weakly stratified background state and the complete suppression of RTI growth in the strongly stratified scenario. To better understand these results, the vorticity transport equation budget was compared to the simplified scenarios of vortex pairs (2D) and vortex rings (3D) moving in a stratified medium.

2:48PM R38.00007 Three-Dimensional DSMC Simulations of the Rayleigh-Taylor Instability in Gases, T.P. KOEHLER, M.A. GALLIS, J.R. TORCZYNSKI, S.J. PLIMPTON, Sandia National Laboratories — 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 by 1-mm by 4-mm cuboid uniformly divided into 62.5 billion cubical cells. A total of 1 trillion computational molecules are used, and time steps of 0.1 ns are used. Simulations are performed to quantify the growth of perturbations on an initially flat interface as a function of the Atwood number. The DSMC results reproduce many features of the RTI and are in reasonable agreement with theoretical and empirical models. Consistent with previous work, the DSMC simulations indicate that the growth of the RTI follows a universal behavior. The numbers of bubble-spike pairs that eventually appear agree with theoretical values based on the most unstable wavelength and are independent of the statistical representation of the gas. 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.

3:01PM R38.00008 On new non-modal hydrodynamic stability modes and resulting non-exponential growth rates - a Lie symmetry approach, MARTIN OBERLACK, Tech Univ Darmstadt, ANDREAS NOLD, Imperial College London, CEDRIC WILFRIED SANJON, YONGQI WANG, JAN HAU, Tech Univ Darmstadt — Classical hydrodynamic stability theory for laminar shear flows, no matter if considering long-term stability or transient growth, is based on the normal-mode ansatz, or, in other words, on an exponential function in space (stream-wise direction) and time. Recently, it became clear that the normal mode ansatz and the resulting Orr-Sommerfeld equation is based on essentially three fundamental symmetries of the linearized Euler and Navier-Stokes equations: translation in space and time and scaling of the dependent variable. Further, Kelvin-mode of linear shear flows seemed to be an exception in this context as it admits a fourth symmetry resulting in the classical Kelvin mode which is rather different from normal-mode. However, very recently it was discovered that most of the classical canonical shear flows such as linear shear, Couette, plane and round Poiseuille, Taylor-Couette, Lamb-Oseen vortex or asymptotic suction boundary layer admit more symmetries. This, in turn, led to new problem specific non-modal ansatz functions. In contrast to the exponential growth rate in time of the modal-ansatz, the new non-modal ansatz functions usually lead to an algebraic growth or decay rate, while for the asymptotic suction boundary layer a double-exponential growth or decay is observed.

3:14PM R38.00009 Motion of multiple superposed viscous fluids, MAGNUS VARTDAL, Norwegian Defence Research Establishment (FFI) — In this study, the initial-value problem arising from small-amplitude disturbances on the interfaces between multiple superposed viscous fluids is analysed. First, linearized governing equations for the evolution of the amplitudes, valid in the general case, are presented. These equations are then used to study the effect of the presence of nearby interfaces on the initial growth-rate of a Rayleigh-Taylor instability. The present work is an extension of the analysis of Prosperetti (Prosperetti, A. (1981). Motion of two superposed viscous fluids. Physics of Fluids (1958-1988), 24(7), 1217-1223) to the multiple interface case.

3:27PM R38.00010 Stabilization of a finite slice in miscible displacement in homogeneous porous media, SATYAJIT PRAMANIK, NORDITA, MANORANJAN MISHRA, Indian Institute of Technology Ropar, India — We numerically studied the miscible displacement of a finite slice of variable viscosity and density. The stability of the finite slice depends on different flow parameters, such as displacement velocity U , mobility ratio R , and the density contrast. Series of numerical simulations corresponding to different ordered pair (R, U) in the parameter space, and a given density contrast reveal six different instability regions. We have shown that independent of the width of the slice, there always exists a region of stable displacement, and below a critical value of the slice width, this stable region increases with decreasing slice width. Further we observe that the viscous fingering (buoyancy-induced instability) at the upper interface induces buoyancy-induced instability (viscous fingering) at the lower interface. Besides the fundamental fluid dynamics understanding, our results can be helpful to model CO₂ sequestration and chromatographic separation.

Tuesday, November 22, 2016 1:30PM - 3:27PM –
Session R39 Bio: Suspensions and Colloids Portland Ballroom 256 - Sophie Ramananarivo, UC San Diego

1:30PM R39.00001 Manipulating colloidal assemblies with active dopants, SOPHIE RAMANARIVO, JEREMIE PALACCI, UC San Diego — The dynamics of a densely packed 2D layer of colloids can be significantly altered upon introducing a small amount of active microparticles. Those motile intruders drive the system out-of-equilibrium, which produces a variety of new complex phenomena such as the accentuation of density heterogeneities or the reorganization of crystalline colloidal structures. We investigate the altered dynamics of the passive spheres, as well as the behavior of micro-swimmers propelling in such crowded environment where interactions with passive obstacles or other active units become important. Ultimately, understanding and controlling such mixed systems could open new routes toward activity-assisted manipulation of colloids, potentially guiding the design of materials able to self-anneal their defects.

1:43PM R39.00002 Effective diffusivity in active Brownian suspensions¹, ERIC BURKHOLDER, JOHN BRADY, California Institute of Technology — We study the single-particle diffusion of a Brownian probe of size R in a suspension comprised of a Newtonian solvent and a dilute dispersion of active Brownian particles (ABPs) with size a , characteristic swim velocity U_0 , and a reorientation time τ_R . These ABPs, or swimmers, have a run length $\ell = U_0\tau_R$, and a mechanical activity $k_s T_s = \zeta_a U_0^2 \tau_R / 6$, where ζ_a is the Stokes drag coefficient of a swimmer. When the swimmers are inactive, collisions between the probe and the swimmers sterically hinder the probe's diffusive motion. When the activity of the swimmers is greater than the Boltzmann energy, $k_s T_s > k_B T$, rather than being sterically hindered, the probe diffusivity is actually greater than its Stokes-Einstein-Sutherland diffusivity due to the mechanical energy imparted to the probe upon collisions with the swimmers. The active contribution to the effective diffusivity is a non-monotonic function of the swimmers' run length compared to the contact length between the probe and a swimmer: $\ell/(R+a)$. Comparisons are made to previous theoretical and experimental investigations of the hydrodynamic diffusion of a colloidal particle in a dilute suspension of swimming bacteria.

¹NSF Grant no. CBET 1437570

1:56PM R39.00003 Shear bands in concentrated bacterial suspensions under oscillatory shear¹, XIANG CHENG, DEVRAJAN SAMANTA, University of Minnesota, Minneapolis, USA, XINLIANG XU, Beijing Computational Science Research Center, Beijing, China — Bacterial suspensions show interesting rheological behaviors such as a remarkable "superfluidic" state with vanishing viscosity. Although the bulk rheology of bacterial suspensions has been experimentally studied, shear profiles within bacterial suspensions have not been systematically explored so far. Here, by combining confocal rheometry with PIV, we investigated the flow behaviors of concentrated *E. coli* suspensions under planar oscillatory shear. We found that concentrated bacterial suspensions exhibit strong non-homogeneous flow profiles at low shear rates, where shear rates vanish away from the moving shear plate. We characterized the shape of the nonlinear shear profiles at different applied shear rates and bacterial concentrations and activities. The shear profiles follow a simple scaling relation with the applied shear rates and the enstrophy of suspensions, unexpected from the current hydrodynamic models of active fluids. We demonstrated that this scaling relation can be quantitatively understood by considering the power output of bacteria at different orientations with respect to shear flows. Our experiments reveal a profound influence of shear flows on the locomotion of bacteria and provide new insights into the dynamics of active fluids.

¹The research is funded by ACS Petroleum Research Fund (54168-DNI9) and by the David & Lucile Packard Foundation. X. X. acknowledges support by the National Natural Science Foundation of China No. 11575020.

2:09PM R39.00004 Interaction of localized convection cells in the bioconvection of *Euglena gracilis*¹, MAKOTO IIMA, TAKAYUKI YAMAGUCHI, Hiroshima University — *Euglena gracilis* is a unicellular flagellated photosynthetic alga. The suspension of *Euglena* has behavioral responses to light, which causes a macroscopic localized bioconvection pattern when illuminated from below. One of the fundamental structures of this is a pair of convection cells, and high cell density region exists in the middle of the pair[1]. Experimental studies show various types of interaction in the localized convection cells; bound state, collision, etc. We performed numerical simulation of a hydrodynamic model of this system, and show results of the interactions. Long-range interaction due to the conservation of cell number and merging process of two localized structures will be discussed. [1] "Localized Bioconvection Patterns and Their Initial State Dependency in *Euglena gracilis* Suspensions in an Annular Container", E. Shoji, et al. J. Phys. Soc. Jpn. 83, 043001 (2014)

¹KAKENHI

2:22PM R39.00005 Vortex lattices and defect-mediated viscosity reduction in active liquids, JONASZ SLOMKA, JORN DUNKEL, Massachusetts Institute of Technology — Generic pattern-formation and viscosity-reduction mechanisms in active fluids are investigated using a generalized Navier-Stokes model that captures the experimentally observed bulk vortex dynamics in microbial suspensions. We present exact analytical solutions including stress-free vortex lattices and introduce a computational framework that allows the efficient treatment of previously intractable higher-order shear boundary conditions. Large-scale parameter scans identify the conditions for spontaneous flow symmetry breaking, defect-mediated low-viscosity phases and negative-viscosity states amenable to energy harvesting in confined suspensions. The theory uses only generic assumptions about the symmetries and long-wavelength structure of active stress tensors, suggesting that inviscid phases may be achievable in a broad class of non-equilibrium fluids by tuning confinement geometry and pattern scale selection.

2:35PM R39.00006 Flagellum synchronization inhibits large-scale hydrodynamic instabilities in sperm suspensions, SIMON F. SCHÖLLER¹, ERIC E. KEAVENY, Imperial College London — Sperm in suspension can exhibit large-scale collective motion and form coherent structures. Our picture of such coherent motion is largely based on reduced models that treat the swimmers as self-locomoting rigid bodies that interact via steady dipolar flow fields. Swimming sperm, however, have many more degrees of freedom due to elasticity, have a more exotic shape, and generate spatially-complex, time-dependent flow fields. While these complexities are known to lead to phenomena such as flagellum synchronization and attraction, how these effects impact the overall suspension behaviour and coherent structure formation is largely unknown. Using a computational model that captures both flagellum beating and elasticity, we simulate suspensions on the order of 10^3 individual swimming sperm cells whose motion is coupled through the surrounding Stokesian fluid. We find that the tendency for flagella to synchronize and sperm to aggregate inhibits the emergence of the large-scale hydrodynamic instabilities often associated with active suspensions. However, when synchronization is repressed by adding noise in the flagellum actuation mechanism, the picture changes and the structures that resemble large-scale vortices appear to re-emerge.

¹Supported by an Imperial College PhD scholarship.

2:48PM R39.00007 Self-Driven Droplet Powered By Active Nematics , TONG GAO, Michigan

State University, ZHAORUI LI, Texas AM University-Corpus Christi, MICHAEL SHELLEY, Courant Institute of Mathematical Sciences — Active matter defines a class of emerging bio-inspired materials composed of self-driven micro-particles and far away from equilibrium. Their anomalous physical properties and the means to control them, suggest novel methods in mixing/separation, micro-pumps and motors, self-healing materials etc. The possibility of realizing these applications hinges on a thorough understanding of the physical mechanisms as well as developing means to manipulate various active systems. By using of a coarse-grained active liquid crystal model, we design and investigate self-driven droplets encapsulating a dense suspension of active particles. We show that a single droplet can be set into motion due to the internal collective motions that are featured by active flows and motile disclination defects. We illustrate that the interplays between the induced directional flows, liquid crystalline structures, and the deformable interface with surface tension can result in tunable mobilities of motile droplets that undergo novel locomotion and rotation.

3:01PM R39.00008 Theory of meiotic spindle assembly , SEBASTIAN FURTHAUER, Center for

Computational Biology, Simons Foundation, PETER FOSTER, DANIEL NEEDLEMAN, SEAS, Harvard University, MICHAEL SHELLEY, New York Univ NYU & Simons Foundation — The meiotic spindle is a biological structure that self assembles from the intracellular medium to separate chromosomes during meiosis. It consists of filamentous microtubule (MT) proteins that interact through the fluid in which they are suspended and via the associated molecules that orchestrate their behavior. We aim to understand how the interplay between fluid medium, MTs, and regulatory proteins allows this material to self-organize into the spindle's highly stereotyped shape. To this end we develop a continuum model that treats the spindle as an active liquid crystal with MT turnover. In this active material, molecular motors, such as dyneins which collect MT minus ends and kinesins which slide MTs past each other, generate active fluid and material stresses. Moreover nucleator proteins that are advected with and transported along MTs control the nucleation and depolymerization of MTs. This theory captures the growth process of meiotic spindles, their shapes, and the essential features of many perturbation experiments. It thus provides a framework to think about the physics of this complex biological suspension.

3:14PM R39.00009 Transient aggregation and long-time diffusion of bacterial suspensions in time periodic flows , BOYANG QIN, University of Pennsylvania, REBECCA WINTER, Drexel University,

MADHURA GURJAR, DAVID GAGNON, ALISON PATTESON, PAULO ARRATIA, University of Pennsylvania — In this talk, the transport dynamics of swimming bacteria in time-periodic flows is investigated in experiments and simulations. Experiments are performed by introducing swimming bacteria (*Vibrio cholerae*) in a low Reynolds number, two-dimensional flow driven electromagnetically. We observe two distinct transport regimes: (i) entrapment of bacteria inside vortex and near elliptic points and (ii) aggregation and subsequent transport along the flow manifolds. These time-dependent behaviors are set by the interaction between swimmer kinematics (e.g. speed, tumbling frequency, etc) and flow properties. Numerical simulation using a stochastic Langevin model are able to capture the main experimental results including the entrapment of bacteria near elliptic points and the rapid spreading along manifolds. Results show a significant reduction in long-time effective diffusion of the swimmer as vortex strength is increased. The conditions for bacterial entrapment in vortex flows are discussed.