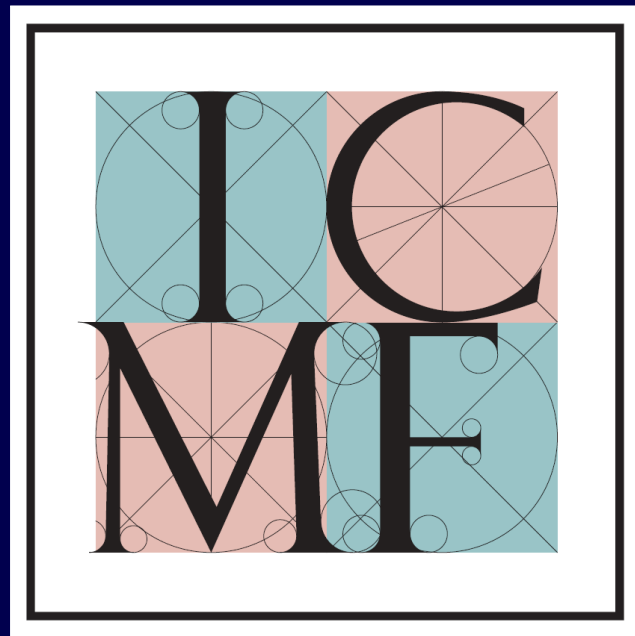


9th International Conference on  
Multiphase Flow

May 22-27, Firenze, Italy



2016



Book of Abstracts



*ICMF 2016  
the 9th International Conference on Multiphase Flow  
Firenze (Italy), May 22-27, 2016.*

*BOOK of ABSTRACT*



*Welcome to the 9th International Conference on Multiphase Flow (ICMF 2016) in Firenze (Italy), May 22-27, 2016.*

*Since 1991, ICMF is the largest international forum to present and discuss progress in research, development, standards, and applications of the topics related to multiphase flows. The conference brings together more than 1000 graduate students, postdoctoral researchers, university faculty, and researchers across government and industry to share the latest developments in the field.*

*The scientific programme of ICMF 2016 features more than 1000 oral and poster presentations, including 5 plenary lectures and 10 keynote lectures by outstanding researchers in the multiphase flow community. Special events also include the awarding of the ICMF Senior and Junior awards and vendor exhibits.*

*The conference is hosted by the University of Udine, which we gratefully acknowledge for providing funding and resources. The conference convenes at the Firenze Fiera Conference Center, located next to the wonderful Basilica di Santa Maria Novella, in the historical center of Firenze.*

*As we are all aware, the efforts required in organizing and holding this kind of Conference are extensive. The organizers would like to express their sincere appreciation to the organizing committee members for their exemplary efforts. They also want to acknowledge the productive co-operation with the congress management AIDIC Servizi. Sponsors and exhibitors are thankfully acknowledged. Finally, we acknowledge our gratitude to the ICMF 2016 invited speakers, authors, session chairpersons and attendees, whose contributions and efforts have made this conference a great success.*

*On Behalf of the Organizing Committee*

*Alfredo Soldati*

A handwritten signature in blue ink, appearing to be 'A. Soldati', with a stylized flourish at the end.

*ICMF2016 Chairman*





# CONTENT

## Paper Index

Monday	I
Tuesday	XIII
Wednesday	XXV
Thursday	XXXV
Friday	XLV
Posters	XLIX

Author Index	LIII
--------------	------

Abstracts	1
-----------	---



# Paper Index

**MONDAY, May 23, 2016**

## PARALLEL SESSIONS I

Mo 9:50-11:10	<b>Bio-Fluids</b>	
	<i>Room: Affari-2.B</i>	
	NUMERICAL ANALYSIS OF THE FLOW AND PARTICLE PATTERN IN THE HUMAN NASAL CAVITY	
	<i>Farnoud Ali, Cui Xinguang, Baumann Ingo, Gutheil Eva</i>	1/1
	THROMBUS/BUBBLES FORMATION AND MULTIPHASE FLOW IN BLOOD VESSEL AFTER LASER IRRADIATION	
Mo 9:50-11:10	<i>Wu Wenjuan, Li Dong, Chen Bin</i>	1/2
	NANOPARTICLE DEPOSITION ANALYSIS IN HUMAN AND RAT NASAL CAVITIES: A NUMERICAL COMPARISON STUDY	
	<i>Dong Jingliang, Shang Yidan, Inthavong Kiao, Tu Jiyuan, Chen Rui, Chen Chunying</i>	1/3
	<b>Boiling, Condensation, Evaporation</b>	
	<i>Room: Affari-3.D</i>	
Mo 9:50-11:10	EVAPORATION HEAT TRANSFER COEFFICIENT OF R410A INSIDE MICRO/MINI-CHANNEL	
	<i>Oh Jong-Taek, Chien Nguyen Ba, Choi Kwang-Il</i>	1/4
	NUMERICAL SIMULATIONS OF NUCLEATE BOILING	
	<i>Huber Gregory, Tanguy Sebastien, Colin Catherine</i>	1/5
	DIRECT NUMERICAL SIMULATION OF INTERACTING EVAPORATING DROPLETS	
Mo 9:50-11:10	<i>Dairay Thibault, Naso Aurore, Spelt Peter</i>	1/6
	RESEARCH ON MARANGONI CONDENSATION MODES OF ETHANOL-WATER MIXTURES ON A HORIZONTAL SURFACE	
	<i>Li Yong, Wang Jinshi, Chen Nana, Yan Junjie, Xia Kai</i>	2/1
	<b>Cavitation/Nucleation</b>	
	<i>Room: Affari-3.C</i>	
Mo 9:50-11:10	LES MODELING OF CAVITATION FLOW IN A DIESEL INJECTOR NOZZLE	
	<i>Chen Boxiong, Falgout Zachary, Linne Mark, Oevermann Michael</i>	2/2
	CAVITATION MODELLING IN MICRO CHANNELS	
	<i>Kayakol Nuray</i>	2/3
	SIMULATION OF A HIGH PRESSURE FUEL PUMP, INCLUDING CAVITATION EFFECTS, WITH TWO MULTIPHASE MODELS	
Mo 9:50-11:10	<i>Koukouvini Foivos C., Karathanassis Ioannis, Li Jason, Wang Lifeng, Gavaises Manolis</i>	2/4
	NUMERICAL SIMULATION OF VAPOR-GAS BUBBLE DYNAMICS BASED ON MOLECULAR GAS DYNAMICS	
	<i>Nagayama Takahiro, Kon Misaki, Kobayashi Kazumichi, Watanabe Masao, Fujii Hiroyuki, Takahira Hiroyuki</i>	2/5
	<b>Collision, Agglomeration and Breakup</b>	
	<i>Room: Affari-3.A</i>	
Mo 9:50-11:10	TERNARY DROP COLLISIONS	
	<i>Planchette Carole, Hinterbichler Hannes, Brenn Guenter</i>	2/6
	ANALYSIS OF THE MECHANISMS FOR GAS BUBBLE AND DROPLET BREAKUP IN TURBULENT FLOWS	
	<i>Andersson Ronnie</i>	3/1
	NOVEL PHENOMENOLOGICAL MODELS FOR VISCOUS BINARY DROPLET COLLISIONS	
Mo 9:50-11:10	<i>Finotello Giulia, Deen Niels G., Padding Johan, Kuipers J. A. M (hans), Jongsma Alfred, Innings Fredrik</i>	3/2
	MULTI-SCALE MODELING OF BUBBLE COALESCENCE AND BREAKUP IN HETEROGENEOUS BUBBLY FLOWS	
	<i>Battistella Alessandro, Roghair Ivo, Van Sint Annaland Martin</i>	3/3
	<b>Computational Techniques for Multiphase Flows</b>	
	<i>Room: Affari-1.A</i>	
Mo 9:50-11:10	GENERALIZED IMMERSED BOUNDARY METHOD FOR UNSTRUCTURED MESH	
	<i>Abdol Azis Mohd Hazmil Syahidy, Van Wachem Berend</i>	3/4
	A NEW APPROACH TO DEFINE A NON-ITERATIVE IMMERSED BOUNDARY METHOD FOR SPHERICAL PARTICLES OF ARBITRARY DENSITY RATIO	
	<i>Tschisgale Silvio, Kempe Tobias, Frohlich Jochen</i>	3/5

	AN IMMersed BOUNDARY METHOD FOR COMPLEX-SHAPED BUBBLES REPRESENTED BY SPHERICAL HARMONIC FUNCTIONS <i>Krull Benjamin, Tschisgale Silvio, Frohlich Jochen</i>	3/6
	IMMersed BOUNDARY – THERMAL LATTICE BOLTZMANN METHOD FOR PARTICLE SEDIMENTATION IN NON-NEWTONIAN LIQUID <i>Amiri Delouei Amin, Mohsen Nazari, Mohammad Hasan Kayhani, Ahmadi Goodarz</i>	4/1
<b>Mo 9:50-11:10</b>	<b>Environmental and Geophysical Flows</b> <b>Room: Affari-3.E</b>	
	TANGENTIAL INLET SUPERSONIC SEPARATORS: A NOVEL APPARATUS FOR GAS PURIFICATION <i>Wen Chuang, Yang Yan, Walther Jens</i>	4/2
	CONTRAIL MODELING AND SIMULATION <i>Paoli Roberto</i>	4/3
	ESCOSED: OBSERVATIONS OF MORPHODYNAMICS DURING BORA AT THE MOUTH OF THE MISA RIVER <i>Calantoni Joseph, Sheremet Alexandru, Brocchini Maurizio, Postacchini Matteo</i>	4/4
	THE EFFECT OF BED GEOMETRY AND COHERENT VORTICES ON INCIPIENT GRANULAR BED MOTION IN OSCILLATORY FLOW: LABORATORY MEASUREMENTS AND NUMERICAL SIMULATIONS <i>Simeonov Julian, Frank Donya, Calantoni Joseph</i>	4/5
<b>Mo 9:50-11:10</b>	<b>Experimental Methods for Multiphase Flows</b> <b>Room: Affari-4.A</b>	
	STUDY OF A STATIONARY TAYLOR BUBBLE USING THE BRIGHTNESS BASED LASER INDUCED FLUORESCENCE TECHNIQUE (BBLIF) <i>Vasques Joao, Cherdantsev Andrey, Hann David, Hewakandamby Buddhikha, Azzopardi Barry James</i>	4/6
	ACOUSTICAL CHARACTERISTICS OF TWO-PHASE HORIZONTAL INTERMITTENT FLOW THROUGH AN ORIFICE <i>Lee Joohwa Sarah, Violato Daniele, Polifke Wolfgang</i>	5/1
	SPATIAL- AND TIME-RESOLVED MEASUREMENTS OF LIQUID HOLD-UP DISTRIBUTION IN A PACKED BED USING ELECTRICAL RESISTANCE TOMOGRAPHY AND VOIDAGE PROBES <i>Singh Brajesh Kumar, Jain Ekta, Buwa Vivek</i>	5/2
	SIMULTANEOUS BRIGHTNESS-BASED AND PLANAR LASER-INDUCED FLUORESCENCE MEASUREMENT OF DOWNWARDS ANNULAR FLOW <i>An Jae Sik, Cherdantsev Andrey, Zadrazil Ivan, Markides Christos N.</i>	5/3
<b>Mo 9:50-11:10</b>	<b>Industrial Applications</b> <b>Room: Affari-0.A</b>	
	MECHANISTIC SIMULATION OF VERTICAL SLUG FLOW WITH THE ONE-DIMENSIONAL TWO-FLUID MODEL <i>Galleni Francesco, Issa Raad</i>	5/4
	VISCOUS OIL-WATER HORIZONTAL WAVY STRATIFIED FLOW: CFD AND EXPERIMENTAL DATA COMPARISON <i>Avila Ricardo, Arroxellas Andre, Souza De Castro Marcelo, Hernandez Rodriguez Oscar Mauricio</i>	5/5
	COMPARISON OF NUMERICAL METHODS FOR SLUG CAPTURING WITH THE TWO-FLUID MODEL <i>Hendrix Maurice, Eskerud Smith Ivar, Van Zwieten Joost, Sanderse Benjamin</i>	5/6
	EFFECTS OF SURFACE ROUGHNESS AND INTERPARTICLE COLLISIONS ON ELBOW EROSION <i>Dos Santos Vinicius Vagundes, De Souza Francisco Jose, Ribeiro Duarte Carlos Antonio</i>	6/1
<b>Mo 9:50-11:10</b>	<b>Interfacial Flows</b> <b>Room: Affari-2.A</b>	
	A HYBRID NUMERICAL METHOD FOR TWO-PHASE FLOW WITH SOLUBLE SURFACTANT <i>Booty Michael, Siegel Michael</i>	6/2
	LIQUID FILM FLOW WITH SOLUBLE SURFACTANTS: THEORY AND EXPERIMENT <i>Georgantaki Anna, Karapetsas George, Vlachogiannis Michalis, Bontozoglou Vasileios</i>	6/3
	ON THE INFLUENCE OF SURFACTANTS ON THE WAKE AND SHAPE OF GASEOUS BUBBLES ASCENDING IN LIQUIDS <i>Schranner Felix, Heinz Johannes, Adams Nikolaus</i>	6/4
	FINITE VOLUME / FINITE AREA INTERFACE-TRACKING METHOD FOR SOLUBLE SURFACTANTS <i>Marshall Holger, Pesci Chiara, Weber Paul S., Bothe Dieter</i>	6/5

<b>Mo 9:50-11:10</b>	<b>Multiphase Flow in Heat and Mass Transfer</b> <i>Room: Affari-2.C</i> A NEW SUBMODEL FOR MOVING PARTICLES UNDERGOING PHASE CHANGE SOLID-LIQUID-GAS <i>Bansal Hemant, Nikrityuk Petr</i> 6/6 DIRECT NUMERICAL SIMULATION (DNS) OF FLUID-SOLID MASS TRANSFER USING A GHOST-CELL BASED IMMERSED BOUNDARY METHOD <i>Lu Jiangtao, Peters E.A.J.F., Kuipers J. A. M (hans)</i> 7/1 DIRECT NUMERICAL SIMULATION OF HEAT TRANSFER IN DENSE PARTICLE-LIQUID TWO-PHASE MEDIA <i>Gu Jingchen, Kondo Katsuya, Takeuchi Shintaro, Kajishima Takeo</i> 7/2 DETAILED NUMERICAL SIMULATIONS OF RESOLVED MULTI-COMPONENT FUEL DROPLETS EVAPORATING IN HOMOGENEOUS ISOTROPIC TURBULENCE <i>Palmore John, Desjardins Olivier</i> 7/3
<b>Mo 9:50-11:10</b>	<b>Particle and Drop Dynamics</b> <i>Room: Villa-Sala Verde</i> OBLIQUE IMPACT OF DROPLETS ON FLOWING LIQUID FILMS <i>Che Zhizhao, Ismail Renad, Adebayo Idris, Matar Omar K.</i> 7/4 DROP IMPACT ON SOLIDS <i>Josserand Christophe</i> 7/5 PHENOMENA OF DROP IMPACT ON HOT SURFACES <i>Breitenbach Jan, Roisman Ilia V., Tropea Cameron</i> 7/6 SPLASH OF A LIQUID DROP IMPACTING ONTO A WALL FILM OF DIFFERENT LIQUID <i>Kittel Hannah, Roisman Ilia V., Tropea Cameron</i> 8/1
<b>Mo 9:50-11:10</b>	<b>Particle-Laden Flows</b> <i>Room: Villa-Auditorium</i> TURBULENCE MODULATION IN CONCENTRATED, HIGH REYNOLDS NUMBER SOLID-LIQUID FLOWS <i>Breakey David, Agrawal Yogesh, Amini Parsa, Ghaemi Sina, Nobes David, Sanders Sean</i> 8/2 ROLE OF PREFERENTIAL CONCENTRATION IN THE DESTABILIZATION OF SHEARED PARTICLE-LADEN FLOWS <i>Kasbaoui Mohamed Housseem, Koch Donald, Desjardins Olivier</i> 8/3 PARTICLE CONCENTRATIONS IN RIB-ROUGHENED WALL TURBULENCE <i>Zhao Lihao, Andersson Helge</i> 8/4 SIMULATION OF TURBULENT FLOW AND PARTICLE TRANSPORT AND DEPOSITION USING LATTICE BOLTZMANN METHOD AND RANS MODEL <i>Sajjadi Hasan, Salmanzadeh Mazyar, Ahmadi Goodarz, Jafari Saeed</i> 8/5
<b>Mo 9:50-11:10</b>	<b>Multiphase Pipe Flows</b> <i>Room: Affari-3.B</i> A SLUG CAPTURING METHOD APPLIED TO UNCONVENTIONAL SCENARIOS: HIGH VISCOUS OILS AND ACTUAL PIPELINE GEOMETRIES <i>Ferrari Marco, Bonzanini Arianna, Poesio Pietro</i> 8/6 A MECHANISTIC APPROACH FOR HORIZONTAL SLUG FLOW FROM TWO-PHASE (GAS-LIQUID) TO THREE-PHASE WITH THE FORMATION OF GAS HYDRATES <i>Bassani Carlos, Barbuto Fausto, Sum Amadeu, Morales Rigoberto</i> 9/1 AN EXPERIMENTAL INVESTIGATION ON THE EVOLUTION OF HIGHLY VISCOUS OIL SLUG LENGTHS IN HORIZONTAL PIPES <i>Losi Gianluca, Poesio Pietro</i> 9/2 LINEAR STABILITY ANALYSES OF DISCRETE REPRESENTATIONS OF THE TWO-FLUID MODEL FOR STRATIFIED PIPE FLOW <i>Akselsen Andreas, Nydal Ole Jørgen</i> 9/3
<b>Mo 9:50-11:10</b>	<b>Bubble Dynamics</b> <i>Room: Affari-2.D</i> EXPERIMENTAL INVESTIGATION OF THE FLOW STRUCTURE AROUND TAYLOR BUBBLES IN VERTICAL DUCTS IN THE PRESENCE OF DISPERSED BUBBLES <i>Paladino Emilio, Cerqueira Rafael, Brito Ronei, Maliska Clovis, Meneghini Cristiano</i> 9/4 NUMERICAL STUDY OF TAYLOR BUBBLES RISING THROUGH SUDDEN EXPANSIONS AND CONTRACTIONS <i>Zhou Chengsi, Knikker Ronnie, Revellin Remi</i> 9/5 TAYLOR BUBBLE DESTRUCTION AT SUDDEN EXPANSIONS <i>Carter William, Agunlejika Ezekiel, Azzopardi Barry James</i> 9/6 INTERACTION OF A BUBBLE RISING IN A THIN-GAP CELL WITH THE VORTICES RELEASED BY A COMPANION BUBBLE <i>Ern Patricia, Filella Audrey, Roig Veronique</i> 10/1

## PARALLEL SESSIONS II

Mo 11:30-12:50	<b>Bubbly Flows</b>	
	<i>Room: Affari-2.B</i>	
	DIRECT NUMERICAL SIMULATION OF A TURBULENT PIPE FLOW WITH DEFORMABLE BUBBLES TO INVESTIGATE THE DYNAMICS OF BUBBLES	
	<i>Kim Kiyoun, Kim Jungwoo, Choi Haecheon</i>	10/2
	NUMERICAL SIMULATION OF A SINGLE RISING BUBBLE BY A VOF METHOD WITH ENHANCED CURVATURE ESTIMATION BASED ON POINT-CLOUD SAMPLING	
Mo 11:30-12:50	<i>Kassar Bruno, Carneiro Joao, Nieckele Angela</i>	10/3
	NUMERICAL SIMULATION OF UNDERWATER EXPLOSION WITH GHOST FLUID METHOD	
	<i>Chen Hong, Zhu Weibing, Khoo Boo Cheong</i>	10/4
	EFFECT OF BUBBLE CLUSTERS ON TURBULENT FLOW STRUCTURES IN UPWARD BUBBLY FLOW	
	<i>Zhang Wenhao, Sekiguchi Yoshito, Nakanishi Hiroaki, Sakakibara Jun, Takagi Shu</i>	10/5
Mo 11:30-12:50	<b>Boiling, Condensation, Evaporation</b>	
	<i>Room: Affari-3.D</i>	
	NUMERICAL SIMULATION OF MICROLAYER FORMATION DURING THE INERTIA-CONTROLLED GROWTH PHASE OF BUBBLES IN NUCLEATE BOILING	
	<i>Guion Alexandre, Buongiorno Jacopo, Zaleski Stephane, Afkhami Shahriar</i>	10/6
	PREDICTION OF THE BUBBLE DEPARTURE IN POOL AND FORCED CONVECTION BOILING	
Mo 11:30-12:50	<i>Considering the sublayer: A sub model of CFD approaches</i>	
	<i>Ding Wei, Krepper Eckhard, Hampel Uwe</i>	11/1
	PRELIMINARY RESULTS OF SOBER-SJ10 ON SINGLE BUBBLE POOL BOILING HEAT TRANSFER IN MICROGRAVITY	
	<i>Wu Ke, Zhao Jianfu, Li Hui-Xiong, Li Kai</i>	11/2
Mo 11:30-12:50	<b>Cavitation/Nucleation</b>	
	<i>Room: Affari-3.C</i>	
	NUMERICAL ANALYSIS OF BUBBLE DYNAMICS WITH VAPOR-LIQUID BOUNDARY CONDITIONS DERIVED FROM MOLECULAR GAS DYNAMICS	
	<i>Morikawa Shotaro, Jinbo Yoshinori, Tatekura Yuki, Kobayashi Kazumichi, Watanabe Masao, Fujii Hiroyuki, Sasaki Koichi</i>	11/3
	TESTING A BUBBLE DYNAMIC MODEL FOR BULK CAVITATION IN UNDERWATER EXPLOSION SIMULATIONS	
Mo 11:30-12:50	<i>Wedberg Rasmus, Alin Niklas</i>	11/4
	MODELING CAVITATION BUBBLES IN A FOCUSED ULTRASOUND FIELD	
	<i>Maeda Kazuki, Colonius Tim, Kreider Wayne, Maxwell Adam</i>	11/5
	CAVITATION CONTROL BY ACTIVE MASS INJECTION THROUGH A SLOT CHANNEL IN THE SURFACE OF A 2D HYDROFOIL	
	<i>Timoshevskiy Mikhail, Pervunin Konstantin, Markovich Dmitriy</i>	11/6
Mo 11:30-12:50	<b>Collision, Agglomeration and Breakup</b>	
	<i>Room: Affari-3.A</i>	
	A NEW EXPERIMENTAL DATASET TO VALIDATE CFD MODELS OF AIRBORNE NANOPARTICLES AGGLOMERATION	
	<i>Belut Emmanuel, Christophe Theo</i>	12/1
	BUBBLE DYNAMICS IN PLUG FLOW REGIME FOR HORIZONTAL GAS-LIQUID FLOW	
Mo 11:30-12:50	<i>Banerjee Jyotirmay, Thaker Jignesh</i>	12/2
	HYDRATES FORMATION, AGGREGATION AND SLURRY FLOW PROPERTIES IN DIFFERENT MULTIPHASE FLOW PATTERNS	
	<i>Ding Lin, Shi Bohui, Lv Xiaofang, Liu Yang, Wu Haihao, Wang Wei, Gong Jing</i>	12/3
	PREFERENTIAL CONCENTRATION IN COHERENT FLOW STRUCTURES: FOCUS ON PARTICLE-PARTICLE INTERACTIONS	
	<i>Kosinski Pawel, Balakin Boris V., Kartushinsky Alexander</i>	12/4
Mo 11:30-12:50	<b>Computational Techniques for Multiphase Flows</b>	
	<i>Room: Affari-1.A</i>	
	PERFORMANCE ASSESSMENT OF EULERIAN VERSUS EULERIAN-LAGRANGIAN LEVEL SET IMPLEMENTATIONS FOR THE SIMULATION OF TWO PHASE FLOWS	
	<i>Lorieul Gael, Chatelain Philippe, Bartosiewicz Yann</i>	12/5
	DEVELOPMENT OF A FULLY COUPLED SOLVER FOR THE EULERIAN-EULERIAN APPROACH	
Mo 11:30-12:50	<i>Bartholomew Paul, Marquis Andrew, Denner Fabian, Van Wachem Berend</i>	12/6

	ON THE INFLUENCE OF THE NUMERICAL STRATEGY ON THE PREDICTIVE CHARACTER OF AN EULER-EULER MODEL FOR TWO-PHASE FLOW SIMULATIONS IN SOLID ROCKET MOTOR INSTABILITIES <i>Dupif Valentin, Massot Marc, Dupays Joel, Laurent Frederique, Le Touze Clement</i>	13/1
	A THERMODYNAMICALLY CONSISTENT PHASE-FIELD MODEL FOR TWO-PHASE FLOWS AND ITS ENERGY-LAW PRESERVING COMPUTATIONAL METHOD <i>Lin Ping</i>	13/2
<b>Mo 11:30-12:50</b>	<b>Environmental and Geophysical Flows</b> <b>Room: Affari-3.E</b> NUMERICAL PREDICTION OF SEDIMENT DEPOSITION AT A RIVER CONFLUENCE USING AN EULER-LAGRANGE METHOD <i>Hamidouche Souria, Arcen Boris, Taniere Anne</i>	13/3
	A NUMERICAL INVESTIGATION OF PARTICLE-BED-TURBULENCE INTERACTIONS IN OSCILLATORY FLOWS <i>Ghodke Chaitanya, Apte Sourabh</i>	13/4
	COMPUTATIONAL MODELLING OF PROPPANT TRANSPORT IN FRACTURES <i>Ahmadi Goodarz, Abdollahi Mofakham Amir, Crandall Dustin, Bromhal Grant</i>	13/5
	MODELING OF CORIUM SPREADING IN A CORE CATCHER <i>Perrier Hugo</i>	13/6
<b>Mo 11:30-12:50</b>	<b>Experimental Methods for Multiphase Flows</b> <b>Room: Affari-4.A</b> PHASE RETRIEVAL METHOD FOR HOLOGRAPHIC PARTICLE MEASUREMENT WITH TWO CAMERAS <i>Tanaka Yohsuke, Murata Shigeru, Tani Shunsuke</i>	14/1
	SIMULTANEOUS ANALYSIS OF ELECTROCHEMICAL IMPEDANCE SPECTROSCOPY AND VISUALIZATION TECHNIQUE TO EXPLORE TWO-PHASE FLOW BEHAVIOUR IN THE PROTON EXCHANGE MEMBRANE FUEL CELLS (PEMFCs) <i>Kanani Homayoon, Mehrzad Shams</i>	14/2
	APPLICATION OF AN ARRAY OF FOUR PHASE-DETECTION PROBE SENSORS TO HYDRAULIC JUMPS AT LARGE REYNOLDS NUMBERS <i>Wang Hang, Chanson Hubert</i>	14/3
	MULTIPHASE FLOW QUANTIFICATION USING MAGNETIC RESONANCE IMAGING AND COMPUTATIONAL FLUID DYNAMICS <i>Sederman Andrew, Maru Wes, Lakshmanan Susithra, Holland Daniel</i>	14/4
<b>Mo 11:30-12:50</b>	<b>Industrial Applications</b> <b>Room: Affari-0.A</b> A NUMERICAL STRATEGY TO ACCOUNT FOR THE EFFECT OF SELF-INDUCED GEOMETRY CHANGES IN WEAR ESTIMATION <i>Messa Gianandrea Vittorio, Malavasi Stefano</i>	14/5
	COMPARISON OF TWO METHODS CALCULATING PARTICLE IMPACT ON A FLAT PLATE IN ORDER TO DEDUCT HYDRO-ABRASIVE EROSION <i>Boden Wiebke, Aubert Stephane, Marongiu Jean-Christophe, Perkins Richard</i>	14/6
	FUEL SPRAY SIMULATION DURING OPENING VALVE OF FUEL INJECTOR WITH PARTICLE METHOD <i>Ishii Eiji, Yoshimura Kazuki, Yasukawa Yoshihito, Ehara Hideharu</i>	15/1
	USING CENTRIFUGAL FORCES IN A PITOT-TUBE JET-PUMP FOR CLEANING AND PUMPING OIL-WATER MIXTURES IN A CONTINUOUS LOOP: AN EXPERIMENTAL STUDY OF SEPARATION EFFICIENCY <i>Meyer Jan, Thevenin Dominique</i>	15/2
<b>Mo 11:30-12:50</b>	<b>Interfacial Flows</b> <b>Room: Affari-2.A</b> SURFACTANT INDUCED LATERAL MIGRATION OF BUBBLES AND DROPS IN A SHEAR FLOW <i>Muradoglu Metin, Tryggvason Gretar</i>	15/3
	COMPUTATIONAL STUDY OF THE ROLE OF SURFACTANTS IN SHEARED FOAMS FOR FOAM STABILITY <i>Titta Andrea, Detcheverry Francois, Le Merrer Marie, Biance Anne-Laure, Spelt Peter</i>	15/4
	STUDY OF THREE-DIMENSIONAL STRUCTURE AND COALESCENCE OF DISTURBANCE WAVES ON GAS-SHEARED LIQUID FILM IN A HORIZONTAL RECTANGULAR DUCT <i>Cherdantsev Andrey, McCarthy Ian, Hann David, Hewakandamby Buddhikha, Azzopardi Barry James</i>	15/5
	LIQUID-LIQUID ENTRAINMENT AND FILM WRAPPING AROUND A ROTATING CYLINDER IN STRATIFIED LAYER <i>Rana Basanta Kumar, Sharma Yagya, Das Arup Kumar, Das Prasanta Kumar</i>	15/6



<b>Mo 11:30-12:50</b>	<b>Multiphase Flow in Heat and Mass Transfer</b>	
	<i>Room: Affari-2.C</i>	
	MULTISCALE MODELLING OF HEAT TRANSFER FROM ARRAYS OF SPHERICAL PARTICLES	16/1
	<i>Singhal Arpit, Cloete Schalk, Radl Stefan, Amini Shahriar</i>	
	3D NUMERICAL SIMULATION OF UPFLOW BUBBLING FLUIDIZED BED IN OPAQUE TUBE UNDER HIGH FLUX SOLAR HEATING	
	<i>Benoit Hadrien, Ansart Renaud, Trinanes Pablo Garcia, Gauthier Daniel, Flamant Gilles, Simonin Olivier</i>	16/2
	EULERIAN AND FULL LAGRANGIAN MODELING OF A FLOW AND HEAT TRANSFER IN A GAS-DROPLET SWIRLING FLOW IN A PIPE SUDDEN EXPANSION	
	<i>Pakhomov Maksim, Terekhov Viktor</i>	16/3
	LARGE EDDY & INTERFACE SIMULATION (LEIS) OF DISTURBANCE WAVES AND HEAT TRANSFER IN ANNULAR FLOWS	
	<i>Yang Junfeng, Hewitt Geoffrey, Narayanan Chidambaram, Lakehal Djamel</i>	16/4
<b>Mo 11:30-12:50</b>	<b>Particle and Drop Dynamics</b>	
	<i>Room: Villa-Sala Verde</i>	
	OBSERVATION OF NON-SPHERICAL DROPLET IMPACTS WITH VELOCITIES OF O(10 M/S)	
	<i>Kataoka Yutaka, Hori Yuusaku, Watanabe Masao, Kobayashi Kazumichi, Fujii Hiroyuki, Sanada Toshiyuki</i>	16/5
	EFFECTS OF SURROUNDING GAS TEMPERATURE ON ETHANOL DROPLET IMPACT ON SOLID SURFACE	
	<i>Isono Toshihisa, Watanabe Masao, Kobayashi Kazumichi, Fujii Hiroyuki, Sanada Toshiyuki</i>	16/6
	FREEZING DELAY AND NUCLEATION RATES OF SUPERCOOLED WATER DROPS IMPACTING ONTO A COLD SURFACE	
	<i>Schremb Markus, Roisman Ilia V., Jakirlic Suad, Tropea Cameron</i>	17/1
	DISCUSSION OF SURFACTANTS MIGRATION ON DYNAMIC INTERACTIONS BETWEEN DEFORMABLE DROPLET	
	<i>Jin Hang, Wang Wei, Liu Fangyuan, Ma Mengyu, Li Kai, Gong Jing</i>	17/2
<b>Mo 11:30-12:50</b>	<b>Particle-Laden Flows</b>	
	<i>Room: Villa-Auditorium</i>	
	DEPOSITION CHARACTERISTICS IN PARTICLE-LADEN TURBULENT FLOWS	
	<i>Nicolaou Laura, Zaki Tamer</i>	17/3
	FINE PARTICLE DEPOSITION IN DUCT FLOW WITH STAGGERED BAFFLES	
	<i>Wu Jiandong, Shen Hongwei, Liu Qiao, Wang Hao</i>	17/4
	LATTICE BOLTZMANN MODEL FOR PREDICTING THE DEPOSITION OF INERTIAL PARTICLES IN TURBULENT CHANNEL FLOWS	
	<i>Lepoutere Guillaume, Fede Pascal, Sofonea Victor, Fournier Richard, Blanco Stephane, Simonin Olivier</i>	17/5
	TENSORIALLY INVARIANT NORMALIZED LANGEVIN MODEL FOR PARTICLE DISPERSION AND DEPOSITION IN TURBULENT FLOWS	
	<i>Sikovskiy Dmitrii</i>	17/6
<b>Mo 11:30-12:50</b>	<b>Multiphase Pipe Flows</b>	
	<i>Room: Affari-3.B</i>	
	PHENOMENOLOGICAL STUDY OF PHASE INVERSION IN A LIQUID-LIQUID DISPERSION PIPE FLOW	
	<i>Idoughi Schehinez, Le Sauze Nathalie, Cid Emmanuel, Masbernat Olivier</i>	18/1
	WIRE-MESH-BASED TOMOGRAPHY IN UPWARD-VERTICAL OIL-WATER-AIR THREE-PHASE FLOW	
	<i>Velasco Pena Hugo Fernando, Bonilla Riano Adriana, Monteiro De Carvalho Carlos Henrique, Estevam Valdir, Hernandez Rodriguez Oscar Mauricio</i>	18/2
	AN EXPERIMENTAL STUDY OF UPWARD-VERTICAL OIL-WATER-GAS THREE-PHASE FLOW: FLOW PATTERN, PRESSURE DROP AND VOLUMETRIC PHASE FRACTION	
	<i>Rocha Douglas Martins, Monteiro De Carvalho Carlos Henrique, Estevam Valdir, Hernandez Rodriguez Oscar Mauricio</i>	18/3
	EXPERIMENTAL STUDY OF HEAVY OIL-WATER CORE-ANNULAR FLOW PATTERN	
	<i>Cavicchio Caio, Bannwart Antonio Carlos, Souza De Castro Marcelo, Biazussi Jorge Luiz</i>	18/4
<b>Mo 11:30-12:50</b>	<b>Bubble Dynamics</b>	
	<i>Room: Affari-2.D</i>	
	BUBBLE GROWTH AND AGGLOMERATION IN DRYING PAINT FILMS	
	<i>Saranjam Nazli, Chandra Sanjeev</i>	18/5
	MODEL OF BUBBLE RISE AND COLLISION WITH COMPLEX DEFORMABLE INTERFACES	
	<i>Chan Derek, Manica Rogerio, Klaseboer Evert</i>	18/6
	DYNAMICS OF BUBBLE GROWTH AND DETACHMENT IN A SHEAR FLOW	
	<i>Lebon Michel, Sebilliau Julien, Colin Catherine</i>	19/1

NUMERICAL SIMULATION OF BUBBLE FORMATION AND DYNAMIC CHARACTERISTICS AT A  
 SUBMERGED MICRONOZZLE

Qu Chen, Zhang Jian, Yu Yong, Hu Jun

19/2

## PARALLEL SESSIONS III

Mo 14:40-16:00

### Bubbly Flows

Room: Affari-2.B

EFFECT OF LIQUID AND COLUMN DIAMETER OF THE FLOW REGIME IN BUBBLE COLUMNS

Escrig Escrig Josep, Sharaf Safa, Hewakandamby Buddhikha, Azzopardi Barry James

19/3

HOMOGENEOUS FLOW IN A LARGE SCALE BUBBLE COLUMN: EXPERIMENTAL  
 INVESTIGATION AND CFD MODELLING

Besagni Giorgio, Ziegenhein Thomas, Lucas Dirk, Inzoli Fabio

19/4

HYDRODYNAMICS OF BUBBLE COLUMNS IN THE HETEROGENEOUS REGIME

Raimundo Pedro Maximiano, Cartellier Alain, Beneventi Davide, Forret Ann, Augier Frederic

19/5

TWO-PHASE TURBULENCE IN LAMINAR BUBBLY PIPE FLOWS

Park Hyungmin, Kim Minki, Lee Jun Ho

19/6

Mo 14:40-16:00

### Boiling, Condensation, Evaporation

Room: Affari-3.D

EXPERIMENTAL STUDY OF TRANSIENT FLOW BOILING HEAT TRANSFER UNDER  
 EXPONENTIALLY ESCALATING HEAT INPUTS

Su Guan-Yu, Bucci Matteo, Mckrell Thomas, Buongiorno Jacopo

20/1

VORTEX STRUCTURES OF LOW STEAM QUALITY BOILING TWO-PHASE FLOW IN BUBBLE-  
 TYPE PLATE CHANNEL

Jiang Chen, Yang Zonglin, Bai Bofeng

20/2

LAGRANGIAN TRACKING OF DROPLETS EVAPORATING IN A HI TURBULENCE BY USING IN-  
 LINE DIGITAL HOLOGRAPHY

Marie Jean-Louis, Tronchin Thibaut, Grosjean Nathalie, Mees Loic, Lance Michel

20/3

EXPERIMENTALLY STUDYING OF THE EFFECT OF THE HEATING SURFACE MORPHOLOGY

ON CHARACTERISTICS OF HIGHLY SUBCOOLED WATER BOILING

Vasiliev Nikolay, Khodakov Konstantin, Varaksin Aleksei, Zeigarnik Yuriy

20/4

Mo 14:40-16:00

### Cavitation/Nucleation

Room: Affari-3.C

CFD SIMULATION OF CAVITATION IN FUEL NOZZLES

Bretones Alvaro, De Souza Francisco Jose

20/5

EFFECTS OF MASS TRANSFER ON THE OSCILLATIONS OF CAVITATION BUBBLES

Zhang Yuning, Gao Yuhang, Du Xiaozhe

20/6

SYNCHROTRON X-RAY PHASE CONTRAST IMAGING OF CAVITATION IN FUEL INJECTOR  
 NOZZLES WITH VARIOUS SIZES

Sou Akira, Prasetya Rubby, Moon Seoksu, Wada Yoshitaka, Yokohata Hideaki

21/1

TWO-PHASE FLOW SIMULATIONS OF THE INTER-BLADE VORTICES IN A FRANCIS TURBINE

Wack Jonas, Riedelbauch Stefan, Yamamoto Keita, Avellan Francois

21/2

Mo 14:40-16:00

### Collision, Agglomeration and Breakup

Room: Affari-3.A

A MATHEMATICAL ANALYSIS OF PARTICLE CLUSTERING IN TURBULENT FLOWS

Esmaily Moghadam Mahdi, Mani Ali

21/3

RAPID STRETCHING OF NEWTONIAN AND NON-NEWTONIAN LIQUID BRIDGES

Roisman Ilia V., Weickgenannt Christina, Tropea Cameron

21/4

JOINT INVESTIGATION OF CLUSTERING AND SETTLING OF HEAVY SUB-KOLMOGOROV  
 PARTICLES IN TURBULENT FLOWS

Sumbekova Sholpan, Aliseda Alberto, Bourgoin Mickael, Cartellier Alain

21/5

CLUSTERING, INTERMITTENCY OF RELATIVE VELOCITY AND CAUSTICS OF HEAVY  
 PARTICLES UNDER GRAVITY IN ISOTROPIC TURBULENT FLOWS

Jin Guodong, Chen Jincai, He Guo-Wei

21/6

Mo 14:40-16:00

### Computational Techniques for Multiphase Flows

Room: Affari-1.A

NUMERICAL SIMULATION OF THREE-DIMENSIONAL MULTIPHASE FLOWS WITH ADAPTIVE  
 UNSTRUCTURED MESHES

Xie Zhizhua, Pavlidis Dimitrios, Che Zhizhao, Salinas Pablo, Pain Christopher, Matar Omar K.

22/1

DIFFUSE INTERFACE SIMULATIONS OF PHASE SEPARATION OF SHEARED BINARY  
 MIXTURES USING A FINITE VOLUME SOLVER

Mauri Roberto, Di Vitanonio Giuseppe, Galletti Chiara, Lamorgese Andrea

22/2

	DISCONTINUOUS GALERKIN FINITE ELEMENT METHOD APPLIED TO THE COUPLED NAVIER-STOKES/CAHN-HILLIARD EQUATIONS <i>Pigeonneau Franck, Saramito Pierre</i>	22/3
	FAST SIMULATIONS OF RECURRENT MULTIPHASE FLOWS' LONG-TIME BEHAVIOUR <i>Lichtenegger Thomas, Pirker Stefan, Holzinger Gerhard</i>	22/4
<b>Mo 14:40-16:00</b>	<b>Environmental and Geophysical Flows</b> <i>Room: Affari-3.E</i> FRAGMENTATION OF A POROUS VISCOELASTIC LIQUID BY RAPID DECOMPRESSION: IMPLICATION TO VOLCANIC ERUPTION <i>Kameda Masaharu, Ichihara Mie</i>	22/5
	INFLUENCE OF ANISOTROPIC PERMEABILITY ON CONVECTION IN POROUS MEDIA <i>De Paoli Marco, Zonta Francesco, Soldati Alfredo</i>	22/6
	SURFACING AND CLUSTERING OF GYROTACTIC SWIMMERS IN FREE-SURFACE TURBULENCE <i>Lovecchio Salvatore, Marchioli Cristian, Sardina Gaetano, Brandt Luca, Soldati Alfredo</i>	23/1
	CRUDE OIL PLUMES IN CROSSFLOW: EFFECT OF TURBULENT FLOW STRUCTURE ON OIL RESIDENCE IN PLUME <i>Murphy David, Xue Xinzhi, Sampath Kaushik, Katz Joseph</i>	23/2
<b>Mo 14:40-16:00</b>	<b>Experimental Methods for Multiphase Flows</b> <i>Room: Affari-4.A</i> LOCAL CONCENTRATION MEASUREMENTS IN THE WAKE OF BUBBLES BASED ON IN-SITU RAMAN SPECTROSCOPY AND STATISTICAL ANALYSIS <i>Schurr Daniela, Guhathakurta Jainabalkya, Baroud Yousef, Simon Sven, Rinke Gunter, Dittmeyer Roland</i>	23/3
	CONDITIONAL SAMPLING FOR THE ANALYSIS OF TWO-PHASE FLOW PIV/PTV EXPERIMENTAL IMAGES <i>Estrada Perez Carlos E., Yoo Junsoo, Hassan Yassin A.</i>	23/4
	THE ROLE OF INLET FLOW INSTABILITIES IN LIQUID-LIQUID FLOW <i>Morgan Rhys, Markides Christos N., Zadrazil Ivan</i>	23/5
	DEVELOPMENT OF MEASUREMENT TECHNIQUE FOR BUBBLE VELOCITY VECTOR IN GAS-LIQUID TWO-PHASE FLOW BY USING 4-SENSOR PROBE <i>Saito Yasushi, Ariyoshi Gen, Ito Daisuke</i>	23/6
<b>Mo 14:40-16:00</b>	<b>Industrial Applications</b> <i>Room: Affari-0.A</i> A LARGE SCALE MULTI-FLUID/DISPERSED PHASE APPROACH FOR SPRAY GENERATION IN AERONAUTICAL FUEL INJECTORS <i>Zuzio Davide, Estivalezes Jean-Luc, Blanchard Ghislain</i>	24/1
	SIMULATING THE INTERACTION OF SPRAY AND SURFACE FILM FLOW WITH COMPLEX GEOMETRIES IN FIRE SUPPRESSION APPLICATIONS <i>Meredith Karl V. M., Jamison Kristin, De Vries Jaap, Zhou Xiangyang, Wang Yi</i>	24/2
	NOZZLE TIP WETTING OF MULTI-HOLE GASOLINE INJECTORS WITH VARIOUS NOZZLE OUTLET CONFIGURATIONS UNDER FLASHING AND NON-FLASHING CONDITIONS <i>Park Chansoo, Jung Jinyoung, Bae Choongsik, Oh Heechang, Lee Juhun</i>	24/3
	PRESSURE SWIRL NOZZLES WITH A CONVEX DEFLECTION BODY TO INCREASE THE SPRAY ANGLE <i>Kamplade Jens, Hohlfeld Ivette, Kelz Marcus, Thommes Markus, Walzel Peter</i>	24/4
<b>Mo 14:40-16:00</b>	<b>Interfacial Flows</b> <i>Room: Affari-2.A</i> INTERFACE TREATING TECHNIQUE FOR COMPRESSIBLE MULTI-MEDIUM FLOWS <i>Khoo Boo Cheong, Chen Hong, Zhu Weibing</i>	24/5
	INTERFACIAL PERTURBATIONS IN TWO-PHASE OIL-WATER PIPE FLOWS INDUCED BY A CYLINDER <i>Park Kyeong Hyeon, Chinaud Maxime, Angeli Panagiota</i>	24/6
	HYBRID INTERFACE CAPTURING SCHEME FOR MIXED MULTIPHASE FLOWS <i>Tandon Mohit P., Gada Vinesh H., Elias Jebin, Lo Simon</i>	25/1
	FRONT-TRACKING SIMULATION OF ELECTROWETTING DROPLET MANIPULATION <i>Yamamoto Yasufumi, Otomo Ryoko, Ito Takahiro, Wakimoto Tatsuro, Katoh Kenji</i>	25/2
<b>Mo 14:40-16:00</b>	<b>Multiphase Flow in Heat and Mass Transfer</b> <i>Room: Affari-2.C</i> THE EFFECTS OF NANOPARTICLES IN SUBCOOLED FLOW BOILING IN THE DIFFERENT CHANNELS WITH SAME HYDRAULIC DIAMETER <i>Mehrzad Shams, Alimoradi Hasan, Valizadeh Ziba</i>	25/3
	EXPERIMENTAL INVESTIGATION OF HIGH PRESSURE WATER-STEAM FLOW CHARACTERS IN SIMULATED CRACKS <i>Zhendong Yang, Zhang Qiaoling, Bi Qincheng</i>	25/4

	A NEW MODEL FOR THE PREDICTION OF FRICTIONAL PRESSURE DROP OF AIR-WATER TWO-PHASE FLOWS IN VERTICALLY-DOWNWARD TUBES <i>Yao Chao, Li Hui-Xiong, Xue Yuqing, Hao Chenyu</i>	25/5
	EFFECT OF FLOW OBSTACLES ON THE POOL BOILING CRITICAL HEAT FLUX OF DOWNWARD-FACING METAL HEATER <i>Son Hong Hyun, Song Kyusang, Jeong Uiju, Seo Gwang Hyeok, Jeun Gyoodong, Kim Sung Joong</i>	25/6
<b>Mo 14:40-16:00</b>	<b>Particle and Drop Dynamics</b> <i>Room: Villa-Sala Verde</i>	
	SURFACTANT EFFECTS ON DROPLET COALESCENCE WITH A LIQUID-LIQUID INTERFACE <i>Angeli Panagiota, Voulgaropoulos Victor, Wehliye Hashi Wehliye, Chinaud Maxime</i>	26/1
	EXPERIMENTAL EVALUATION OF SURFACTANT CONCENTRATION AT AN INTERFACE OF SINGLE SPHERICAL DROP <i>Hosokawa Shigeo, Masukura Yuya, Hayashi Kosuke, Tomiyama Akio</i>	26/2
	NUMERICAL SIMULATION OF SURFACTANT CONCENTRATION AT AN INTERFACE OF SINGLE SPHERICAL DROP <i>Hayashi Kosuke, Masukura Yuya, Hosokawa Shigeo, Tomiyama Akio</i>	26/3
	EFFECT OF INTERFACIAL PROPERTIES ON DROPLET-INTERFACE COALESCENCE: A COMPUTATIONAL STUDY <i>Li Kai, Ma Mengyu, Liu Nannan, Jin Hang, Wang Wei</i>	26/4
<b>Mo 14:40-16:00</b>	<b>Particle-Laden Flows</b> <i>Room: Villa-Auditorium</i>	
	MODELLING AND SIMULATION OF PARTICLE AGGLOMERATION AND DEPOSITION IN TURBULENT CHANNEL AND PIPE FLOWS <i>Schutte Koen, Portela Luis, Twerda Aris, Henkes Ruud</i>	26/5
	NUMERICAL INVESTIGATION OF PARTICLE DEPOSITION IN TURBULENT DUCT FLOWS <i>Yao Jun, Fairweather Michael, Zhao Yanlin</i>	26/6
	THE EFFECTS OF NEAR WALL CORRECTIONS TO HYDRODYNAMIC FORCES ON PARTICLE DEPOSITION AND TRANSPORT IN VERTICAL TURBULENT BOUNDARY LAYERS <i>Reeks Michael, Jin Chunyu, Potts Ian</i>	27/1
	TWO-WAY COUPLING EFFECTS IN PARTICLE-LADEN TURBULENCE: HOW PARTICLE-TRACKING SCHEME AFFECTS PARTICLE AND FLUID STATISTICS <i>Horwitz Jeremy, Rahmani Mona, Geraci Gianluca, Banko Andrew, Mani Ali</i>	27/2
<b>Mo 14:40-16:00</b>	<b>Multiphase Pipe Flows</b> <i>Room: Affari-4.A</i>	
	OIL-WATER FLOW CHARACTERISATION IN HORIZONTAL PIPES USING SIMULTANEOUS LASER-INDUCED FLUORESCENCE AND PARTICLE IMAGE/TRACKING VELOCIMETRY <i>Ibarra Roberto, Zadrazil Ivan, Markides Christos N., Matar Omar K.</i>	27/3
	EFFECTS OF AIR INJECTION POSITIONS ON THE FLOW STRUCTURE OF HIGH VISCOSITY OIL IN A LARGE DIAMETER PIPES <i>Mohammed Shara, Hasan Abbas, Azzopardi Barry James, Dimitrakis Georgios</i>	27/4
	MEASUREMENT OF WALL SHEAR STRESS IN CHURN FLOW IN A LARGE DIAMETER VERTICAL RISER <i>Zangana Mohammed, Azzopardi Barry James</i>	27/5
	ENTRANCE EFFECTS ON TWO-PHASE FLOW DEVELOPMENT IN A VERTICAL PIPE: EXPERIMENTS AND SIMULATIONS <i>Issa Raad, Galleni Francesco, Azzopardi Barry James, Hewakandamby Buddhikha, Escrig Escrig Josep</i>	27/6
<b>Mo 14:40-16:00</b>	<b>Bubble Dynamics</b> <i>Room: Affari-2.D</i>	
	CONTROLLED BUBBLE FORMATION BY A FORCING MECHANISM IN A PLANAR CO-FLOW CONFIGURATION <i>Ruiz-Rus Javier, Bolanos Jimenez Rocio, Gutierrez Montes Candido, Sevilla Alejandro, Bazan Carlos Martinez</i>	28/1
	STUDY OF A BUBBLE PRODUCTION IN SHEAR LIQUID-GAS FLOWS USING THE BRIGHTNESS BASED LASER INDUCED FLUORESCENCE TECHNIQUE (BBLIF) <i>Hann David, Cherdantsev Andrey, Hewakandamby Buddhikha, Azzopardi Barry James</i>	28/2
	BUBBLES RELEASE FROM HYDROPHOBIC SITE UNDER MAGNETIC FIELD <i>Liu Hongbo, Pan Liangming, Chen Tianming</i>	28/3
	ON THE PHYSICS OF FIZZINESS: HOW BUBBLE BURSTING CONTROLS DROPLETS EJECTION <i>Seon Thomas, Ghabache Elisabeth, Antkowiak Arnaud, Josserand Christophe</i>	28/4

## PARALLEL SESSIONS IV

Mo 16:20-17:40	<b>Bio-Fluids</b>	
	<i>Room: Affari-2.B</i>	
	NUMERICAL SIMULATION OF MAGNETIC DRUG DELIVERY TO HUMAN MAXILLARY SINUS	
	<i>Nikookar Hamed, Abouali Omid, Eghtesad Mohammad, Ahmadi Goodarz</i>	28/5
	MODAL BEHAVIOUR OF THREE-DIMENSIONAL ARTERIAL FLOW STRUCTURES	
Mo 16:20-17:40	<b>Boiling, Condensation, Evaporation</b>	
	<i>Room: Affari-3.D</i>	
	EXPERIMENTS AND MODELS FOR TRANSIENT BOILING	
	<i>Baudin Nicolas, Ruyer Pierre, Colin Catherine, Sebillieu Julien</i>	29/2
	EXPERIMENTAL INVESTIGATION OF THE FLOW BOILING HEAT TRANSFER CHARACTERISTICS WITH SUPER-HYDROPHILIC SURFACE IN NATURAL CIRCULATION THERMOSYPHON LOOP	
Mo 16:20-17:40	<b>Cavitation/Nucleation</b>	
	<i>Room: Affari-3.C</i>	
	AN EULER-LAGRANGE METHOD WITH VOLUMETRIC COUPLING FOR CAVITATING FLOWS	
	<i>Schumichen Michel, Rudiger Frank, Frohlich Jochen</i>	29/6
	VOID FRACTION FLOW FIELD MEASUREMENTS OF CAVITATING FLOW IN THE WAKE OF A CIRCULAR CYLINDER	
Mo 16:20-17:40	<b>Collision, Agglomeration and Breakup</b>	
	<i>Room: Affari-3.A</i>	
	BEHAVIOR OF DROPLETS IMPACT ON DRY AND WET WALLS: A NUMERICAL STUDY	
	<i>Chen Ming, Zhang Baili, Lou Jing</i>	30/4
	THE BREAKUP OF A VISCOUS LIQUID DROP IN A HIGH REYNOLDS NUMBER SHEAR FLOW	
Mo 16:20-17:40	<b>Computational Techniques for Multiphase Flows</b>	
	<i>Room: Affari-1.A</i>	
	A THREE-PHASE NUMERICAL MODEL FOR LIQUID-GAS-GAS FLOWS WITH RELAXATION PROCESSES	
	<i>Pelanti Marica</i>	31/2
	THE SIMULATION OF MULTILAYERED MEDIA OF FLUID AND SOLID USING THE GHOST METHOD	
Mo 16:20-17:40		
	<i>Feng Zhi Wei, Khoo Boo Cheong, Rong Ji Li</i>	31/3
	A VARIATIONAL APPROACH TO DESIGN A NUMERICAL SCHEME ON AN ARBITRARY MOVING GRID FOR N-FLUID FLOW WITH THERMODYNAMIC CONSISTENCY	
	<i>Vazquez Gonzalez Thibaud, Llor Antoine, Fochesato Christophe</i>	31/4
	A PARALLEL ADAPTIVE FINITE ELEMENT METHOD FOR SIMULATING THE INCOMPRESSIBLE FLOWS WITH FREE SURFACE	
Mo 16:20-17:40		
	<i>Ngo Long, Jeon Byoungjin, Choi Hyoung</i>	31/5

<b>Mo 16:20-17:40</b>	<b>Experimental Methods for Multiphase Flows</b> <i>Room: Affari-4.A</i>	
	FLOW MEASUREMENT IN OPAQUE FLUIDS USING ULTRASOUND IMAGING VELOCIMETRY	31/6
	<i>Poelma Christian, Gurung Arati</i> X-RAY RADIOSCOPY AS A POWERFUL TOOL FOR VISUALIZATION OF LIQUID METAL BUBBLY FLOWS	32/1
	<i>Roshchupkina Olga, Shevchenko Natalia, Strumpf Erik, Hoppe Dietrich, Eckert Sven</i> GAS AND LIQUID DYNAMICS IN BUBBLE COLUMNS: ADVANCED FLOW IMAGING WITH ULTRAFAST X-RAY AND RADIOACTIVE PARTICLE TRACKING	32/2
	<i>Azizi Salar, Yadav Ashutosh, Lau Yuk Man, Roy Shantanu, Schubert Markus</i> A STUDY OF LIQUID-LIQUID SECONDARY FRAGMENTATION WITH SOLIDIFICATION	32/3
	<i>Hadj-Achour Miloud, Rimbert Nicolas, Castrillon-Escobar Sebastian, Gradeck Michel</i>	
<b>Mo 16:20-17:40</b>	<b>Industrial Applications</b> <i>Room: Affari-0.A</i>	
	EXPERIMENTAL AND COMPUTATIONAL INVESTIGATION OF FLOW INSIDE A MILLING CIRCUIT HYDROCYCLONE	32/4
	<i>Furlan John, Ke Renjie, Shingote Chinmay, Kadambi Jaikrishnan, Visintainer Robert</i> REMOVAL OF FINE PARTICLES ON A WALL BY HIGH-SPEED AIR JET EJECTED FROM A CAVITY NOZZLE (OPTIMIZATION OF NOZZLE SHAPE)	32/5
	<i>Wakimoto Tatsuro, Nakagawa Atsushi, Soemoto Kazuhiko, Katoh Kenji</i> NONINVASIVE TEMPERATURE DISTRIBUTION IMAGING OF HIGH VISCOSITY PLASTIC FLOW BY ELECTRICAL CAPACITANCE TOMOGRAPHY	32/6
	<i>Hirose Yusuke, Hata Kazuaki, Takei Masahiro</i>	
<b>Mo 16:20-17:40</b>	<b>Interfacial Flows</b> <i>Room: Affari-2.A</i>	
	A NUMERICAL CLOSE-UP ON THE SPRAY FORMATION FROM A GAS-LIQUID MIXING LAYER	33/1
	<i>Ling Yue, Fuster Daniel, Tryggvason Gretar, Scardovelli Ruben, Zaleski Stephane</i> INFLUENCE OF DISSOLVED GASES ON ATOMIZATION PROCESSES.	33/2
	<i>Tinon Emmanuelle, Martin Charles-Etienne, Bazile Rudy, Selle Laurent</i> A CONSISTENT MOMENTUM TRANSPORT IN CLSVOF METHOD: VALIDATION ON JET ATOMIZATION	33/3
	<i>Berlemont Alain, Menard Thibaut, Aniszewski Wojciek, Vaudor Geoffroy</i> ON THE GAS-LIQUID ENTRAINMENT BY IMPINGEMENT OF LIQUID JET ONTO A POOL	33/4
	<i>Sanjay Vatsal, Das Arup Kumar</i>	
<b>Mo 16:20-17:40</b>	<b>Multiphase Flow in Heat and Mass Transfer</b> <i>Room: Affari-2.C</i>	
	EFFECT OF THE ALIGNED FLOW OBSTACLES ON SUBCOOLED FLOW BOILING IN AN INCLINED RECTANGULAR CHANNEL	33/5
	<i>Jeong Uiju, Son Hong Hyun, Song Kyusang, Seo Gwang Hyeok, Jeon Joon Goo, Kim Sung Joong</i> HEAT TRANSFER CHARACTERISTICS OF SUBCOOLED WATER FLOW BOILING IN CIRCULAR CHANNELS WITH NONUNIFORM HEATING FLUXES	33/6
	<i>Zhu Ge, Bi Qincheng, Yan Jianguo, Yuan Qizheng, Lyu Haicai, Pan Hui</i> A STUDY ON THE BEHAVIOR OF MECHANISTIC CRITICAL HEAT FLUX MODELS FOR FLOW BOILING AT AN ISOTHERMAL WALL USING A REFRIGERANT	34/1
	<i>Bruder Moritz, Heublein Norbert, Sattelmayer Thomas</i> CONVECTIVE LOCAL HEAT TRANSFER AROUND ELONGATED BUBBLES PROPAGATING IN DOWNWARD LIQUID PIPE FLOW	34/2
	<i>Fershtman Adam, Barnea Dvora, Shemer Lev</i>	
<b>Mo 16:20-17:40</b>	<b>Particle and Drop Dynamics</b> <i>Room: Villa-Sala Verde</i>	
	INVESTIGATION ON THE DEFORMATION BEHAVIOR BETWEEN OIL DROPLETS IN AQUEOUS SOLUTION	34/3
	<i>Liu Fangyuan, Wang Wei, Jin Hang, Li Kai, Chang Hongli, Gong Jing</i> EFFECT OF LIQUID PROPERTIES ON THE INTERFACIAL INSTABILITY AND ATOMIZATION BEHAVIOR OF THE ACOUSTICALLY LEVITATED DROPLET	34/4
	<i>Kono Kengo, Hasegawa Koji, Imai Hiroaki, Ohtake Hiroyasu, Goda Atsushi, Abe Yutaka</i> DROP DEFORMATION IN VISCOUS COMPRESSIONAL FLOW: FLAT AND TOROIDAL DROPS	34/5
	<i>Nir Avinoam, Lavrenteva Olga, Smagin Irina, Zabaranin Michael</i> HOMOGENIZATION: SIMPLIFIED DROP DEFORMATION ESTIMATE DURING TRANSITION THROUGH ORIFICES	34/6
	<i>Walzel Peter</i>	

Mo 16:20-17:40

## Particle-Laden Flows

*Room: Villa-Auditorium*

TWO WAY COUPLING IN PARTICLE-LADEN TURBULENT JET

*Battista Francesco, Gualtieri Paolo, Sinibaldi Giorgia, Casciola Carlo Massimo*

35/1

AN EXPERIMENTAL INVESTIGATION OF THE THREE-WAY COUPLING BETWEEN  
 TURBULENCE, PARTICLES, AND RADIATION

*Banko Andrew, Villafane Laura, Elkins Christopher, Eaton John*

35/2

TWO-WAY INTERACTIONS IN PARTICLE-LADEN TURBULENT CHANNEL FLOW

*Peng Cheng, Wang Lian-Ping*

35/3

MEASUREMENTS OF PARTICLE SETTLING VELOCITY IN HOMOGENEOUS TURBULENCE  
 WITH NO MEAN FLOW

*Coletti Filippo, Petersen Alec, Carter Douglas, Baker Lucia*

35/4

Mo 16:20-17:40

## Multiphase Pipe Flows

*Room: Affari-4.A*

MODELLING OF CHURN-ANNULAR FOAM FLOWS

*Westende Jos, Shoeibiomrani Pejman, Vercauteren Frank, Nennie Erik*

35/5

SOME CHARACTERISTICS OF LIQUID DROPLETS IN VERTICAL UPWARD ANNULAR-MIST  
 FLOW

*Wang Yuchen, Andayi Samwel, Hazuku Tatsuya, Takamasa Tomoji*

35/6

IMPACT OF GAS VELOCITY PROFILE ON SIMULATED WAVE FREQUENCIES IN VERTICAL  
 AIR-WATER CHURN FLOW

*Tekavcic Matej, Koncar Bostjan, Kljenak Ivo*

36/1

PRESSURE GRADIENT VARIATIONS IN GAS-LIQUID DOWNWARD FLOWS IN A VERTICAL  
 LARGE DIAMETER PIPE

*Aliyu Aliyu, Lao Liyun, Yeung Hoi*

36/2

Mo 16:20-17:40

## Bubble Dynamics

*Room: Affari-2.D*

ON THE DYNAMICS OF NON-CONDENSABLE GAS IN A DIESEL INJECTOR

*Arienti Marco, Sussman Mark*

36/3

MODELLING DROP OR BUBBLE BREAKUP IN TURBULENT FLOWS

*Lalanne Benjamin, Masbernat Olivier, Risso Frederic*

36/4

INTERACTION OF CLEAN OR CONTAMINATED SPHERICAL PAIR BUBBLES IN QUIESCENT  
 LIQUIDS

*Sanada Toshiyuki, Kusuno Hiroaki*

36/5

ENTRAINMENT CAUSED BY A SINGLE BUBBLE THROUGH LIQUID-LIQUID INTERFACE IN  
 UNCONFINED SPACE

*Kar Abhimanyu, Das Prasanta Kumar*

36/6

## TUESDAY, May 24, 2016

### PARALLEL SESSIONS V

Tu 9:50-11:10	<b>Bubbly Flows</b>	
	<i>Room: Affari-2.B</i>	
	A PENALIZATION METHOD FOR THE SIMULATION OF BUBBLY FLOWS <i>Morente Antoine, Lavieville Jerome, Legendre Dominique</i>	37/1
	TURBULENT FLOWS WITH COALESCING MICROBUBBLES: AN EULER-LAGRANGE APPROACH BASED ON LARGE-EDDY SIMULATION <i>Hoppe Felix, Breuer Michael</i>	37/2
	SIMULATION OF BUBBLY FLOW IN AN INDUSTRIAL ELECTROCHEMICAL REACTOR <i>Arjmand Mehdi, Vreman Bert, Bokkers Albert, Lif Johan, Pelin Kalle</i>	37/3
Tu 9:50-11:10	EFFECT OF INLET CONDITIONS ON TWO PHASE FLOW PATTERN PREDICTION IN SMALL TUBES <i>Gregorc Jurij, Zun Iztok</i>	37/4
	<b>Boiling, Condensation, Evaporation</b>	
	<i>Room: Affari-3.D</i>	
	INFLUENCE OF CONVECTION IN TUBE BUNDLES DURING BOILING <i>Luke Andrea, Addy Joseph, Mueller Bjoern</i>	37/5
	SYNTHESIS OF CRUD AND ITS EFFECTS ON POOL AND SUBCOOLED FLOW BOILING <i>Coyle Carolyn, Phillips Bren, Lowder Sean, Buongiorno Jacopo, Mckrell Thomas, Rubner Michael</i>	37/6
Tu 9:50-11:10	TRANSIENT MASS TRANSFER FROM A SPHERICAL DROPLET EVAPORATING IN GASEOUS ENVIRONMENT <i>Tonini Simona, Cossali Gianpietro Elvio</i>	38/1
	SUPPRESSION MEASURES OF STEAM EXPLOSIONS WITH AND WITHOUT EXTERNAL PRESSURE PULSE <i>Furuya Masahiro, Arai Takahiro</i>	38/2
	<b>Cavitation/Nucleation</b>	
	<i>Room: Affari-3.C</i>	
	EFFECT OF TRIPLE LINE ON SPHERICAL-SUBSTRATE-MEDIATED CONDENSATION <i>Singha Sanat Kumar, Das Prasanta Kumar, Maiti Biswajit</i>	38/3
Tu 9:50-11:10	EFFECT OF THE RECESS GEOMETRY ON CRYOGENIC CAVITATING FLOW THROUGH THE FLOATING-RING CLEARANCE <i>Oike Mamoru, Kikuchi Masataka, Takada Hitoshi, Sudo Takayuki, Takano Tomoyuki</i>	38/4
	BECKER-DÖRING MODEL FOR NON-EQUILIBRIUM STEAM CONDENSATION IN A SUPERSONIC NOZZLE <i>Hagmeijer Rob, Van Der Weide Edwin, Kalikmanov Vitaly</i>	38/5
	INTERFACE CAPTURING SIMULATIONS OF SHOCK-ACCELERATED BUBBLE PAIRS <i>Shukla Ratnesh</i>	38/6
	<b>Collision, Agglomeration and Breakup</b>	
	<i>Room: Affari-3.A</i>	
Tu 9:50-11:10	NUMERICAL STUDY OF WATER DROPS BEHAVIOUR IN OIL UNDER ELECTRIC FIELD <i>Chen Bin, Huang Meng</i>	39/1
	NUMERICAL SIMULATIONS ON COPPER DROPLET COLLISIONS IN THE SLAG CLEANING PROCESS <i>Yang Hongkun, Wolters Jorg, Pischke Philipp, Soltner Helmut, Frohlich Jochen, Eckert Sven</i>	39/2
	EXPERIMENTAL OBSERVATION OF THREE-DIMENSIONAL PARTICLE CLUSTERING IN TURBULENT CHANNEL FLOW <i>Nemes Andras, Coletti Filippo, Fong Kee Onn, Sloan Nicholas</i>	39/3
	DETERMINISTIC AGGLOMERATION MODELS FOR THE EULER-LAGRANGE APPROACH USING LARGE-EDDY SIMULATION <i>Almohammed Naser, Breuer Michael</i>	39/4
	<b>Computational Techniques for Multiphase Flows</b>	
Tu 9:50-11:10	<i>Room: Affari-1.A</i>	
	3D NUMERICAL SIMULATIONS OF MACROSCOPIC FLOWS WITH MOVING CONTACT LINES <i>Solomenko Zlatko, Spelt Peter, Lance Michel, Alix Pascal, Dalmazzone Christine, Haroun Yacine</i>	39/5
	DETAILED MODELING OF SLOSHING IN SATELLITES TANK FOR LOW BOND NUMBERS <i>Lepilliez Mathieu, Tanguy Sebastien</i>	39/6
	LBE-DEM SIMULATION OF GAS-SOLID TWO-PHASE IMPINGING STREAM <i>Xu Wenkai, Gui Nan, Li Zhenlin</i>	40/1
	NUMERICAL INVESTIGATION OF DUAL BUBBLE SYSTEM <i>Al-Behadili Mustapha, Hargreaves David, Hewakandamby Buddhikha, Azzopardi Barry James</i>	40/2



<b>Tu 9:50-11:10</b>	<b>Experimental Methods for Multiphase Flows</b> <b>Room: Affari-4.A</b> FLAPPING OF AN AXISYMMETRIC LIQUID JET <i>Delon Antoine, Matas Jean-Philippe, Cartellier Alain</i> 40/3 DROPLET SIZE MEASUREMENT IN STATIC MIXERS USING REFRACTIVE INDEX MATCHING AND LASER-INDUCED FLUORESCENCE <i>Rudolf von Rohr Philipp, Hafeli Richard, Ruegg Oliver, Altheimer Marco</i> 40/4 DEVELOPMENT OF AN EXPERIMENTAL APPARATUS AND MEASUREMENT TECHNIQUES FOR THE STUDY OF QUASI-REAL SLUG-FLOW IN VERTICAL DUCTS <i>Cerqueira Rafael, Paladino Emilio, Brito Ronei, Maliska Clovis</i> 40/5 EXPERIMENTAL CHARACTERIZATION OF INTERFACIAL WAVE STRUCTURE OF A FALLING LIQUID FILM IN A VERTICAL LARGE PIPE DIAMETER <i>Hasan Abbas, Azzopardi Barry James, Hewakandamby Buddhikha, Alaofi Yousuf, Watson Nicholas, Mohammed Shara</i> 40/6
<b>Tu 9:50-11:10</b>	<b>Industrial Applications</b> <b>Room: Affari-0.A</b> VOID FRACTION CHARACTERIZATION THROUGH AN OPTICAL METHOD FOR AIR-WATER UPWARD FLOW ACROSS TUBE BUNDLE <i>Kanizawa Fabio Toshio, Ribatski Gherhardt</i> 41/1 COMPUTATIONALLY-EFFICIENT SIMULATION OF INDUSTRIAL BUBBLE COLUMN BIOREACTORS: INVESTIGATION OF SCALE AND MIXING EFFECTS ON YIELD <i>Fletcher David, McClure Dale, Kavagnah John, Barton Geoff</i> 41/2 TOWARDS THE NUMERICAL SIMULATION OF TWO-PHASE FLOW AND INTERFACIAL SPECIES TRANSFER IN STRUCTURED PACKINGS <i>Hill Simon, Acher Thomas, Deising Daniel, Hoffmann Rainer, Rarrek Andreas, Marschall Holger, Bothe Dieter, Klein Harald</i> 41/3 MEASURING LIQUID FLOW IN STRUCTURED PACKINGS USING GAMMA AND X-RAY TOMOGRAPHY <i>Hoffmann Rainer</i> 41/4
<b>Tu 9:50-11:10</b>	<b>Interfacial Flows</b> <b>Room: Affari-2.A</b> A NUMERICAL AND ANALYTICAL STUDY ON THE MICRO-JET INDUCED BY A SHOCK <i>Bai Xiao, Deng Xiaolong</i> 41/5 NUMERICAL SIMULATION OF AN AIR JET IN CROSS WATER FLOW WITH APPLICATION TO AIR-LAYER DRAG REDUCTION <i>Moon Hojun, You Donghyun</i> 41/6 NUMERICAL SIMULATION OF A HIGHLY-VISCOUS MICROJET GENERATED BY AN IMPULSIVE FORCE <i>Onuki Hajime, Yuto Oi, Tagawa Yoshiyuki</i> 42/1 ANALYSIS OF INTERMITTENT FLOW PATTERNS OF WATER-STEAM FLOW IN SOLAR THERMAL POWER PLANTS WITH DIRECT STEAM GENERATION <i>Hoffmann Alexander, Hirsch Tobias, Schleicher Eckhard, Pitz-Paal Robert</i> 42/2
<b>Tu 9:50-11:10</b>	<b>Modelling of Multiphase Flows</b> <b>Room: Affari-2.D</b> THREE DIMENSIONAL MULTI-PHASE EULERIAN MODEL FOR NON-UNIFORM SEDIMENT- LADEN FLOW <i>Ota Kazuyuki, Sato Takahiro, Nakagawa Hajime</i> 42/3 THE MODELING AND EXPERIMENTS OF AN ADVANCED OZONATION SYSTEM WITH RADIAL DIFFUSER <i>Swiderski Karol, Heiniger Bruno, Ramoino Luca, Uttinger Walter, Lakehal Djamel, Narayanan Chidambaram</i> 42/4 NUMERICAL MODELLING OF MULTIPHASE FLOW PHENOMENA COUPLED WITH REACTION KINETICS IN CHEMICAL LOOPING COMBUSTION PROCESS OF A FUEL REACTOR <i>Chen Luming, Yang Xiaogang, Li Xia, Snape Collin</i> 42/5 TURBULENCE AND MOMENTUM INTERFACIAL TRANSFER IN AN AXISYMMETRIC BUBBLY JET FLOW <i>Aouadi Aroua, Bellakhal Ghazi, Chahed Jamel</i> 42/6
<b>Tu 9:50-11:10</b>	<b>Multiphase Flow in Heat and Mass Transfer</b> <b>Room: Affari-2.C</b> THE EFFECT OF THE CHANNEL INCLINATION ON FLOW PATTERN CHARACTERISTICS DURING GAS-LIQUID FLOW IN A SMALL RECTANGULAR CHANNEL <i>Loyola Francisco, Ribatski Gherhardt</i> 43/1 EFFICIENT COMPRESSED AIR ENERGY STORAGE THROUGH SPRAY-COOLING <i>Qin Chao, Loth Eric</i> 43/2

	VISUALIZATION AND HEAT TRANSFER OF HIGH PRESSURE CARBON DIOXIDE IN A SMALL HORIZONTAL TUBE <i>Lyu Haicai, Bi Qincheng, Li Hang, Guo Yajun</i>	43/3
	FLOW BOILING HEAT TRANSFER AND TWO-PHASE FLOW CHARACTERISTICS IN 4.0 MM TUBE AT MICROGRAVITY CONDITIONS <i>Valencia Castillo Cesar Manuel, Gugliemetti Luca, Bragelli Leonardo, Saraceno Luca, Zummo Giuseppe</i>	43/4
<b>Tu 9:50-11:10</b>	<b>Particle and Drop Dynamics</b> <i>Room: Villa-Sala Verde</i> ON THE CLASSIFICATION OF DROPLET BREAKUP IN AN EXTERIOR STREAM <i>Rimbert Nicolas, Hadj-Achour Miloud</i>	43/5
	BINARY DROPLET INTERACTIONS AND COALESCENCE – SUPERPOSITION OF HYDRODYNAMICS AND IONIC EFFECTS <i>Gebauer Felix, Villvock Jorn, Kraume Matthias, Bart Hans-Jorg</i>	43/6
	CONTACTLESS COALESCENCE TECHNIQUE OF ACOUSTICALLY LEVITATED DROPLETS USING ULTRASONIC TRANSDUCER ARRAY <i>Watanabe Ayumu, Watahiki Soma, Niwa Motonori, Goda Atsushi, Kanagawa Tetsuya, Kaneko Akiko, Abe Yutaka</i>	44/1
<b>Tu 9:50-11:10</b>	<b>Particle-Laden Flows</b> <i>Room: Villa-Auditorium</i> DIRECT NUMERICAL SIMULATION OF THE FORMATION OF SEDIMENT PATTERNS UNDER THE ACTION OF AN OSCILLATING FLOW <i>Mazzuoli Marco, Kidanemariam Aman, Blondeaux Paolo, Vittori Giovanna, Uhlmann Markus</i>	44/2
	STRONGLY COUPLED PARTICLE-LADEN FLOWS IN VERTICAL CHANNELS <i>Fox Rodney, Capecehatro Jesse, Desjardins Olivier</i>	44/3
	INVESTIGATION OF SUBGRID SCALE EFFECTS TO PARTICLE PAIR DYNAMICS WITH COHERENT VORTEX EXTRACTION <i>Xiong Yan, Li Jing, Liu Zhaohui, Zheng Chuguang</i>	44/4
	PREFERENTIAL ACCUMULATION AND ENHANCED RELATIVE VELOCITY OF INERTIAL DROPLETS DUE TO INTERACTIONS WITH HOMOGENEOUS ISOTROPIC TURBULENCE <i>Bateson Colin, Aliseda Alberto</i>	44/5
<b>Tu 9:50-11:10</b>	<b>Multiphase Pipe Flows</b> <i>Room: Affari-3.E</i> COMPARISON OF VOID FRACTION MEASUREMENTS USING DIFFERENT TECHNIQUES IN DOWNWARD TWO-PHASE FLOW <i>Kawaji Masahiro, Kalaga Dinesh, Turney Damon, Hernandez Freddy, Kleinbart Simon, Pulido Jorge</i>	44/6
	LOW GAS LOADING FLOWS IN DOWNWARD INCLINED PIPES <i>Johansen Stein Tore, Mo Sjur, Meese Ernst A.</i>	45/1
	STUDIES OF DOWNWARD GAS-LIQUID ANNULAR FLOW <i>Anupriya Sethu, Jayanti Sreenivas</i>	45/2
	AN EXPERIMENTAL INVESTIGATION OF INTERFACE SHAPE IN SEPARATED FLOW REGIMES INSIDE THE INCLINED TUBE. <i>Ahn Taehwan, Yun Byongjo, Jeong Jaejun</i>	45/3
<b>Tu 9:50-11:10</b>	<b>Bubble Dynamics</b> <i>Room: Affari-3.B</i> MODELLING OF TWO SPHERICAL OSCILLATING BUBBLES IN AN ELASTIC TUBE <i>Takahira Hiroyuki, Kitahara Tatsuya, Ogasawara Toshiyuki</i>	45/4
	AN EXPERIMENTAL INVESTIGATION INTO THE CLUSTERING OF SPHERICAL BUBBLES RISING ALONG AN INCLINED FLAT PLATE <i>Ogasawara Toshiyuki, Shirai Shota, Takahira Hiroyuki</i>	45/5
	INTERACTION OF TWO BUBBLES RISING IN A THIN-GAP CELL: EXPERIMENTS AND MODEL FOR THE VERTICAL ENTRAINMENT <i>Roig Veronique, Filella Audrey, Ern Patricia</i>	45/6
	ADHESION AND SLIDING DYNAMICS OF AIR BUBBLES ON SUPERAEROPHOBIC SURFACES <i>Ozbay Ridvan, Kibar Ali, Choi Chang-Hwan</i>	46/1

## PARALLEL SESSIONS VI

<b>Tu 11:30-12:50</b>	<b>Bubbly Flows</b> <i>Room: Affari-2.B</i> EXPERIMENTAL INVESTIGATION OF HEAT TRANSPORT IN BUBBLY FLOW <i>Gvozdic Biljana, Almeras Elise, Mathai Varghese, Sun Chao, Lohse Detlef</i>	46/2
-----------------------	---	------

	FLOW STRUCTURES ANALYSIS DOWNSTREAM GAS SPARGERS OF DIFFERENT ORIFICE PATTERN IN BUBBLE COLUMNS <i>Moeller Felix, Bieberle Andre, Barthel Frank, Schubert Markus, Hampel Uwe, Weber Manfred, Weber Markus</i>	46/3
	ON THE FORMATION OF MESO-SCALE STRUCTURES IN DISPERSED GAS-LIQUID FLOWS <i>Strom Henrik, Jareteg Klas, Sasic Srdjan, Demaziere Christophe</i>	46/4
	INVESTIGATION OF SINGLE BUBBLES AND BUBBLE CHAINS IN LIQUID METAL UNDER THE INFLUENCE OF MAGNETIC FIELDS <i>Strumpf Erik, Shevchenko Natalia, Roshchupkina Olga, Eckert Sven</i>	46/5
<b>Tu 11:30-12:50</b>	<b>Boiling, Condensation, Evaporation</b> <i>Room: Affari-3.D</i> BOILING FRONT FORMATION IN FLASH DEPRESSURIZATION <i>Djeridi Henda, Matas Jean-Philippe, Boivin Pierre, Cartellier Alain</i>	46/6
	COMPARATIVE ASSESSMENT OF FLOW BOILING HEAT TRANSFER CORRELATIONS FOR HORIZONTAL MINI AND MICRO CHANNELS WITH CO <sub>2</sub> AS WORKING FLUID <i>Gamit Jigar, Channiwala Salim</i>	47/1
	A LOOK ON RELEVANT NONDIMENSIONAL NUMBERS FOR FLOW BOILING IN MICROCHANNELS AND MINICHANNELS <i>Awad Mohamed, Wu Zan, Sunden Bengt</i>	47/2
	A MECHANISTIC IR CALIBRATION TECHNIQUE FOR BOILING HEAT TRANSFER INVESTIGATIONS <i>Richenderfer Andrew, Bucci Matteo, Mckrell Thomas, Buongiorno Jacopo</i>	47/3
<b>Tu 11:30-12:50</b>	<b>Cavitation/Nucleation</b> <i>Room: Affari-3.C</i> EXPERIMENTAL AND NUMERICAL INVESTIGATION OF FLASH ATOMIZATION WITH CRYOGENIC FLUIDS <i>Luo Meng, Ramcke Thomas, Pfitzner Michael, Haidn Oskar</i>	47/4
	STRUCTURE OF A LASER-INDUCED UNDERWATER SHOCK WAVE <i>Tagawa Yoshiyuki, Yamamoto Shota, Hayasaka Keisuke, Kameda Masaharu</i>	47/5
	CAVITY SHEDDING MECHANISMS ON A NACA0015 HYDROFOIL <i>Ganesh Harish, Wu Juliana, Ceccio Steven L.</i>	47/6
	JATROPHA BIODIESEL PRODUCTION USING HYDRODYNAMIC CAVITATION <i>Dwivedi Prateek, Ghosh Sumana</i>	48/1
<b>Tu 11:30-12:50</b>	<b>Collision, Agglomeration and Breakup</b> <i>Room: Affari-3.A</i> EXPERIMENTAL INVESTIGATION OF THE LORENTZ-FORCE-DRIVEN CONVECTION AROUND A GROWING HYDROGEN BUBBLE <i>Baczyszalski Dominik, Karnbach Franziska, Yang Xuegeng, Mutschke Gerd, Uhlemann Margitta, Eckert Kerstin, Cierpka Christian</i>	48/2
	CHARACTERIZATION OF FLOC MORPHOLOGY IN A TURBULENT TAYLOR-COUPETTE REACTOR <i>Guerin Lea, Coufort Saudejaud Carole, Line Alain, Frances Christine</i>	48/3
	JETTING REGIME IN CO-FLOWING CAPILLARY JETS IN CONFINED GEOMETRY <i>Nowak Emilia, Kahouadji Lyes, Matar Omar K., Simmons Mark</i>	48/4
	NUMERICAL STUDY OF AGGREGATION DYNAMICS WITH A LATTICE BOLTZMANN METHOD COUPLED WITH DISCRETE ELEMENTS THROUGH FLOW PENALIZATION <i>Ngoma Jeff, Kroll-Rabotin Jean-Sebastien</i>	48/5
<b>Tu 11:30-12:50</b>	<b>Computational Techniques for Multiphase Flows</b> <i>Room: Affari-1.A</i> DIRECT NUMERICAL SIMULATIONS OF FLOW AROUND A NON-SPHERICAL PARTICLE <i>Pacha Sanjeevi Sathish, Padding Johan, Kuipers J. A. M (hans)</i>	48/6
	A COUPLED EULERIAN-LAGRANGIAN METHOD FOR SPRAY ATOMIZATION ON UNSTRUCTURED MESHES <i>Evrard Fabien, Denner Fabian, Serfaty Ricardo, Van Wachem Berend</i>	49/1
	MULTI-MATERIAL MODELLING FOR COMPRESSIBLE FLOWS WITH THE CONSERVATIVE SHARP-INTERFACE METHOD <i>Pan Shucheng, Hu Xiangyu, Adams Nikolaus</i>	49/2
	SIMULATION OF MICROSCOPIC FLUID PARTICLE MOTION ON SOLID SURFACE USING A DIFFUSE-INTERFACE MODEL <i>Takada Naoki, Matsumoto Junichi, Matsumoto Sohei, Kurihara Kazuma, Hokari Ryohei</i>	49/3
<b>Tu 11:30-12:50</b>	<b>Experimental Methods for Multiphase Flows</b> <i>Room: Affari-4.A</i> EXPERIMENTAL STUDY ON SLUG FLOW DEVELOPMENT IN A RISER AND VERTICAL TO HORIZONTAL 90° BEND <i>Omar Rajab, Azzopardi Barry James, Hewakandamby Buddhikha</i>	49/4

	PRESSURE DROP THROUGH ORIFICES FOR SINGLE- AND TWO-PHASE VERTICALLY UPWARDS FLOW	
	<i>Azzi Abdelwahid, Zeghloul Ammar, Saidj Faiza, Messilem Abdelkader, Azzopardi Barry James</i>	49/5
	SOLID VOLUME FRACTION FLUCTUATIONS IN A LIQUID FLUIDIZED BED	
	<i>Marefatallah Maedeh, Sanders Sean</i>	49/6
	CHARACTERIZATION OF DYNAMICS OF THREE-PHASE SLURRY FLOW USING GAS VOLUME FRACTION FLUCTUATION MEASUREMENTS	
	<i>Tyagi Parul, Buwa Vivek</i>	50/1
<b>Tu 11:30-12:50</b>	<b>Industrial Applications</b>	
	<i>Room: Affari-0.A</i>	
	NUMERICAL OPTIMIZATION OF THE PHASE INTERFACE IN HIGH-PRESSURE DIE CASTING	
	<i>Frings Markus, Behr Marek, Elgeti Stefanie</i>	50/2
	AGGLOMERATION AND TRANSPORT OF A POPULATION OF OXIDE INCLUSIONS IN AN ALUMINUM INDUCTION FURNACE	
	<i>Bellot Jean-Pierre, Bansal Akshay, Chapelle Pierre, Waz Emmanuel, Delannoy Yves, Lebrun Pierre</i>	50/3
	SPREADING BEHAVIORS ON MOLTEN-METAL FALLING-FLOW AT ELEVATED TEMPERATURES IN NUCLEAR SEVERE ACCIDENT CONDITIONS	
	<i>Mori Michitsugu, Matsumoto Tatsuki, Ogura Takahito, Miwa Shuichiro</i>	50/4
	NUMERICAL AND EXPERIMENTAL STUDY OF TURBULENT FLOW IN A REAL-SCALE MODEL OF A CONTINUOUS CASTING MOLD	
	<i>Cho Hyun-Jin, Han Sang-Woo, Kang Youn-Bae</i>	50/5
<b>Tu 11:30-12:50</b>	<b>Interfacial Flows</b>	
	<i>Room: Affari-2.A</i>	
	STUDY ON THE REMOVAL OF LOW LOADING HEAVIER PHASE BY LIGHTER PHASE IN MULTIPHASE PIPE	
	<i>Koguna Aminu, Lao Liyun, Yeung Hoi</i>	50/6
	HYDRODYNAMICS OF GAS ENTRAPMENT IN MICROCAVITIES	
	<i>Gomez-Lopez Miguel, Torabi Soroosh, Trinkle Christine, Grana-Otero Jose</i>	51/1
	DYNAMICS OF VERTICAL FALLING FILMS – AN EXPERIMENTAL AND NUMERICAL STUDY	
	<i>Akesjo Anders, Gourdon Mathias, Vamling Lennart, Sasic Srdjan</i>	51/2
	NONLINEAR FORCED WAVES IN A VERTICAL RIVULET FLOW	
	<i>Alekseenko Sergey, Aktershev Sergey, Markovich Dmitriy, Kharlamov Sergey, Bobylev Aleksey</i>	51/3
<b>Tu 11:30-12:50</b>	<b>Modelling of Multiphase Flows</b>	
	<i>Room: Affari-2.D</i>	
	EULER-EULER MODELING OF HYDRODYNAMICS AND MASS-TRANSFER IN BUBBLY FLOWS	
	<i>Rzehak Roland, Kriebitzsch Sebastian, Krepper Eckhard</i>	51/4
	A SUBGRID MODEL FOR LARGE-EDDY SIMULATION OF TURBULENT BUBBLY FLOWS	
	<i>Chaib Ina Fathia, Bellakhal Ghazi, Chahed Jamel</i>	51/5
	DNS ASSISTED MODELING OF BUBBLY FLOWS	
	<i>Tryggvason Gretar, Lu Jiakai, Ma Ming</i>	51/6
	STUDY OF AN INTERFACE SHARPENING METHOD FOR LARGE INTERFACE TRACKING WITHIN A TWO-FLUID MODEL	
	<i>Fleau Solene, Mimouni Stephane, Vincent Stephane</i>	52/1
<b>Tu 11:30-12:50</b>	<b>Multiphase Flow in Heat and Mass Transfer</b>	
	<i>Room: Affari-2.C</i>	
	TWO-PHASE HEAT TRANSFER AND FLUID FLOW WITHIN PLATE HEAT EXCHANGERS	
	<i>Kim Jungho, Solotych Valentin, Donghyeon Lee, Amalfi Raffaele, Thome John</i>	52/2
	HEAT TRANSFER IN FLAT PLATE PULSATING HEAT PIPES WITH ASYMMETRIC CHANNEL DESIGNS	
	<i>Jang Dong Soo, Lee Joo Seong, Kim Dongwoo, Mun Hanseok, Kim Yongchan</i>	52/3
	EXPERIMENTAL INVESTIGATION OF TAYLOR BUBBLE DISSOLUTION IN MILLI-CHANNELS	
	<i>Haghnegahdar Mohammadreza, Boden Stephan, Hampel Uwe</i>	52/4
	MASS TRANSFER FROM SINGLE CARBON DIOXIDE BUBBLES IN WATER-ALCOHOL SOLUTIONS IN A VERTICAL PIPE	
	<i>Hori Yohei, Aoki Jiro, Hayashi Kosuke, Hosokawa Shigeo, Tomiyama Akio</i>	52/5
<b>Tu 11:30-12:50</b>	<b>Particle and Drop Dynamics</b>	
	<i>Room: Villa-Sala Verde</i>	
	AEROSOL SCAVENGING BY DROPS: DERIVING A COLLECTION KERNEL FROM MICROPHYSICAL MODELLING	
	<i>Cherrier Gael, Belut Emmanuel, Taniere Anne, Rimbert Nicolas, Gerardin Fabien</i>	52/6
	ANISOTROPIC TURBULENT CLUSTERING OF CLOUD DROPLETS SETTLING IN ISOTROPIC TURBULENCE	
	<i>Matsuda Keigo, Onishi Ryo, Takahashi Keiko</i>	53/1

	A NEW LIQUID PHASE MODEL FOR DROPLET HEATING AND EVAPORATION BASED ON HILL'S VORTEX	53/2
	<i>Zhou Zhifu, Li Weiyu, Chen Bin, Wang Guo-Xiang</i>	
	INVESTIGATION OF IMPINGING DROPLET SPREADINGS ON HOT/COLD SURFACES WITH TR-PIV	53/3
	<i>Erkan Nejdett, Okamoto Koji</i>	
<b>Tu 11:30-12:50</b>	<b>Particle-Laden Flows</b>	
	<i>Room: Villa-Auditorium</i>	
	LAGRANGIAN SUBGRID MODELLING FOR THE RELATIVE SEPARATION OF TRACERS IN TURBULENCE	53/4
	<i>Toschi Federico, Lanotte Alessandra Sabina, Mazzitelli Irene M.</i>	
	FEEDBACK EFFECT IN GAS-SOLID DISPERSE FLOW THROUGH A VERTICAL CHANNEL	53/5
	<i>Mito Yoichi</i>	
	MACROSCOPIC CHARACTERISTICS OF HEAVY PARTICLE RESUSPENSION OBTAINED FROM DIRECT NUMERICAL SIMULATIONS OF PRESSURE DRIVEN CHANNEL FLOW	53/6
	<i>Yiantsios Stergios, Saliakellis Panagiotis</i>	
	MIXTURE SIMULATIONS TO INVESTIGATE THE EFFECT OF SUSPENDED SEDIMENT ON VORTEX GENERATION OVER A RIPPLED SAND BED	54/1
	<i>Penko A., Calantoni Joseph</i>	
<b>Tu 11:30-12:50</b>	<b>Multiphase Pipe Flows</b>	
	<i>Room: Affari-3.E</i>	
	MEASUREMENT OF LIQUID DROPLETS IN ANNULAR-MIST FLOW USING SHADOWGRAPH TECHNIQUE	54/2
	<i>Andayi Samwel, Wang Yuchen, Hazuku Tatsuya, Takamasa Tomoji</i>	
	THE EFFECT OF SHEAR STRESS FORMULATION AND OF THE SHAPE FACTOR ON SLUG CAPTURING	54/3
	<i>Bonzanini Arianna, Ferrari Marco, Poesio Pietro</i>	
	THE EFFECT OF PHYSICAL PROPERTIES ON CHURN FLOW IN A LARGER DIAMETER RISER	54/4
	<i>Damir Garaev, Escrig Escrig Josep, Azzopardi Barry James, Hewakandamby Buddhikha</i>	
	TURBULENCE CONTRIBUTION IN MOMENTUM INTERFACIAL TRANSFER IN AN UPWARD LIQUID-LIQUID PIPE FLOW	54/5
	<i>Rezig Mariem, Bellakhal Ghazi, Chahed Jamel</i>	
<b>Tu 11:30-12:50</b>	<b>Bubble Dynamics</b>	
	<i>Room: Affari-3.B</i>	
	NUMERICAL SIMULATIONS OF A RISING DROP OR BUBBLE WITH SHAPE OSCILLATIONS IN THE PRESENCE OF SURFACTANTS	54/6
	<i>Piedfert Antoine, Risso Frederic, Masbernat Olivier, Lalanne Benjamin</i>	
	LIFT FORCE ACTING ON SINGLE BUBBLES IN LINEAR SHEAR FLOWS	55/1
	<i>Aoyama Shohei, Zun Iztok, Hayashi Kosuke, Hosokawa Shigeo, Tomiyama Akio</i>	
	THE INTERACTION OF A HIGH-RE MODERATE-WE BUBBLE WITH AN INCLINED WALL	55/2
	<i>Barbosa Christophe, Legendre Dominique, Zenit Roberto</i>	
	A PARALLEL FRONT-TRACKING ALGORITHM FOR THE SIMULATION OF RISING BUBBLES	55/3
	<i>Silveira Neto Aristeu, Villar Millena Martins, Roma Alexandre, Serfaty Ricardo, Van Wachem Berend, Pivello Marcio</i>	

## PARALLEL SESSIONS VII

<b>Tu 14:40-16:00</b>	<b>Bubbly Flows</b>	
	<i>Room: Affari-2.B</i>	
	MEASUREMENT OF BUBBLE DISPERSION IN A TURBULENT BOUNDARY LAYER	55/4
	<i>Gabillet Celine, Bon Alexandre, Billard Jean Yves, Cazin Sebastien, Colin Catherine</i>	
	EFFECT OF DEFORMABILITY ON BUBBLE MIGRATION IN TURBULENT GAS-LIQUID FLOWS	55/5
	<i>Vichansky Alexander</i>	
	FLOW PAST A BUBBLE-GENERATING HYDROFOIL FOR SHIP DRAG REDUCTION	55/6
	<i>Kumagai Ichiro, Tsukahara Takahiro, Murai Yuichi</i>	
	INVESTIGATION OF THE BUBBLE SIZE DISTRIBUTION AROUND AN UNDERWATER VEHICLE	56/1
	<i>Vahaji Sara, Chen Li, Cheung Sherman, Tu Jiyuan</i>	
<b>Tu 14:40-16:00</b>	<b>Boiling, Condensation, Evaporation</b>	
	<i>Room: Affari-3.D</i>	
	VARIATIONS IN GEOMETRIC VARIABLES OF WATER DROPLET ON LOW-SURFACE-ENERGY SOLIDS	56/2
	<i>Yonemoto Yukihiro, Kunugi Tomoaki</i>	

	STRATIFICATION OF INTERMITTENT AND ANNULAR R245-FA FLOW BOILING IN HORIZONTAL ORIENTATION <i>Layssac Thibaut, Revellin Remi, Lips Stephane</i>	56/3
	PREDICTION OF SOLID-LIQUID EQUILIBRIUM IN PARAFFINIC SYSTEMS WITH AN IMPROVED ACTIVITY COEFFICIENT METHOD FOR THE NON-IDEALITY OF SOLID PHASE <i>Yang Juheng, Wang Wei, Song Pan, Ma Qianli, Quan Qing, Gong Jing</i>	56/4
	A MULTISCALE APPROACH TOWARDS IN-TUBE CONVECTIVE BOILING FLOW SIMULATION <i>Yazdani Miad, Alahyari Abbas, Radcliff Thomas, Soteriou Marios</i>	56/5
<b>Tu 14:40-16:00</b>	<b>Cavitation/Nucleation</b> <i>Room: Affari-3.C</i>	
	THE EFFECTS OF NOSE SHAPES AND VISCOSITY ON THE CAVITY EVOLUTION AROUND THE UNDERWATER PROJECTILE <i>Okuno Kohei, Matsuo Akiko</i>	56/6
	LIQUID-FOUNTAIN INSTABILITY AND CAVITATION INSIDE BEADS-FOUNTAIN ASSOCIATED WITH ULTRASONIC ATOMIZATION <i>Myosen Kenta, Ito Yui, Mori Yasushige, Tsuchiya Katsumi</i>	57/1
	INFLUENCE OF SORPTION EFFECTS ON THE EFFICIENCY OF MULTIPHASE CONVEYANCE <i>Wagner Stefan, Luke Andrea</i>	57/2
	EXPERIMENTAL STUDY OF A CAVITATING BACKWARD FACING STEP FLOW: VAPOR PHASE INVESTIGATION USING X-RAY ATTENUATION MEASUREMENTS AND WALL PRESSURE MEASUREMENTS <i>Maurice Guillaume, Djeridi Henda, Barre Stephane</i>	57/3
<b>Tu 14:40-16:00</b>	<b>Collision, Agglomeration and Breakup</b> <i>Room: Affari-3.A</i>	
	DROPLET-PARTICLE COLLISIONS IN HOMOGENEOUS AND ISOTROPIC TURBULENCE <i>Charalampous Georgios, Hardalupas Yannis</i>	57/4
	PARTICLE AND AGGLOMERATE DISPERSION IN GAS FLOW BY COLLISIONS AND FLOW INDUCED STRESSES <i>Wollborn Tobias, Knoop Claas, Fritsching Udo</i>	57/5
	THE PARTICLE COLLISION FREQUENCY IN PARTICLE-LADEN TURBULENT CHANNEL FLOW <i>Kuerten Hans, Vreman Bert</i>	57/6
	THE INFLUENCE OF STOKES NUMBER ON PARTICLE CLUSTERING AT THE EXIT OF A TURBULENT JET ISSUING FROM A LONG PIPE <i>Lau Timothy, Nathan Graham</i>	58/1
<b>Tu 14:40-16:00</b>	<b>Computational Techniques for Multiphase Flows</b> <i>Room: Affari-1.A</i>	
	THE SPARSE CARDINAL SINE DECOMPOSITION APPLIED TO STOKES INTEGRAL EQUATIONS <i>Alouges Francois, Aussal Matthieu, Lefebvre Lepot Aline, Pigeonneau Franck, Sellier Antoine</i>	58/2
	AN ACCURATE METHOD TO TAKE INTO ACCOUNT LUBRICATION FORCES IN NUMERICAL SIMULATION OF SUSPENSIONS <i>Lefebvre Lepot Aline, Chivot Guillaume, Girardin Mathieu, Gouarin Loic, Merlet Benoit</i>	58/3
	NON-CONSERVATIVE PRESSURE-BASED COMPRESSIBLE FORMULATION FOR MULTIPHASE FLOWS WITH HEAT AND MASS TRANSFER <i>Labois Mathieu, Narayanan Chidambaram</i>	58/4
	NUMERICAL SIMULATIONS OF VISCOUS AND COMPRESSIBLE TWO-PHASE FLOWS USING HIGH-ORDER SCHEMES <i>Capuano Marion, Bogey Christophe, Spelt Peter</i>	58/5
<b>Tu 14:40-16:00</b>	<b>Experimental Methods for Multiphase Flows</b> <i>Room: Affari-4.A</i>	
	COMPARISON OF THREE INJECTOR GEOMETRIES IN GAS LIFT PERFORMANCE IN A LARGE DIAMETER PIPE <i>Ibrahim Abubakr, Hewakandamby Buddhikha, Azzopardi Barry James</i>	58/6
	COMPARISON AMONG EXPERIMENTAL TECHNIQUES FOR PARTICLE SIZE MEASUREMENTS IN FLOWS <i>Romano Giampaolo, Pereira Francisco Alves, Van Rijsbergen Martijn, Di Felice Fabio</i>	59/1
	MULTI-CAMERA X-RAY TOMOGRAPHY OF MULTIPHASE FLOWS <i>Skjaeraasen Olaf, Denis Heidi, Langsholt Morten</i>	59/2
	THIN FILM THICKNESS MEASUREMENTS USING ULTRASONIC PULSE ECHO TECHNIQUE <i>Alaoui Yousuf, Watson Nicholas, Azzopardi Barry James, Hewakandamby Buddhikha, Dimitrakis Georgios, Hasan Abbas, Kalashnikov Alexander</i>	59/3

<b>Tu 14:40-16:00</b>	<b>Interfacial Flows</b> <b>Room: Affari-2.A</b> NUMERICAL AND EXPERIMENTAL INVESTIGATION OF LIQUID FILM FLOWS ON PACKINGS IN ABSORBERS FOR POST-COMBUSTION CO <sub>2</sub> CAPTURE <i>Iso Yoshiyuki, Ikeda Ryosuke, Huang Jian, Saga Mariko, Nakamura Shiko, Takano Kenji</i> 59/4 A COMBINED EXPERIMENTAL/COMPUTATIONAL STUDY OF THE HYDRODYNAMIC CHARACTERISTICS OF HARMONICALLY EXCITED THIN-FILM FLOWS <i>Charogiannis Alexandros, Denner Fabian, Van Wachem Berend, Kalliadasis Serafim, Markides Christos N.</i> 59/5 REDUCED ORDER MODELLING OF LIQUID-GAS CHANNEL FLOWS <i>Lavalle Gianluca, Lucquiaud Mathieu, Valluri Prashant</i> 59/6 GROWTH OF A LOCALIZED PERTURBATION IN A COUNTERCURRENT TWO-PHASE FLOW <i>Coppola Gennaro, Capuano Francesco, De Luca Luigi</i> 60/1
<b>Tu 14:40-16:00</b>	<b>Modelling of Multiphase Flows</b> <b>Room: Affari-2.D</b> A HIERARCHY OF COMPRESSIBLE TWO-FLUID MODELS WITH IDENTIFIED PHYSICAL RELAXATIONS AND RELATED NUMERICAL METHODS FOR THE SIMULATION OF SEPARATED AND DISPERSED PHASES <i>Druil Florence, Massot Marc, Kokh Samuel, Larat Adam</i> 60/2 MASSIVE PARALLEL DIRECT NUMERICAL SIMULATIONS OF THREE-DIMENSIONAL JET FLOWS <i>Kahouadji Lyes, Matar Omar K., Chergui Jalel, Juric Damir, Shin Seungwon</i> 60/3 MULTISCALE SIMULATION OF LIQUID JET DISINTEGRATION AND PRIMARY ATOMIZATION USING EULERIAN-LAGRANGIAN COUPLING <i>Saeedipour Mahdi, Pirker Stefan, Schneiderbauer Simon</i> 60/4 DIRECT NUMERICAL SIMULATION OF PRIMARY BREAK-UP IN SWIRLING LIQUID JETS <i>Galbiati Claudio, Tonini Simona, Cossali Gianpietro Elvio, Weigand Bernhard</i> 60/5
<b>Tu 14:40-16:00</b>	<b>Multiphase Flow in Heat and Mass Transfer</b> <b>Room: Affari-2.C</b> LOCAL MASS TRANSFER PHENOMENA AT SINGLE TAYLOR BUBBLES IN A VERTICAL MINI CHANNEL <i>Kastens Sven, Aoki Jiro, Pesci Chiara, Marschall Holger, Hayashi Kosuke, Schlueter Michael, Bothe Dieter, Tomiyama Akio</i> 60/6 STEADY STATE ANALYSIS OF FLASHING DRIVEN NATURAL CIRCULATION LOOP: INFLUENCE OF HEATER AND COOLER ORIENTATION <i>Dewangan Kush Kumar, Das Prasanta Kumar</i> 61/1 HYDRODYNAMICS AND MASS TRANSFER PROPERTIES OF A ROTATING BUBBLE COLUMN <i>Lau Yuk Man, Herrmann Heber Robert, Dashliborun Amir Motamed, Schubert Markus</i> 61/2 MASS TRANSFER OF SOLUTE AND SOLVENT ACROSS A DEFORMING PERMEABLE MEMBRANE <i>Takeuchi Shintaro, Miyauchi Suguru, Kajishima Takeo</i> 61/3
<b>Tu 14:40-16:00</b>	<b>Particle and Drop Dynamics</b> <b>Room: Villa-Sala Verde</b> THE SOLIDIFICATION OF WATER DROPLETS IMPINGING TO A COOLING HORIZONTAL PLATE <i>Kimura Ryota, Hagiwara Yoshimichi</i> 61/4 SLIDING DROPS – MOLECULAR VERSUS CONTINUUM SIMULATIONS <i>Derksen Jos</i> 61/5 EFFECTS OF NANOPARTICLES ON THE DEPINNING FORCE OF A RECEDING DROPLET ON MICROPATTERNED SUPERHYDROPHOBIC SURFACES <i>Jiang Youhua, Xu Wei, Connington Kevin, Choi Chang-Hwan</i> 61/6 DEPINNING OF WATER DROPLETS ON PILLARED SUPERHYDROPHOBIC SURFACES UNDER DYNAMIC ICING CONDITIONS <i>Sarshar Mohammad, Choi Chang-Hwan</i> 62/1
<b>Tu 14:40-16:00</b>	<b>Particle-Laden Flows</b> <b>Room: Villa-Auditorium</b> DIRECT NUMERICAL SIMULATION OF TURBULENT CHANNEL FLOW OVER AN EVOLVING SEDIMENT BED <i>Kidanemariam Aman, Uhlmann Markus</i> 62/2 CHANNEL FLOW OF PARTICLE SUSPENSIONS: MASS AND VOLUME FRACTION EFFECTS <i>Brandt Luca, Lashgari Iman, Picano Francesco, Fornari Walter</i> 62/3 EXPERIMENTS ON INCIPIENT BED LOAD UNDER A TURBULENT STREAM <i>Mendes Penteado Marcos Roberto, De Moraes Franklin Erick</i> 62/4

Tu 14:40-16:00	<b>Multiphase Pipe Flows</b>	
	<i>Room: Affari-3.E</i>	
	A MECHANISTIC MODEL FOR GAS-LIQUID PIPE FLOW WITH SURFACTANTS <i>Van Nimwegen Dries, Van Boven Gijs, Portela Luis, Henkes Ruud</i>	62/5
	NUMERICAL 3-D SIMULATION OF DEVELOPING STRATIFIED GAS- LIQUID FLOW IN CURVED PIPES <i>Haj Ali, Mansoori Zohreh, Saffar Avval Majid, Ahmadi Goodarz</i>	62/6
Tu 14:40-16:00	<b>Bubble Dynamics</b>	
	<i>Room: Affari-3.B</i>	
	MODELLING OF BUBBLE DYNAMICS AND EULER/LAGRANGE PREDICTIONS OF BUBBLE COLUMNS <i>Muniz Marcelo, Sommerfeld Martin</i>	63/2
	THEORETICAL MODEL OF THE BUBBLING REGIME IN A PLANAR CO-FLOW CONFIGURATION <i>Bolanos Jimenez Rocio, Sevilla Alejandro, Bazan Carlos Martinez</i>	63/3
Tu 14:40-16:00	<b>Memorial session dedicated to Prof. Gad Hetsroni</b>	
	<i>Room: Affari-0.A</i>	
	WAKE FEATURES OF BIRDS DURING FLAPPING FLIGHT, CASE STUDY: STARLING, SANDPIPER AND ROBIN <i>Gurka Roi</i>	63/6
	BREAKUP OF FINITE SIZE COLLOIDAL AGGREGATES IN TURBULENT FLOW INVESTIGATED BY 3D PARTICLE TRACKING VELOCIMETRY <i>Babler Matthias, Saha Debashish, Holzner Markus, Soos Miroslav, Luthi Beat, Liberzon Alex</i>	64/1

## PARALLEL SESSIONS VIII

Tu 16:20-17:40	<b>Bubbly Flows</b>	
	<i>Room: Affari-2.B</i>	
	LINEAR VISCOELASTIC ANALYSIS OF BUBBLE SUSPENSIONS BY MEANS OF ULTRASONIC SPINNING RHEOMETRY <i>Tasaka Yuji, Rapberger Richard, Dymowski Konrad, Murai Yuichi</i>	64/2
	THREE-DIMENSIONAL SCENARIO OF DRAG REDUCTION IN BUBBLY TAYLOR-COUETTE FLOWS <i>Murai Yuichi, Ozaki Tsubasa, Tasaka Yuji</i>	64/3
Tu 16:20-17:40	<b>Boiling, Condensation, Evaporation</b>	
	<i>Room: Affari-3.D</i>	
	EFFECT OF SUBCOOLING ON FILM BOILING HEAT TRANSFER CHARACTERISTICS OF JET ON A SUPERHEATED PLATE <i>Hasegawa Koji, Ohtake Hiroyasu, Ueno Masahiro</i>	64/6
	EXPERIMENTAL STUDY OF WATER POOL BOILING ON A SINGLE NUCLEATION SITE, FROM ATMOSPHERIC PRESSURE DOWN TO LOW PRESSURE <i>Michaie Sandra, Rulliere Romuald, Bonjour Jocelyn</i>	65/1
Tu 16:20-17:40	<b>Boiling, Condensation, Evaporation</b>	
	<i>Room: Affari-3.D</i>	
	EFFECT OF WETTABILITY ON POOL BOILING ONSET <i>Marengo Marco, Bourdon Benoit, Villa Fabio, Malavasi Ileana, Rioboo Romain, Bertrand Emilie</i>	65/2
	EFFECT OF OSCILLATORY FLOW FIELD ON BOILING IN SMALL DIAMETER TUBE AT ATMOSPHERIC PRESSURE <i>Kumar Manoj, Das Arup Kumar, Das Prasanta Kumar</i>	65/3



Tu 16:20-17:40	<b>Collision, Agglomeration and Breakup</b> <i>Room: Affari-3.A</i>	
	EXPERIMENTAL VALIDATION OF ANALYTICAL MODELS FOR LIQUID FILM BREAK-UP	65/4
	<i>Line Hakon, Steiner Helfried, Brenn Guenter, Engesser Philipp</i>	
	SPRAY IMPINGEMENT ON SPHERICAL TARGETS IN HOMOGENEOUS, ISOTROPIC	
	TURBULENCE	
	<i>Andrade Paul, Charalampous Georgios, Hardalupas Yannis</i>	65/5
	A STOCHASTIC APPROACH TO COLLOIDAL PARTICLE COLLISION/AGGLOMERATION	
	<i>Mattei Radu Eduard, Bossy Mireille, Minier Jean-Pierre, Profeta Christophe</i>	65/6
	PARTICLE-WALL ADHESION MODEL FOR TURBULENT DISPERSE MULTIPHASE FLOWS	
	WITHIN AN EULER-LAGRANGE LES APPROACH	
	<i>Breuer Michael, Almohammed Naser</i>	66/1
Tu 16:20-17:40	<b>Computational Techniques for Multiphase Flows</b> <i>Room: Affari-1.A</i>	
	COMPUTING COMPLEX MULTIPHASE FLOWS WITH BLUE	
	<i>Juric Damir, Chergui Jalel, Shin Seungwon, Kahouadji Lyes, Matar Omar K.</i>	66/2
	A NEW MIXTURE MODEL FOR COMPRESSIBLE MULTIPHASE FLOWS	
	<i>Henry De Frahan Marc, Alahyari Beig Shahab, Aboulhasanzadeh Bahman, Ganesh Harish, Ceccio</i>	
	<i>Steven L., Johnsen Eric</i>	66/3
	NUMERICAL SIMULATIONS OF SUBCOOLED BOILING USING A TWO-FLUID/DQMOM	
	METHODOLOGY	
	<i>Jareteg Klas, Sasic Srdjan, Vinai Paolo, Demaziere Christophe</i>	66/4
	A NEW HIGH ORDER MOMENT METHOD FOR POLYDISPERSE EVAPORATING SPRAY	
	MODELING AND SIMULATION DEDICATED TO THE COUPLING WITH SEPARATED TWO-	
	PHASE FLOWS IN AUTOMOTIVE ENGINE	
	<i>Essadki Mohamed, De Chaisemartin Stephane, Massot Marc, Laurent Frederique, Larat Adam, Jay</i>	
	<i>Stephane</i>	66/5
Tu 16:20-17:40	<b>Experimental Methods for Multiphase Flows</b> <i>Room: Affari-4.A</i>	
	LDV ANALYSIS OF GAS-LIQUID SLUG FLOW IN A HORIZONTAL PIPE	
	<i>Thaker Jignesh, Banerjee Jyotirmay</i>	66/6
	EXPERIMENTAL ANALYSIS OF DOWNWARD GAS-LIQUID SLUG FLOW IN SLIGHTLY	
	INCLINED PIPES	
	<i>Rodrigues Romulo, Schneider Fabio, Da Silva Marco, Marcelino Neto Moises, Morales Rigoberto</i>	67/1
	DETERMINATION OF HYDRODYNAMIC PARAMETERS OF AIR-WATER SLUG FLOW USING	
	ULTRASONIC TECHNIQUES VIA HILBERT-HUANG TRANSFORMS	
	<i>Abbagoni Baba Musa, Yeung Hoi</i>	67/2
	EXPERIMENTAL RESEARCH OF CONDUCTANCE AND CAPACITANCE COMBINATION	
	SENSOR FOR WATER HOLDUP MEASUREMENT IN OIL-WATER TWO-PHASE FLOW	
	<i>Wu Hao, Tan Chao, Dong Feng</i>	67/3
Tu 16:20-17:40	<b>Granular Media &amp; Granular Flows</b> <i>Room: Affari-3.B</i>	
	COUPLING DIFFERENT LEVELS OF RESOLUTION IN DEM SIMULATIONS	
	<i>Queteschner Daniel, Lichtenegger Thomas, Pirker Stefan</i>	67/4
	CHARACTERIZATION OF NEARLY JAMMED GRANULAR FLOWS IN HIGH SHEAR SYSTEMS	
	THROUGH MOLECULAR DYNAMICS SIMULATIONS	
	<i>Khalilitehrani Mohammad, Sasic Srdjan, Rasmuson Anders</i>	67/5
	ON THE MOTION OF A SINGLE AIR SPOUT THROUGH A QUASI-TWO DIMENSIONAL	
	GRANULAR BED	
	<i>Maiti Ritwik, Das Gargi, Das Prasanta Kumar</i>	67/6
Tu 16:20-17:40	<b>Industrial Applications</b> <i>Room: Affari-3.C</i>	
	EFFECT OF MOLTEN STEEL FLOW IN MOLD ACCORDING TO INTENSITY AND POSITION OF	
	ELECTRO-MAGNETIC FIELD	
	<i>Han Sang-Woo, Cho Hyun-Jin, Jin Seonyong, Seden Martin, Choi Sungmin</i>	68/1
	NUMERICAL ANALYSIS OF EROSION ON SUSPENDED HEAT TRANSFER SURFACES IN A	
	350MW SUPERCRITICAL CFB BOILER	
	<i>Tu Qiuya, Wang Haigang, Sun Yunkai</i>	68/2
	NUMERICAL INVESTIGATION OF ELECTROSTATIC EFFECTS ON PNEUMATIC TRANSPORT	
	OF GRANULAR MATERIALS	
	<i>Yao Jun, Zhao Yanlin, Lim Eldin Wee Chuan</i>	68/3

Tu 16:20-17:40	<b>Interfacial Flows</b>	
	<i>Room: Affari-2.A</i>	
	INTERFACE-RESOLVING SIMULATIONS OF GAS-LIQUID FLOWS IN A SOLID SPONGE STRUCTURE	
	<i>Cai Xuan, Woerner Martin, Deutschmann Olaf</i>	68/4
	INFLUENCE OF ANGULAR MOMENTUM ON THE FORMATION OF AIR-CORE INTAKE	
	VORTICES IN CYLINDRICAL WATER TANKS	
	<i>Szeliga Nicolai, Bezecny Daniel, Richter Steffen, Hoffmann Marko, Schlueter Michael</i>	68/5
	INFLUENCE OF PARTIALLY WETTED MICRO-PARTICLES AT A CONTAMINATED INTERFACE ON COALESCENCE PROCESS.	
	<i>De Malmazet Erik, Risso Frederic, Masbernat Olivier, Pauchard Vincent</i>	68/6
Tu 16:20-17:40	<b>Modelling of Multiphase Flows</b>	
	<i>Room: Affari-2.D</i>	
	NUMERICAL STUDY OF DROPLET STATISTICS RESULTING FROM TURBULENT JET PRIMARY BREAKUP	
	<i>Movaghar Amirreza, Linne Mark, Oevermann Michael, Bravo Luis, Kerstein Alan</i>	69/1
	PARTICLE DISPERSION IN INHOMOGENEOUS TURBULENCE: AN ANALYSIS OF PDF-BASED MODELS	
	<i>Stafford Christopher, Swailes David, Chakraborty Nilanjan</i>	69/2
	3D RSTM MODELING FOR PARTICULATE GRID-GENERATED TURBULENT AND SHEAR FLOWS	
	<i>Kartushinsky Alexander, Hussainov Medhat, Shcheglov Igor, Tisler Sergei, Kosinski Pawel, Balakin Boris V., Lauk Peep, Seegel Karl Erik</i>	69/3
	LAGRANGIAN STOCHASTIC MODELLING IN LARGE-EDDY SIMULATION OF TURBULENT PARTICLE-LADEN FLOWS	
	<i>Innocenti Alessio, Marchioli Cristian, Chibbaro Sergio</i>	69/4
Tu 16:20-17:40	<b>Multiphase Flow in Heat and Mass Transfer</b>	
	<i>Room: Affari-2.C</i>	
	COLLECTING DEVICE OF VOLATILE ORGANIC COMPOUNDS IN WATER LIKE A NOSE: THIN LIQUID FILM CONTAINING ODORANT BINDING PROTEINS	
	<i>Kida Hitoshi, Tagawa Yoshiyuki, Sato Ryoichi, Kameda Masaharu</i>	69/5
	MULTIPHASE INVESTIGATION OF INTENSIVE QUENCHING WITH WATERJETS	
	<i>Waldeck Steffen, Fritsching Udo</i>	69/6
	A SIX-EQUATION TWO-PHASE MODEL IN WENO SCHEME FOR PHASE TRANSFORMATION FRONTS	
	<i>Wu Wangxia, Wang Bing, Zhang Wenbin</i>	70/1
	EFFECT OF THE VAPOR FLOW ON THE DROP SPREADING AT THE IMPACT IN THE LEIDENFROST BOILING	
	<i>Castanet Guillaume, Caballina Ophelie, Lemoine Fabrice</i>	70/2
Tu 16:20-17:40	<b>Particle and Drop Dynamics</b>	
	<i>Room: Villa-Sala Verde</i>	
	DROP IN DROP DIAMETER FROM A VIBRATING CAPILLARY TUBE	
	<i>Duval Herve, Bertrandias Aude, Giorgi Marie-Laurence</i>	70/3
	KINEMATIC AND DYNAMIC CHARACTERISTICS OF A FALLING DROPLET	
	<i>Bergeles Konstantinos, Hardalupas Yannis, Taylor Alex M.K. P.</i>	70/4
	WALL EFFECT ON THE DYNAMICS OF PARTICLE MOTION IN A LAMINAR FLOW	
	<i>Feng Zhi-Gang, Gatewood Jason, Duan Yifei, Michaelides Efstathios</i>	70/5
	STEERING ULTRASOUND CONTRAST AGENTS WITH ULTRASOUND USING THE PRIMARY BJERKNES FORCE	
	<i>Clark Alicia, Aliseda Alberto</i>	70/6
Tu 16:20-17:40	<b>Particle-Laden Flows</b>	
	<i>Room: Villa-Auditorium</i>	
	PARTICLE RESUSPENSION BY A PERIODICALLY-FORCED IMPINGING JET	
	<i>Soligo Giovanni, Wu Wen, Zonta Francesco, Marchioli Cristian, Soldati Alfredo, Piomelli Ugo</i>	71/1
	SEDIMENT-LADEN FRESH WATER ABOVE SALT WATER: STABILITY ANALYSIS AND NONLINEAR SIMULATIONS	
	<i>Meiburg Eckart, Burns Peter</i>	71/2
	MODIFICATION OF WALL TURBULENCE VIA THE EXACT REGULARIZED POINT PARTICLE APPROACH	
	<i>Messina Roberta, Battista Francesco, Gualtieri Paolo, Casciola Carlo Massimo</i>	71/3
Tu 16:20-17:40	<b>Memorial session dedicated to Prof. Gad Hetsroni</b>	
	<i>Room: Affari-0.A</i>	
	PARTICLE FLOW IN GAS OR LIQUID - WHAT DO THEY HAVE IN COMMON?	
	<i>Kalman Haim</i>	71/4

SCAVENGING OF SOLUBLE ATMOSPHERIC TRACE GASES DUE TO CHEMICAL ABSORPTION BY RAIN DROPLETS IN INHOMOGENEOUS ATMOSPHERE <i>Elperin Tov, Fominykh Andrew, Krasovikov Boris</i>	71/5
AN UNIVERSAL BINARY FORCE RELATION ON A PARTICLE THAT ACCOUNTS FOR THE INFLUENCE OF NEAREST NEIGHBORS <i>Akiki Georges, Balachandar S, Jackson Thomas L., Mehta Yash, Neal Christopher R., Thakur        Siddarth</i>	71/6
DNS OF TURBULENT CONVECTIVE FLOW BOILING IN A CHANNEL AT $Re=400$ <i>Caviezel Daniel, Swiderski Karol, Narayanan Chidambaram, Lakehal Djamel</i>	72/1

## WEDNESDAY, May 25, 2016

### PARALLEL SESSIONS IX

We 9:50-11:10	<b>Bubbly Flows</b>	
	<i>Room: Affari-2.B</i>	
	EXPERIMENTAL INVESTIGATIONS ON THE LIFT FORCE FOR TURBULENT FLOWS WITH LOW MORTON NUMBER	
	<i>Ziegenhein Thomas, Lucas Dirk</i>	72/2
	STUDY OF BUBBLY FLOW THROUGH A PERFORATED PLATE USING A TWO-FLUID MODEL AND A DISCRETE ELEMENT METHOD	
We 9:50-11:10	<i>Pena Monferrer Carlos, Martinez Cuenca Raul, Munoz Cobo Gonzalez Jose Luis, Chiva Sergio</i>	72/3
	MOTION OF SURFACTANT CONTAMINATED BUBBLES IN HOMOGENEOUS SHEAR TURBULENCE	
	<i>Tanaka Mitsuru</i>	72/4
	TRANSPORT AND HYDRODYNAMIC PHENOMENON IN A VIBRATING BUBBLE COLUMN	
	<i>Mohagheghian Shahrouz, Elbing Brian. R</i>	72/5
We 9:50-11:10	<b>Boiling, Condensation, Evaporation</b>	
	<i>Room: Affari-3.D</i>	
	VISUALIZATION STUDIES ON THE SPRAY COOLING BEHAVIOR OF THE MICROMETRIC MULTIPLE SUCCESSIVE DROPLETS ABOVE THE LEIDENFROST TEMPERATURE IMPACTING ONTO HOT INCLINED SURFACES	
	<i>Deendarlianto Deendarlianto, Takata Yasuyuki, Kohno Masamichi, Hidaka Sumitomo, Wakui Takaaki, Indarto Indarto</i>	72/6
	IMPLEMENTATION AND DEMONSTRATION OF A BOUNDARY CONDITION WALL FUNCTION FOR DIRECT PRECIPITATION FOULING CFD MODELLING	
We 9:50-11:10	<i>Johnsen Sverre Gullikstad, Johansen Stein Tore, Wittgens Bernd</i>	73/1
	3D SIMULATION OF NUCLEATE POOL BOILING USING DIFFUSE INTERFACE MODEL	
	<i>Tanimoto Takuma, Tsujimoto Koichi, Shakouchi Toshihiko, Ando Toshitake</i>	73/2
	MEASUREMENTS AND VISUALIZATION DURING CONDENSATION INSIDE A 3 MM DIAMETER CHANNEL	
	<i>Azzolin Marco, Bortolin Stefano, Del Col Davide</i>	73/3
We 9:50-11:10	<b>Granular Media &amp; Granular flow</b>	
	<i>Room: Affari-3.B</i>	
	EULERIAN SIMULATION OF 3D FREELY-FALLING GRANULAR JET	
	<i>Audard Francois, Fede Pascal, Belut Emmanuel, Fontaine Jean-Raymond, Simonin Olivier</i>	73/4
	THE PROPAGATION AND DEPOSITION OF A FINITE DRY GRANULAR MASS DOWN A ROUGH INCLINE	
We 9:50-11:10	<i>Lee Keng Lin, Yang Fuling</i>	73/5
	COLLAPSE OF A WATER-SATURATED GRANULAR COLUMN IN AIR	
	<i>Aussillous Pascale, Nobili Clement</i>	73/6
	DIRECT AND INDIRECT MEASUREMENTS OF FRICTION COEFFICIENTS OF STEADY DRY GRANULAR FLOWS DOWN A ROUGH NARROW INCLINE OF ROUGH BED	
	<i>Yang Fuling, Tingyen Chiu</i>	74/1
We 9:50-11:10	<b>Industrial Applications</b>	
	<i>Room: Affari-1.A</i>	
	STUDY OF THE PROCESS OF ELECTROMAGNETIC IMPACT ON HIGH-VISCOSITY HYDROCARBON LIQUID IN A PIPELINE	
	<i>Kireev Victor, Musin Ayrat, Zinnatullin Rasul, Miroshnichenko Vladislav</i>	74/2
	PERFORMANCE OF HEAVY OIL VISCOSITY REDUCER (HOVR) ON CRUDE OIL EMULSION	
We 9:50-11:10	<i>Bonetti Darlan, Biazussi Jorge Luiz, Perles Carlos Eduardo, Monte Verde William, Guersoni Bizotto Vanessa Cristina, Bannwart Antonio Carlos</i>	74/3
	CFD-PBM MODELLING OF AGGLOMERATION AND DEPOSITION IN PETROLEUM INDUSTRY	
	<i>Balakin Boris V., Kartushinsky Alexander, Hoffmann Alex Christian, Kosinski Pawel</i>	74/4
	A STUDY OF THE PERFORMANCE OF PILOT SCALE GRAVITY SORTERS USING A CFD-DEM COMPUTATIONAL FRAMEWORK	
	<i>Kannan Ananda Subramani, Lund Jacob, Carstensen Jens Michael, Sasic Srdjan</i>	74/5
We 9:50-11:10	<b>Interfacial Flows</b>	
	<i>Room: Affari-2.A</i>	
	A COMPARATIVE STUDY OF TEMPORAL EVOLUTION OF AIR AND STEAM BUBBLES EMERGING OUT FROM AN ORIFICE IN A QUIESCENT WATER MEDIA	
	<i>Ahmad.S Thariq, Datta Saikat, Singha Sanat Kumar, Datta Santanu Prasad, Das Prasanta Kumar</i>	74/6

	PHASE-FIELD SIMULATIONS OF THE BUOYANCY-DRIVEN DETACHMENT OF A WALL-BOUND PENDANT DROP <i>Lamorgese Andrea, Mauri Roberto</i>	75/1
	TRIBO-CHARGING AND FLAME-SYNTHESIS TO CREATE SUPER-HYDROPHOBIC COATINGS WITH STRONG WATER JET RESISTANCE <i>Bayer Ilker, Loth Eric</i>	75/2
	VALIDATION OF TURBULENCE PARAMETERS AT THE INTERFACE OF HORIZONTAL MULTIPHASE FLOWS <i>Hoehne Thomas</i>	75/3
<b>We 9:50-11:10</b>	<b>Mixing</b> <b>Room: Affari-3.A</b> CFD SIMULATIONS OF A STIRRED-TANK REACTOR FOR MULTIPHASE SYSTEMS USING OPENFOAM <i>Hu Xiaofei, Passalacqua Alberto, Fox Rodney, Bertola Francesco</i>	75/4
	SCALE-UP OF MIXING PROPERTIES IN BUBBLE COLUMN REACTORS <i>Plais Cecile, Gemello Luca, Raimundo Pedro Maximiano, Forret Ann, Marchisio Daniele, Augier Frederic</i>	75/5
	INVESTIGATION OF LOCAL MIXING AND BUBBLE SIZE DISTRIBUTION IN A STIRRED BUBBLE COLUMN REACTOR <i>Rosseburg Annika, Schaefer Jan Erik, Wucherpfennig Thomas, Berger Martina, Schlueter Michael</i>	75/6
	STUDY OF HYDRODYNAMICS AND MIXING INDUCED BY BUBBLE SWARMS IN HIGHLY VISCOUS FLUIDS <i>Laupsien David, Le Men Claude, Line Alain, Cockx Arnaud</i>	76/1
<b>We9:50-11:10</b>	<b>Modelling of Multiphase Flows</b> <b>Room: Affari-2.D</b> DIRECT NUMERICAL SIMULATION OF FLOW AROUND A ROTATING SPHERE AT HIGH MACH AND LOW REYNOLDS NUMBERS CONDITION FOR SUPERSONIC GAS-PARTICLE FLOW ANALYSIS <i>Nagata Takayuki, Nonomura Taku, Takahashi Shun, Mizuno Yusuke, Fukuda Kota</i>	76/2
	A PSEUDO-DRY WET CONTACT MODEL FOR DISCRETE ELEMENT SIMULATION OF IMMERSED PARTICLE-WALL COLLISIONS <i>Lin Cheng Chuan, Yang Fuling</i>	76/3
	A STOCHASTIC MODEL FOR THE PREDICTION OF PARTICLE/PARTICLE INTERACTIONS IN THE FRAME OF THE EULER-LAGRANGE APPROACH <i>Arcen Boris, Taniere Anne</i>	76/4
	ANALYSIS OF SUBFILTER EFFECTS ON INERTIAL PARTICLES IN FORCED ISOTROPIC TURBULENCE <i>Rosa Bogdan, Pozorski Jacek</i>	76/5
<b>We 9:50-11:10</b>	<b>Multiphase Flow in Heat and Mass Transfer</b> <b>Room: Affari-2.C</b> NUMERICAL SIMULATION OF A SETTLING ICE BALL WITH COUPLED HEAT AND MASS TRANSFER <i>Chouippe Agathe, Stumpf Michael, Uhlmann Markus, Dusek Jan, Kiselev Alexei, Leisner Thomas</i>	76/6
	EXPERIMENTAL ANALYSIS ON THE CHARACTERISTIC AND MECHANISM OF THE SECOND DOMINANT FREQUENCY OF SUPERSONIC STEAM JET <i>Chen Weixiong, Wang Yingchun, Zhao Quanbin, Chong Daotong, Yan Junjie</i>	77/1
	ENHANCED RECOVERY OF URANIUM(VI) AND EUROPIUM(III) USING IONIC LIQUIDS IN SMALL CHANNEL <i>Li Qi, Tsaoulidis Dimitrios, Angeli Panagiota</i>	77/2
	SPATIOTEMPORALLY RESOLVED HEAT TRANSFER MEASUREMENTS IN WAVY LIQUID FILM-FLOWS FALLING OVER AN INCLINED HEATED-SUBSTRATE <i>Markides Christos N., Baptiste Heiles, Mathie Richard, Charogiannis Alexandros</i>	77/3
<b>We 9:50-11:10</b>	<b>Non-Newtonian Multiphase Flows</b> <b>Room: Affari-3.C</b> VISCOELASTIC FLOW MODELLING FOR POLYMER FLOODING <i>De Shauvik, Peters Frank, Padding Johan, Kuipers J. A. M (hans)</i>	77/4
	EXPLORING THE RHEOLOGICAL BEHAVIOUR OF ATHERMAL SUSPENSIONS OF ATTRACTIVE MICROPARTICLES <i>Murphy Eric, Lomboy Gilson, Sundararajan Sriram, Wang Kejin, Subramaniam Shankar</i>	77/5
	ATOMIZATION OF NON-NEWTONIAN FLUIDS USING AIRLESS SPRAYERS: EFFECT OF VISCOUS DAMPING AND SHEAR THINNING ON LIQUID SHEET INSTABILITY AND ITS BREAKUP <i>Watanabe Hiroki, Fujimoto Shuhei, Ogata Masayuki, Kumagai Ichiro</i>	77/6
	COALESCENCE DURING BUBBLE FORMATION FROM TWO NEIGHBOURING PORES IN A NON-NEWTONIAN LIQUID <i>Oikonomou George, Passos Agathoklis, Mouza Aikaterini, Paras Spiros</i>	78/1

<b>We 9:50-11:10</b>	<b>Particle and Drop Dynamics</b>	
	<i>Room: Villa-Sala Verde</i>	
	SIMULATION OF GASIFYING PARTICLE, INTERACTION WITH FREE INTERFACE FLUID FLOW	78/2
	<i>Moradi Nour Zeinab, Do Quang Minh, Amberg Gustav</i>	
	A WALL-FUNCTION MODEL FOR NON-BROWNIAN HYDROSOL PARTICLES DEPOSITION	78/3
	<i>Dupuy Magali, Xayasenh Arunvady, Waz Emmanuel, Duval Herve</i>	
	BREAK UP OF CAPILLARY BRIDGES: EFFECT OF SOLUBLE SURFACTANT	78/4
	<i>Kovalchuk Nina, Nowak Emilia, Simmons Mark</i>	
	DEFORMATION AND DYNAMICS OF COMPOUND CAPSULES IN SIMPLE SHEAR FLOW	78/5
	<i>Luo Zhengyuan, Bai Bofeng</i>	

<b>We 9:50-11:10</b>	<b>Particle-Laden Flows</b>	
	<i>Room: Villa-Auditorium</i>	
	TOWARDS STOCHASTIC MODELLING OF INERTIAL PARTICLE DISPERSION IN WALL-BOUNDED TURBULENT FLOWS	78/6
	<i>Pozorski Jacek, Knorps Maria</i>	
	EXPERIMENTAL AND NUMERICAL INVESTIGATION OF A PULSATED EVAPORATING SPRAY	79/1
	<i>Brandle De Motta Jorge Cesar, Robbes Benjamin, Shum Kivan Francis, Boileau Matthieu, Massot Marc, Riber Eleonore</i>	
	TURBOPHORESIS; THE TURBULENT SOROT EFFECT	79/2
	<i>Haugen Nils Erland L., Mitra Dhrubaditya, Rogachevskii Igor</i>	
	THE INFLUENCE OF THE REYNOLDS NUMBER AND THE PARTICLE SIZE ON PARTICLE-LADEN TURBULENT CHANNEL FLOW IN THE DENSE REGIME	79/3
	<i>Costa Pedro, Picano Francesco, Brandt Luca, Breugem Wim-Paul</i>	

<b>We 9:50-11:10</b>	<b>Memorial session dedicated to Prof. Gad Hetsroni</b>	
	<i>Room: Affari-0.A</i>	
	TRANSIENT CHANGES OF THE LIQUID LEVEL IN STRATIFIED FLOW ALONG THE PIPE	79/4
	<i>Barnea Dvora, Taitel Yehuda</i>	
	EXPERIMENTAL INVESTIGATION OF THE INTERACTION BETWEEN A STATIONARY RIGID SPHERE AND A TURBULENT BOUNDARY LAYER	79/5
	<i>Van Hout Rene, Eisma Jerke, Overmars Edwin, Elsinga Gerrit, Westerweel Jerry</i>	
	BUBBLE DYNAMICS AT A NON-FLAT WALL	79/6
	<i>Shmueli Havatzelet, Letan Ruth, Ziskind Gennady</i>	

## PARALLEL SESSIONS X

<b>We 11:30-12:50</b>	<b>Bubbly Flows</b>	
	<i>Room: Affari-2.B</i>	
	EFFECTS OF GRAVITY AND ITS ORIENTATION OF BUBBLE DEPARTURE	80/1
	<i>Di Marco Paolo, Morganti Nicolo, Saccone Giacomo</i>	
	FLOW AND SHEAR INDUCED BY BUBBLES RISING THROUGH HOLLOW-FIBER MEMBRANES IN A PARALLEL ARRANGEMENT	80/2
	<i>Tsuchiya Katsumi, Yamada Saori, Umemura Naoki, Mori Yasushige</i>	
	BUBBLY FREE AND IMPINGING FORCED JETS: EXPERIMENTAL STUDY BY PIV/PFBI/PTV	80/3
	<i>Pervunin Konstantin, Timoshevskiy Mikhail, Markovich Dmitriy</i>	
	HYDRODYNAMIC STUDIES OF CUTTING BUBBLES IN MICRO-STRUCTURED SQUARE BUBBLE COLUMN USING ULTRAFast X-RAY TOMOGRAPHY	80/4
	<i>Thiruvalluvan Sujatha Krushnathej, Kuipers J. A. M (hans), Lau Yuk Man, Barthel Frank, Hampel Uwe, Schubert Markus, Deen Niels G.</i>	

<b>We 11:30-12:50</b>	<b>Boiling, Condensation, Evaporation</b>	
	<i>Room: Affari-3.D</i>	
	SUBCOOLED FLOW BOILING EXPERIMENTS: TOWARDS MODELLING THE NEAR-WALL BEHAVIOUR	80/5
	<i>Estrada Perez Carlos E., Yoo Junsoo, Hassan Yassin A.</i>	
	BOILING HEAT TRANSFER TO IMMISCIBLE MIXTURES FOR ADVANCED COOLING OF SEMICONDUCTORS	80/6
	<i>Ohta Haruhiko, Iwata Keisuke, Yamamoto Daisuke, Okayama Satoshi, Shinmoto Yasuhisa</i>	
	POOL NUCLEATE BOILING FOR SEAWATER CONTAINING MINERALS	81/1
	<i>Uesawa Shinichiro, Koizumi Yasuo, Yoshida Hiroyuki</i>	

<b>We 11:30-12:50</b>	<b>Droplet Flows</b>	
	<i>Room: Affari-0.A</i>	
	CFD ANALYSIS OF OIL-WATER TWO-PHASE FLOW AT FLOW REGIME TRANSITION IN HORIZONTAL PIPES	81/2
	<i>Burlutskii Evgenii, Morvan Herve, Kumar Kiran</i>	

	OIL-WATER PIPE FLOW DEVELOPMENT AFTER A VALVE <i>Schumann Heiner, Chandra Pankaj, Nydal Ole Jørgen</i>	81/3
	EXPERIMENTAL INVESTIGATION AND MODELLING OF DISPERSED OIL-WATER PIPE FLOW <i>Hernandez Rodriguez Iara, Hernandez Rodriguez Oscar Mauricio</i>	81/4
	THE DISPERSED OIL-WATER FLOW VELOCITY MEASUREMENT BASED ON CONTINUOUS- WAVE ULTRASONIC DOPPLER <i>Dong Xiaoxiao, Tan Chao, Yuan Ye, Dong Feng</i>	81/5
<b>We 11:30-12:50</b>	<b>Experimental Methods for Multiphase Flows</b> <b>Room: Affari-3.C</b> NEW GAS HOLDUP EXPERIMENTAL DATA IN LARGE COUNTER-CURRENT BUBBLE COLUMNS <i>De Guido Giorgia, Besagni Giorgio, Inzoli Fabio, Pellegrini Laura</i>	81/6
	ULTRASOUND REFLECTOR RECOGNITION AND TRACKING TECHNIQUE FOR GAS BUBBLES IN LIQUID <i>Povolny Antonin, Kikura Hiroshige, Ihara Tomonori</i>	82/1
	FLOW REGIME AND MASS TRANSFER STUDY IN STRUCTURED PIPE REACTORS USING ELECTROCHEMICAL MICRO-PROBES <i>Alzheimer Marco, Becker Dustin, Rudolf von Rohr Philipp</i>	82/2
	TOMOGRAPHIC INTERROGATION OF GAS-LIQUID FLOWS IN INCLINED RISERS <i>Abdulkareem Lokman A., Escrig Escrig Josep, Reinecke Sebastian, Hewakandamby Buddhikha,              Azzopardi Barry James</i>	82/3
<b>We 11:30-12:50</b>	<b>Granular Media &amp; Granular flow</b> <b>Room: Affari-3.B</b> DEVELOPMENT OF A DISCRETE GRAIN BREAKAGE MODEL TO PREDICT EROSION MECHANISM <i>Bharadwaj Rahul, Ropelato Karoline, Potapov Alexander, Nelson Brent</i>	82/4
	THE EFFECT OF AIR PRESSURE GRADIENTS ON THE DISCHARGE OF GRANULAR MATERIALS FROM A SILO <i>Zhou Yixian, Aussillous Pascale, Ruyer Pierre</i>	82/5
	DENSE GRANULAR FLOW IN A BUBBLING FLUIDIZED BED AS A VISCOPLASTIC FLUID <i>Sasic Srdjan, Jareteg Klas, Strom Henrik</i>	82/6
	NUMERICAL STUDY OF MULTI-SIZED GRANULAR PARTICLES IN ROTATING TUMBLER <i>Zhang Ziwei, Ge Liang, Gui Nan, Li Zhenlin</i>	83/1
<b>We 11:30-12:50</b>	<b>Industrial Applications</b> <b>Room: Affari-1.A</b> A TWO-DIMENSIONAL CFD MODEL FOR GAS-LIQUID FLOWS IN A COIL-WOUND HEAT EXCHANGER <i>Acher Thomas, Knaup Manuel, Goll Stephan, Zander Hans-Jorg, Lenz Stefan, Kerber Christiane</i>	83/2
	SAND TRANSPORT IN SUBSEA MULTIPHASE FLOW PIPELINES <i>Yang Yan, Wang Shuli, Wen Chuang</i>	83/3
	CFD MODELLING OF GRAVITY SEPARATORS USING PID CONTROLLER <i>Vani Manoj, Das Saurish, Puneekar Hemant, Mohan Srinivasa</i>	83/4
	COMPREHENSIVE STUDY OF ELECTRO-CHLORINATION PROCESS IN AN UNDIVIDED CELL: A COMPUTATIONAL FLUID DYNAMICS APPROACH <i>Momeni Maryam, Grann Helge, Dooleweerd Karin</i>	83/5
<b>We 11:30-12:50</b>	<b>Interfacial Flows</b> <b>Room: Affari-2.A</b> EXPERIMENTS FOR CORE-ANNULAR FLOW (TO STUDY THE LEVITATION MECHANISM AND THE EFFECT OF BENDS) <i>Ingen Housz Eduard, Ooms Gijsbert, Henkes Ruud, Pourquoi Mathieu, Kidess Anton</i>	83/6
	AN IMPROVED SHARP INTERFACE METHOD FOR SPH <i>Zhang Mingyu, Deng Xiaolong</i>	84/1
	EXPERIMENTAL STUDY ON THE DEVELOPMENT OF SURFACE GEOMETRY UNDER FINGERING INSTABILITY <i>Ito Takahiro, Yamashita Masatoshi, Nakagawa Kazuyuki, Tsuji Yoshiyuki</i>	84/2
	GROWTH AND SPECTRA OF GRAVITY-CAPILLARY WAVES IN COUNTERCURRENT AIR/WATER TURBULENT FLOW <i>Zonta Francesco, Onorato Miguel, Soldati Alfredo</i>	84/3
<b>We 11:30-12:50</b>	<b>Micro- and Nano-Scale Multiphase Flows</b> <b>Room: Affari-2.C</b> EXPERIMENTAL INVESTIGATION OF DROPLET GENERATION IN A MICROFLUIDIC T- JUNCTION USING NEWTONIAN NANOFLUIDS <i>Loizou Katerina, Lester Edward, Hewakandamby Buddhikha</i>	84/4

	CFD STUDY ON DROPLET FORMATION IN NON-NEWTONIAN FLUIDS FLOWING THROUGH T-JUNCTION MICROCHANNEL <i>Atta Arnab, Sontti Somasekhara Goud</i>	84/5
	EFFECT OF THE LUBRICATION FILM DYNAMICS ON THE DROPLET MOTION IN A HELE-SHAW MICROCHANNEL <i>Fullana Jose-Maria, Ling Yue, Popinet Stephane, Josserand Christophe</i>	84/6
	BREAKUP OF DROPLETS IN BRANCHING MICRO-CHANNELS <i>Portela Luis, Haringa Cees, Jong Conrad, Hoang Duong, Kleijn Chris, Kreutzer Michiel, Van Steijn Volkert</i>	85/1
<b>We 11:30-12:50</b>	<b>Mixing</b> <b>Room: Affari-3.A</b> PARTICLE DISTRIBUTION IN UNBAFFLED STIRRED VESSELS <i>Tamburini Alessandro, Cipollina Andrea, Scargiali Francesca, Micale Giorgio, Ciofalo Michele, Brucato Alberto</i>	85/2
	EXPERIMENTAL ANALYSIS OF GAS-LIQUID DISPERSION IN MECHANICALLY STIRRED TANK BY MEANS OF QUANTITATIVE ELECTRICAL RESISTANCE TOMOGRAPHY <i>Busciglio Antonio, Montante Giuseppina, Paglianti Alessandro</i>	85/3
	A CONDITIONAL QUADRATURE METHOD OF MOMENTS FOR TURBULENT MIXING COUPLED TO A POPULATION BALANCE EQUATION <i>Passalacqua Alberto, Madadi Kandjani Ehsan, Heylmun Jeffrey, Fox Rodney</i>	85/4
	COMPARISON BETWEEN DIFFERENT METHODS FOR THE SOLUTION OF THE BIVARIATE POPULATION BALANCE EQUATION <i>Buffo Antonio, Ville Alopaeus</i>	85/5
<b>We 11:30-12:50</b>	<b>Modelling of Multiphase Flows</b> <b>Room: Affari-2.D</b> MODELLING OF THE FLOW REGIME TRANSITION WITH THE SMOOTHED PARTICLE HYDRODYNAMICS <i>Olejnik Michal, Szewc Kamil, Pozorski Jacek</i>	85/6
	A COMPARATIVE ASSESSMENT OF LATTICE BOLTZMANN AND VOLUME OF FLUID (VOF) APPROACHES FOR GENERIC MULTIPHASE PROBLEMS <i>Mukherjee Siddhartha, Zarghami Ahad, Haringa Cees, Kenjeres Sasa, Van Den Akker Harry</i>	86/1
	ACCURATE MODELING OF MOVING CONTACT LINE IN TWO-PHASE IMMISCIBLE FLOWS <i>Holmgren Hanna, Kreiss Gunilla, Kronbichler Martin, Sticko Simon</i>	86/2
	EULERIAN TRAJECTORY METHOD AND IMPINGEMENT MODEL FOR NON-SPHERICAL ICE CRYSTALS <i>Norde Ellen, Van Der Weide Edwin, Hoeijmakers Harry</i>	86/3
<b>We 11:30-12:50</b>	<b>Particle and Drop Dynamics</b> <b>Room: Villa-Sala Verde</b> SINGLE ELLIPSOID IN SIMPLE SHEAR FLOW: INFLUENCE OF INERTIA <i>Rosen Tomas, Kotsubo Yusuke, Do Quang Minh, Aidun Cyrus K., Lundell Fredrik</i>	86/4
	GAS ENTRAINMENT DUE TO A PLUNGING LIQUID JET IN FREE-SURFACE DOWNWARD ANNULAR-CHANNEL FLOW <i>Cavallini Barbosa Marcel, Ortiz Vidal Luis Enrique, Hernandez Rodriguez Oscar Mauricio</i>	86/5
	INTRODUCING A MULTISCALE MODELLING TOOL FOR THE SIMULATION OF POLYURETHANE FOAMS <i>Ferkl Pavel, Karimi Mohsen, Marchisio Daniele, Kosek Juraj</i>	86/6
	MODELING OF THE TRANSIENT TERRAIN-INDUCED AND SEVERE SLUGGING PROBLEMS USING THE DRIFT-FLUX MODEL <i>Sinkov Konstantin, Kharlashkin Andrey, Spesivtsev Pavel</i>	87/1
<b>We 11:30-12:50</b>	<b>Particle-Laden Flows</b> <b>Room: Villa-Auditorium</b> VOLUME-AVERAGED EQUATIONS FOR DIRECT NUMERICAL SIMULATION OF PARTICLE-DISPERSED FLOW UNDER A GRID RESOLUTION COMPARABLE TO PARTICLE DIAMETER AND KOLMOGOROV LENGTH SCALE <i>Fukada Toshiaki, Takeuchi Shintaro, Kajishima Takeo</i>	87/2
	MODELING AND PREDICTION OF GAS-SOLID TURBULENT MULTIPHASE FLOWS USING LAGRANGIAN AND EULERIAN APPROACHES <i>Dunn Dennis, Squires Kyle</i>	87/3
	FULLY EULERIAN SIMULATION OF 3D TURBULENT PARTICLE LADEN FLOW BASED ON THE ANISOTROPIC GAUSSIAN CLOSURE <i>Sabat Macole, Vie Aymeric, Larat Adam, Massot Marc</i>	87/4
	A SPATIAL PARTICLE CORRELATION-FUNCTION ANALYSIS IN NON-ISOTHERMAL DILUTE PARTICLE-LADEN TURBULENT FLOWS <i>Masi Enrica, Fede Pascal, Simonin Olivier</i>	87/5



## PARALLEL SESSIONS XI

We 14:40-16:00	<b>Bubbly Flows</b>	
	<i>Room: Affari-2.B</i>	
	MODELING OF MACROSCALE SCALAR TRANSPORT IN BUBBLY SUSPENSIONS	
	<i>Spelt Peter, Loisy Aurore, Naso Aurore</i>	87/6
	THE IMPACT LOAD AND THE POWER DENSITY DUE TO BUBBLE COLLAPSE IN 1D BUBBLY FLOWS AS INDICATORS FOR A MATERIAL EROSION MODEL	
We 14:40-16:00	<b>Boiling, Condensation, Evaporation</b>	
	<i>Room: Affari-3.D</i>	
	TWO-PHASE FLOW AND HEAT TRANSFER CHARACTERISTICS IN NON-UNIFORMLY HEATED HELICAL COILED TUBES	
	<i>Niu Xiao-Juan, Fan Liang-Liang, Luo Shuaishuai, Zhao Liang</i>	88/4
	A COMPUTATIONAL STUDY OF SURFACTANT INDUCED ENHANCED HEAT TRANSFER IN FILM BOILING	
We 14:40-16:00	<b>Droplet Flows</b>	
	<i>Room: Affari-0.A</i>	
	ELECTROSTATIC CONTROL OF IMPACT DYNAMICS OF A DROPLET ON SOLID SURFACES: SUPERHYDROPHOBIC BEHAVIOR	
	<i>Datta Saikat, Das Arup Kumar, Das Prasanta Kumar</i>	89/1
	A NOVEL DROPLET DRYING MODEL APPLIED TO A LABORATORY SPRAY DRYER	
We 14:40-16:00	<b>Experimental Methods for Multiphase Flows</b>	
	<i>Room: Affari-3.C</i>	
	WET GAS MEASUREMENT BY V-CONE THROTTLE DEVICE	
	<i>He Denghui, Bai Bofeng</i>	89/5
	COMPARISON BETWEEN EXPERIMENTAL CHARACTERISATION OF THE MIXING OF IMMISCIBLE FLUIDS IN STATIC MIXER USING PLIF AND CFD MODELLING USING OPEN FOAM.	
We 14:40-16:00	<b>Granular Media &amp; Granular flow</b>	
	<i>Room: Affari-3.B</i>	
	COMPUTATIONAL STUDY OF DENSE GRANULAR FLOWS IN STIRRED REACTORS	
	<i>Bennani Lokman, Neau Herve, Baudry Cyril, Lavieville Jerome, Fede Pascal, Simonin Olivier</i>	90/3
	LARGE EDDY SIMULATION OF SAND WAVE EVOLUTION	
We 14:40-16:00	<b>Granular Media &amp; Granular flow</b>	
	<i>Room: Affari-3.B</i>	
	DRAG REDUCTION DUE TO ROTATION WITHIN GRANULAR MEDIA	
	<i>Khosronejad Ali, Sotiropoulos Fotis</i>	90/4
	JUNG WONJONG, CHOI SUNG MOK, KIM WONJUNG, KIM HO-YOUNG	90/5
We 14:40-16:00	<b>Granular Media &amp; Granular flow</b>	
	<i>Room: Affari-3.B</i>	
	ANALYTICAL AND PRACTICAL ANALYSIS OF FRICTIONAL-KINETIC MODEL FOR DENSE AND DILUTE GAS-SOLID FLOWS	
	<i>Fede Pascal, Audard Francois, Belut Emmanuel, Fontaine Jean-Raymond, Simonin Olivier</i>	90/6

<b>We 14:40-16:00</b>	<b>Industrial Applications</b> <b>Room: Affari-1.A</b> DEVELOPMENT OF AN INDUSTRIAL SCALE MICROBUBBLE GENERATOR FOR THE PURPOSES OF AEROBIC WASTE WATER TREATMENT <i>Majid Akmal Irfan, Deendarlianto Deendarlianto, Wiratni Wiratni, Indarto Indarto, Brata Enggar Diwangkara, Purwono Baskoro Adiputro, Tontowi Alva Edy</i>	91/1
	EFFICIENT MODELLING OF BUBBLE SIZE DISTRIBUTION IN A FERMENTATION REACTOR WITH CFD <i>Eppinger Thomas, Aglave Ravindra, Lo Simon</i>	91/2
	PHYSICAL AND MATHEMATICAL MODELLING OF MULTIPHASE FLOW IN THE NEAR-WALL REGION OF ENTRAINED FLOW REACTORS <i>Tretola Giovanni, Troiano Maurizio, Marra Francesco Saverio, Salatino Piero, Montagnaro Fabio, Solimene Roberto</i>	91/3
	EXPERIMENTAL INVESTIGATION OF AN AXIAL OIL-WATER SWIRL SEPARATOR <i>Mudde Robert, Hoeijmakers Harry, Ashok Anand, Hospers Jacco</i>	91/4
<b>We 14:40-16:00</b>	<b>Micro- and Nano-Scale Multiphase Flows</b> <b>Room: Affari-2.C</b> PRESSURE DROP FOR GAS-NON-NEWTONIAN LIQUID TWO-PHASE FLOW IN CIRCULAR MICROCHANNEL <i>Kawahara Akimaro, Sadatomi Michio, Law Wen Zhe, Mori Akifumi, Mansour Mohamed H.</i>	91/5
	MASS TRANSFER WITH NANOFLUIDS – THE ROLE OF MICROCONVECTION <i>Michaelides Efsthios</i>	91/6
	PROPERTIES OF FINE BUBBLES AND OF NANO PERTICLES IN PURE WATER <i>Ueda Yoshikatsu, Tokuda Yomei, Nihei Naoto, Hamamoto Shoichiro, Ogawa Yuichi, Sugiyama Akifumi</i>	92/1
<b>We 14:40-16:00</b>	<b>Mixing</b> <b>Room: Affari-3.A</b> USING TOMOGRAPHY TECHNIQUE TO ANALYZE THE SUSPENSION OF THE SOLID PARTICLES IN A SLURRY REACTOR <i>Ein Mozaffari Farhad, Mishra Prakash</i>	92/2
	BOTTOM BUBBLING CONDITIONS ON MIXING CHARACTERISTICS IN A GAS-STIRRED LADLE <i>Jeong Eunju, Lee Sang-Beom</i>	92/3
	MIXING INDUCED BY A HOMOGENOUS BUBBLE SWARM IN PRESENCE OF EXTERNAL TURBULENCE <i>Almeras Elise, Mathai Varghese, Lohse Detlef, Sun Chao</i>	92/4
	EXPERIMENTAL CHARACTERIZATION OF INTERCHANNEL MIXING OF MULTIPHASE FLOW THROUGH A NARROW GAP <i>Makharju Simo, Gose James W., Buchanan John, Mychkovsky Alexander, Hogan Kevin, Lowe Kirk, Ceccio Steven L.</i>	92/5
<b>We 14:40-16:00</b>	<b>Modelling of Multiphase Flows</b> <b>Room: Affari-2.D</b> MULTIPHASE EQUATIONS SUITABLE FOR THE NUMERICAL SIMULATION OF ICE PRODUCTION IN OCEAN <i>Covello Vanessa, Abba Antonella, Bonaventura Luca, Valdetaro Lorenzo</i>	92/6
	PREDICTION OF AEROSOL SAMPLING EFFICIENCY USING A SECTIONAL DROPLET DRIFT FLUX MODEL <i>Frederix Edo, Kuczaj Arkadiusz, Nordlund Markus, Geurts Bernard</i>	93/1
	THE IMPACT OF DIFFERENT FINE GRID SIMULATIONS ON THE SUB-GRID MODIFICATION FOR GAS-SOLID DRAG <i>Schneiderbauer Simon, Pirker Stefan</i>	93/2
	NUMERICAL MODELING OF CONVECTIVE FLOW IN COMPLEX HYDROCARBON LIQUID UNDER RADIO-FREQUENCY HEATING <i>Kovaleva Liana, Kireev Victor, Musin Ayrat</i>	93/3
<b>We 14:40-16:00</b>	<b>Particle-Laden Flows</b> <b>Room: Villa-Auditorium</b> AN EXPERIMENTAL AND NUMERICAL VALIDATION STUDY OF PARTICLE LADEN SUPERSONIC FLOWS <i>Rudek Arthur, Russ Gerald, Duignan Barry</i>	93/4
	PARTICLE IMAGE VELOCIMETRY MEASUREMENTS OF INJECTED PARTICLE PLUMES IN A TWO-DIMENSIONAL VERTICAL CHANNEL <i>Yates Matthew, Hann David, Hewakandamby Buddhikha, Azzopardi Barry James</i>	93/5
	TURBULENCE ATTENUATION IN PARTICLE-LADEN FLOW <i>Vreman Bert</i>	93/6

<b>We 14:40-16:00</b>	<b>Turbulence in Multiphase Flows</b> <i>Room: Villa-Sala Verde</i>	
	TURBULENT HEAT FLUXES UNDER THE INFLUENCE OF EVAPORATING DROPLETS <i>Richter David, Helgans Brian</i>	94/1
	INTERACTION BETWEEN A FINITE-SIZE BUOYANT BUBBLE AND TURBULENCE <i>Loisy Aureole, Naso Aureole, Spelt Peter</i>	94/2
	MODELING TURBULENT LIQUID/GAS PHASE INTERFACE USING A VOF BASED DUAL-SCALE LES SUBGRID MODEL <i>Herrmann Marcus</i>	94/3
	STUDY OF PARTICLE-TURBULENCE INTERACTIONS IN TURBULENT PARTICLE-LADEN FLOW BY INTERFACE-RESOLVED SIMULATIONS <i>Wang Lian-Ping, Peng Cheng</i>	94/4

<b>We 14:40-16:00</b>	<b>Fiber suspension flow (focus session)</b> <i>Room: Affari-2.A</i>	
	ROTATION OF NON-SPHERICAL PARTICLES IN TURBULENCE - EFFECT OF SIZE ON KINEMATIC STATISTICS <i>Variano Evan, Pujara Nimish, Bordoloi Nimish</i>	94/5
	RELATIVE ROTATIONAL MOTION BETWEEN RIGID FIBERS AND FLUID IN TURBULENT CHANNEL FLOW <i>Marchioli Cristian, Zhao Lihao, Andersson Helge</i>	94/6
	COMBINED MEASUREMENTS OF FLOW FIELD AND RIGID, INERTIAL FIBRE ROTATION/TRANSLATION IN NEAR HOMOGENEOUS ISOTROPIC TURBULENCE <i>Sabban Lilach, Cohen Asaf, Van Hout Rene</i>	95/1
	MEASURING ROTATION AND ALIGNMENT OF ANISOTROPIC PARTICLES IN TURBULENCE <i>Voth Greg</i>	95/2

## PARALLEL SESSIONS XII

<b>We 16:20-17:40</b>	<b>Bubbly Flows</b> <i>Room: Affari-2.B</i>	
	TWO-PHASE FLOW AND TOTAL PRESSURE IN HIGH-VELOCITY FREE-SURFACE CHUTE FLOWS <i>Zhang Gangfu, Chanson Hubert</i>	95/3
	EXPERIMENTAL STUDY ON ATOMIZATION OF ALLOPHANE WITH MICRO BUBBLES <i>Inoue Yuzo, Arai Yoshihiro, Kaneko Akiko, Takemura Fumio, Ike Masatoshi, Kanagawa Tetsuya, Abe Yutaka</i>	95/4
	MODELLING OF TURBULENCE MODULATION IN BUBBLY FLOWS WITH THE AID OF DIRECT NUMERICAL SIMULATION <i>Ma Tian, Ziegenhein Thomas, Lucas Dirk, Frohlich Jochen</i>	95/5

<b>We 16:20-17:40</b>	<b>Boiling, Condensation, Evaporation</b> <i>Room: Affari-3.D</i>	
	A SIMPLIFIED ENERGY DISSIPATION BASED MODEL OF HEAT TRANSFER FOR SUBCOOLED FLOW BOILING <i>Mikielewicz Dariusz, Mikielewicz Jaroslaw</i>	95/6
	EXPERIMENTAL ANALYSIS OF CONDENSATION INDUCED WATER HAMMERS IN HORIZONTAL PIPES <i>Urban Christian, Hoffmann Marko, Schlueter Michael</i>	96/1
	PARTICLE PROPERTIES AND PHASE TRANSITION IN AN INDUCTIVELY COUPLED PLASMA SPRAYING <i>Qian Lijuan, Lin Jianzhong</i>	96/2

<b>We 16:20-17:40</b>	<b>Droplet Flows</b> <i>Room: Affari-0.A</i>	
	NUMERICAL SIMULATION OF MULTIPLE DROPLET IMPINGEMENT USING DIFFUSE INTERFACE MODEL <i>Sugitani Yosuke, Tsujimoto Koichi, Shakouchi Toshihiko, Ando Toshitake</i>	96/3
	NOVEL MEASUREMENT FOR DISPERSED DROPLETS THROUGH A SINGLE TIP OPTICAL FIBER PROBE WITH POST-SIGNAL PROCESSING <i>Shimizu Hiromu, Saito Takayuki</i>	96/4
	INFLUENCE OF GAS AND DROPLET PROPERTIES ON DROP-DROP COLLISION OUTCOMES <i>Krishnan Ghoku, Loth Eric</i>	96/5

We 16:20-17:40	<b>Granular Media &amp; Granular flow</b>	
	<i>Room: Affari-3.B</i>	
	AN EXPERIMENTAL STUDY OF HEAT TRANSFER MECHANISM IN GRANULAR FLOWS <i>Hsiau Shu-San, Liu Shang-Yu, Chou Shih-Hao</i>	96/6
	FLUID-GRANULAR MEDIA MULTI-PHASE MODELLING USING THE SMOOTHED PARTICLE HYDRODYNAMICS METHOD <i>Szewc Kamil</i>	97/1
	SATURATION LENGTH SCALE ON WALL PRESSURE DISTRIBUTION DURING A GRANULAR DISCHARGE FROM A NARROW SILO <i>Jiang Mingjer, Yang Fuling</i>	97/2
We 16:20-17:40	<b>Industrial Applications</b>	
	<i>Room: Affari-1.A</i>	
	COMPUTATIONAL MODELING OF DROPLET COALESCENCE IN COMPLEX GEOMETRIES <i>Wang Sheng, Agbaglah Gilou, Desjardins Olivier</i>	97/3
	NUMERICAL INVESTIGATION OF AN AXIAL OIL-WATER SWIRL SEPARATOR <i>Hospers Jacco, Hoeijmakers Harry</i>	97/4
	PREDICTION OF AIR RELEASE IN HYDRAULIC RESERVOIRS USING LAGRANGIAN PARTICLE TRACKING <i>Longhitano Marco, Murrenhoff Hubertus</i>	97/5
	UNCERTAINTY QUANTIFICATION OF THE TWO-FLUID MODEL AND FLOW PATTERN TRANSITION BOUNDARIES PREDICTIONS FOR THE STRATIFIED FLOW REGIME BY MONTECARLO SIMULATIONS AND POLYNOMIAL CHAOS EXPANSIONS. <i>Picchi Davide, Poesio Pietro</i>	97/6
We 16:20-17:40	<b>Instabilities</b>	
	<i>Room: Affari-3.C</i>	
	FLOW REGIMES AND INSTABILITIES IN A T-MIXER OPERATING WITH A BINARY MIXTURE <i>Siconolfi Lorenzo, Galletti Chiara, Camarri Simone, Salvetti Maria Vittoria</i>	98/1
	THE CHARACTERISTIC SCALES OF THE CAPILLARY INSTABILITY <i>Dumouchel Christophe, Aniszewski Wojciek, Menard Thibaut, Vu Trung-Thanh</i>	98/2
	EXPERIMENTAL INVESTIGATION OF INTERFACIAL WAVES INSTABILITIES IN STRATIFIED FLOW <i>Stofer Cordeiro De Farias Paula, Braga De Paula Igor, Ayati Anis Awal, Alzuguir Azevedo Luis Fernando</i>	98/3
	FARADAY INSTABILITY IN THREE FLUID MECHANICALLY OSCILLATED SYSTEMS <i>Ward Kevin, Zoueshtiagh Farzam, Narayanan Ranga</i>	98/4
We 16:20-17:40	<b>Mixing</b>	
	<i>Room: Affari-3.A</i>	
	EXPERIMENTAL STUDY ON FRAGMENTATION OF A MOLTEN JET INTO WATER <i>Hadj-Achour Miloud, Rimbert Nicolas, Castrillon-Escobar Sebastian, Gradeck Michel, Meignen Renaud</i>	98/5
	GRANULAR MIXING INSIDE ROTATING CYLINDERS: PRELIMINARY RESULTS <i>De Monaco Gaetano, Lombardi Davide, Greco Francesco, Maffettone Pier Luca</i>	98/6
	INVESTIGATION ON THE PREDICTION OF DROP SIZE DISTRIBUTION IN SURFACTANT ADDED OIL-WATER EMULSION <i>Liu Nannan, Wang Wei, Tian Yunya, Han Jiacheng, Zhang Meng, Gong Jing</i>	99/1
	SIMULATION OF THE DYNAMICS COMPLEX INFLUENCE ON HYDROCARBON MIXTURES <i>Assylbekuly Aidar, Zhakebayev Dauren, Abdibekov Ualikhan</i>	99/2
We 16:20-17:40	<b>Modelling of Multiphase Flows</b>	
	<i>Room: Affari-2.D</i>	
	PREDICTION OF INTERFACE LEVEL HEIGHT OF STRATIFIED LIQUID-LIQUID FLOW USING ARTIFICIAL NEURAL NETWORK <i>Awad Mohamed, Azizi Sadra, Ahmadloo Ebrahim, Ibarra Roberto, Zadrzil Ivan, Markides Christos N.</i>	99/3
	TOWARDS LARGE-EDDY SIMULATION OF LIQUID ATOMIZATION: A-PRIORI SUBGRID ANALYSIS <i>Ketterl Sebastian, Klein Markus</i>	99/4
	TOWARDS GENERAL PURPOSE LES MODEL OF ATOMIZATION <i>Hecht Nicolas, Demoulin Francois-Xavier, Reveillon Julien</i>	99/5
	TWO-PHASE FLOWS IN SINUSOIDALLY CONSTRICTED CHANNELS USING MOVING MESH MOVING BOUNDARY TECHNIQUE <i>Anjos Gustavo, Oliveira Gustavo, Pontes Jose, Mangiavacchi Norberto, Thome John</i>	99/6

<b>We 16:20-17:40</b>	<b>Particle-Laden Flows</b>	
	<i>Room: Villa-Auditorium</i>	
	LARGE EDDY SIMULATION OF GAS-PARTICLE TURBULENT OPPOSED JET FLOW	
	<i>Wu Dan, Li Jing, Liu Zhaohui, Zheng Chuguang</i>	100/1
	LARGE EDDY SIMULATIONS OF PARTICULATE FLOW INSIDE A DIFFERENTIALLY HEATED	
	CAVITY AT RAYLEIGH NUMBERS UP TO $10^{10}$	
	<i>Dehbi Abdel, Kalilainen Jarmo</i>	100/2
	STUDY ON SEAWEED REMOVAL EFFICIENCY FROM THE WASTEWATER BY SWIRL FLOW	
	SEPARATOR	
	<i>Tsubone Hiroaki, Shimomura Masaya, Takesue Syuichi</i>	100/3
	SWIRLING FLOW CURVATURE SENSITIVE TURBULENCE MODELLING OF A GAS-SOLID	
	CYCLONE	
	<i>Alahmadi Yaser, Nowakowski Andrzej</i>	100/4
<b>We 16:20-17:40</b>	<b>Reactive Multiphase Flows</b>	
	<i>Room: Affari-2.C</i>	
	UNSTEADY THREE-DIMENSIONAL NUMERICAL SIMULATIONS OF THE METHANE	
	COMBUSTION IN DENSE FLUIDIZED BED	
	<i>Hamidouche Ziad, Masi Enrica, Fede Pascal, Simonin Olivier, Ansart Renaud, Hemati Mehdi</i>	100/5
	NUMERICAL SIMULATION OF PROJECTILE LAUNCHED AT THE UNDERWATER USING	
	GRANULAR PROPELLANTS	
	<i>Asano Kei, Matsuo Akiko</i>	100/6
	DIRECT NUMERICAL SIMULATION OF COMBUSTION NOISE OF A SPRAY FLAME	
	<i>Pillai Abhishek, Kitano Tomoaki, Kurose Ryoichi, Komori Satoru</i>	101/1
	EFFECTS OF PERMEABILITY-DEPENDENT POROUS ELECTRODE COMPRESSED RATIO ON	
	THE PERFORMANCE OF VANADIUM REDOX FLOW BATTERY	
	<i>Wang Qiong, Qu Zhiguo</i>	101/2
<b>We 16:20-17:40</b>	<b>Turbulence in Multiphase Flows</b>	
	<i>Room: Villa-Sala Verde</i>	
	MODULATION OF TURBULENCE BY DISPERSED SOLID PARTICLES IN A SPATIALLY	
	DEVELOPING FLAT PLATE BOUNDARY LAYER	
	<i>Li Dong, Luo Kun, Fan Jianren</i>	101/3
	ON THE EMULATION OF BUBBLE-SWARM-INDUCED TURBULENCE WITH FLEXIBLE	
	PARTICLE GRIDS	
	<i>Haase Katharina, Kuck Ulf, Thoming Jorg, Kaehler Christian</i>	101/4
	DIRECT NUMERICAL SIMULATION MODEL INTERCOMPARISON FOR TURBULENT	
	COLLISION STATISTICS OF INERTIAL PARTICLES	
	<i>Rosa Bogdan, Onishi Ryo, Wang Lian-Ping</i>	101/5
	MODELLING TURBULENCE IN LARGE SCALE BUBBLE PLUMES	
	<i>Olsen Jan Erik, Johansen Stein Tore, Skjetne Paal</i>	101/6
<b>We 16:20-17:40</b>	<b>Fiber suspension flow (focus session)</b>	
	<i>Room: Affari-2.A</i>	
	ROTATION OF A NEUTRALLY BUOYANT SPHEROID IN A SIMPLE SHEAR AT SMALL	
	REYNOLDS NUMBER	
	<i>Einarsson Jonas, Candelier Fabien, Lundell Fredrik, Angilella Jean-Regis, Mehlig Bernhard</i>	102/1
	DNS OF TURBULENT CHANNEL FLOW LADEN WITH A DENSE SUSPENSION OF NEUTRALLY	
	BUOYANT FINITE-SIZE SPHEROIDS	
	<i>Niazi Ardekani Mehdi, Costa Pedro, Breugem Wim-Paul, Brandt Luca</i>	102/2
	INVESTIGATION OF DRAG FORCE ON FIBRES OF BONDED SPHERICAL ELEMENTS USING	
	A COUPLED CFD-DEM APPROACH	
	<i>Jensen Anna Lyhne, Soerensen Henrik, Rosendahl Lasse, Adamsen Per, Lykholt Ustrup Flemming</i>	102/3
	SUSPENSIONS OF INERTIAL SPHEROIDAL PARTICLES IN TURBULENT VERTICAL CHANNEL	
	FLOW	
	<i>Andersson Helge, Challabotla Niranjan Reddy, Zhao Lihao</i>	102/4

## THURSDAY, May 26, 2016

### PARALLEL SESSIONS XIII

Thu 9:50-11:10	<b>Computational Techniques for Multiphase Flows</b>	
	<i>Room: Villa-Auditorium</i>	
	UNCERTAINTY QUANTIFICATION OF INJECTED DROPLET SIZE IN MONO-DISPERSE EULERIAN SIMULATIONS	
	<i>Lancien Thea, Dumont Nicolas, Gicquel Olivier, Vicquelin Ronan</i>	102/5
	TWO-SIZE MOMENT EULERIAN MULTI-FLUID METHOD DESCRIBING THE STATISTICAL TRAJECTORY CROSSING: MODELING AND NUMERICAL SCHEME	
	<i>Boileau Matthieu, Dupif Valentin, Laurent Frederique, Lagarde Julien, Massot Marc</i>	102/6
Thu 9:50-11:10	<b>Droplet Flows</b>	
	<i>Room: Affari-3.D</i>	
	DYNAMICS OF DOUBLE EMULSION DROPLETS IN SHEAR AND CHANNEL FLOWS	
	<i>Kim Sangkyu (calvin), Dabiri Sadegh</i>	103/3
	DROP BREAKUP MODELS IN SPRAY SIMULATIONS	
	<i>Kekesi Timea, Amberg Gustav, Prah Wittberg Lisa</i>	103/4
Thu 9:50-11:10	<b>Fluidization</b>	
	<i>Room: Affari-3.A</i>	
	DEVELOPMENT OF CLOSURE MODELS FOR UNRESOLVED PARTICLE SIMULATIONS OF HEAT AND MASS TRANSFER IN DENSE PARTICLE BEDS USING CPPPO: A LIBRARY FOR FAST PARALLEL DATA FILTERING	
	<i>Municchi Federico, Radl Stefan, Goniva Christoph</i>	104/1
	FLUIDIZING PULVERISED COAL WITH A VERY WIDE SIZE DISTRIBUTION	
	<i>Azzopardi Barry James, Li Xia, Escrig Escrig Josep, Hann David</i>	104/2
Thu 9:50-11:10	<b>Industrial Applications</b>	
	<i>Room: Affari-1.A</i>	
	TOWARDS HIGH ORDER ACCURACY CAVITATION SIMULATION WITH THE DISCONTINUOUS GALERKIN SPECTRAL ELEMENT METHOD	
	<i>Hempert Fabian, Boblest Sebastian, Hoffmann Malte, Offenhaeuser Philipp, Iben Uwe, Munz Claus-Dieter</i>	104/5
	EVALUATION OF VOID FRACTION AROUND A TUBE IN TWO-PHASE FLOW ACROSS HORIZONTAL TUBE BUNDLE	
	<i>Murakawa Hideki, Baba Misaki, Sugimoto Katsumi, Takenaka Nobuyuki</i>	104/6
Thu 9:50-11:10	CFD-INFORMED UNIFIED CLOSURE RELATION FOR THE RISE VELOCITY OF TAYLOR BUBBLES IN PIPES WITH LIQUID FLOW	
	<i>Lizarraga-Garcia Enrique, Buongiorno Jacopo, Al Safran Eissa, Lakehal Djamel</i>	105/1
	NUMERICAL ANALYSIS OF FLOW CHARACTERISTICS INSIDE HYDRAULIC COUPLING AT VARIOUS OPERATIONAL CONDITIONS	
	<i>Kwak Minhee, Hur Nahmkeon, Chang Choon-Soo, Kang Nam-Won</i>	105/2

Thu 9:50-11:10	<b>Instabilities</b> <b>Room: Affari-3.C</b> STABILITY OF STRATIFIED TWO-PHASE FLOWS IN INCLINED CHANNELS <i>Barmak Ilya, Gelfgat Alexander, Ullmann Amos, Brauner Neima</i> 105/3 THREE-DIMENSIONAL STUDY OF DISTURBANCE WAVES FORMATION IN DOWNWARD FLOW OF GAS-SHEARED LIQUID FILM <i>Cherdantsev Mikhail, Isaenkov Sergey, Cherdantsev Andrey, Markovich Dmitriy, Alekseenko Sergey</i> 105/4 INVESTIGATION OF PARALLEL CHANNEL INSTABILITIES WITH VAPOR BACKFLOW USING AN IMPROVED DYNAMIC MICROCHANNEL EVAPORATOR MODEL <i>Saenen Tom, Thome John</i> 105/5 INSTABILITIES IN STRATIFIED TWO THERMOVISCOUS LIQUIDS FLOW IN A PLANE CHANNEL <i>Kireev Victor, Nizamova Adelina, Urmanceev Said</i> 105/6
Thu 9:50-11:10	<b>Micro- and Nano-Scale Multiphase Flows</b> <b>Room: Affari-2.B</b> CFD SIMULATION OF GROWING NANOPARTICLES IN A FSP FLAME USING DIRECT QUADRATURE METHOD OF MOMENTS <i>Bianchi Pedro, Noriler Dirceu, Meier Henry França, Fritsching Udo, Meierhofer Florian</i> 106/1 STUDY OF NATURAL CONVECTION IN AN ENCLOSURE WITH WATER AND a AL2O3/WATER NANOFLUID FOR DIFFERENT ASPECT RATIOS USING STATISTICAL ANALYSIS <i>Chowdhury Rajesh, Subudhi Sudhakar</i> 106/2 EXPERIMENT AND COMPUTATIONAL SIMULATIONS OF LIQUID-LIQUID INTERFACIAL INSTABILITIES UPON FLOW DISPLACEMENT IN MICROCHANNELS <i>Lu Yu, Simmons Mark</i> 106/3 CFD SIMULATION OF WATER REMOVAL FROM GAS FLOW CHANNELS OF PEMFC USING WETTABILITY GRADIENT <i>Khan Shadman, Ghosh Sumana</i> 106/4
Thu 9:50-11:10	<b>Modelling of Multiphase Flows</b> <b>Room: Affari-2.D</b> MOFFAT VORTICES IN THE VICINITY OF A CONTACT LINE <i>Febres Soria Mijail, Legendre Dominique</i> 106/5 WET GAS FLOW REGIMES: EXPERIMENTS AND TRANSITION MODEL BASED ON DIMENSIONLESS NUMBERS <i>Van Putten Dennis, Hagmeijer Rob</i> 106/6 VAPORIZATION AND EXPANSION OF MOLTEN FUEL IN A SODIUM FAST REACTOR: A COMBINED NON-EQUILIBRIUM THERMODYNAMICS AND DIMENSIONAL ANALYSIS METHODOLOGY <i>Manchon Xavier, Bertrand Frederic, Marie Nathalie, Lance Michel, Schmitt Damien</i> 107/1 CONDENSATION IN SUPERSONIC STEAM EJECTORS: COMPARISON OF THEORETICAL AND NUMERICAL MODELS <i>Mazzelli Federico, Brezgin Dmitry, Murmanskii Ilia, Nikolay Zhelonkin, Milazzo Adriano</i> 107/2
Thu 9:50-11:10	<b>Particle-Laden Flows</b> <b>Room: Affari-3.B</b> EXPERIMENTAL STUDY OF PARTICLES BEHAVIOR IN TORNADO-LIKE GAS-SOLID FLOW <i>Protasov Mikhail, Vasiliev Nikolay, Varaksin Aleksei</i> 107/3 PARTICLE-LADEN FLOW IN MODELS OF ASTHMATIC AIRWAYS <i>Koullapis Pantelis, Kassinos Stavros</i> 107/4 SIMULATION OF A LIQUID-SOLID FLUIDIZED BED CRYSTALLIZER USING CFD-DEM <i>Kerst Kristin, Medeiros De Souza Luis, Seidel-Morgenstern Andreas, Janiga Gábor</i> 107/5 THERMODYNAMICALLY CONSISTENT TWO-WAY COUPLED MODELLING OF GAS-PARTICLE FLOWS WITH PARTICLE TRAJECTORY CROSSING USING EULERIAN MOMENT METHODS <i>Massot Marc, Dupif Valentin, Dupays Joel, Laurent Frederique</i> 107/6
Thu 9:50-11:10	<b>Reactive Multiphase Flows</b> <b>Room: Affari-2.C</b> TRANSITION BETWEEN ROBUST EQUILIBRIUM DESIGN POINTS FOR STEADY STATE PROCESSES OF PULVERIZED COAL COMBUSTION IN DOWNERS <i>Eluk Tal, Levy Avi, Korytnyi Efim, Bar Kohany Tali</i> 108/1 PHYSICAL AND COMBUSTION CHARACTERIZATION OF BIOMASS PARTICLES PULVERIZED BY DIFFERENT MILLING PROCESSES FOR SUSPENSION FIRING IN UTILITY BOILERS <i>Yin Chungen, Momeni Maryam, Kaer Soren Knudsen</i> 108/2 EXPERIMENTAL UNCERTAINTY RECONCILIATION IN DROP TUBE CHARACTERIZATION OF REACTING STREAMS OF PARTICLES <i>Galletti Chiara, Caposciutti Gianluca, Iavarone Salvatore, Parente Alessandro, Tognotti Leonardo</i> 108/3

	THE INFLUENCE OF REACTOR ENCLOSURE AND GAS ENTRAINMENT ON FLAME SPRAYS WITH APPLICATION IN NANOPARTICLE SYNTHESIS <i>Meierhofer Florian, Buss Lizoel, Noriler Dirceu, Meier Henry França, Fritsching Udo</i>	108/4
Thu 9:50-11:10	<b>Turbulence in Multiphase Flows</b> <i>Room: Villa-Sala Verde</i> MEASUREMENTS OF VOID FRACTION, PRESSURE DROP AND HEAT TRANSFER IN HORIZONTAL AND DOWNWARD INCLINED GAS-LIQUID STRATIFIED FLOW <i>Bhagwat Swanand, Ghajar Afshin</i>	108/5
	STUDY OF A TURBULENCE DAMPING PROCEDURE NEAR THE INTERFACE OF GAS-LIQUID STRATIFIED FLOW <i>Gada Vinesh H., Punde Pranav, Tandon Mohit P., Vikulov Roman</i>	108/6
	MODELING INTER-PHASE MOMENTUM COUPLING VIA THE EXACT REGULARIZED POINT PARTICLE METHOD <i>Gualtieri Paolo, Battista Francesco, Casciola Carlo Massimo</i>	109/1
	DNS OF EVAPORATING DROPLETS IN DECAYING ISOTROPIC TURBULENCE <i>Dodd Michael, Ferrante Antonino</i>	109/2
Thu 9:50-11:10	<b>Fiber suspension flow (focus session)</b> <i>Room: Affari-2.A</i> LONG FLEXIBLE FIBERS IN TURBULENT FLOWS <i>Sulaiman Mostafa, Climent Eric, Delmotte Blaise, Fede Pascal, Plouraboue Franck, Verhille Gautier</i>	109/3
	NUMERICAL SIMULATION AND VALIDATION OF ELONGATED FIBRES DISPERSION AND TRANSPORT <i>Lain Santiago, Quintero Brian, Sommerfeld Martin</i>	109/4
	NUMERICAL SIMULATION OF FLEXIBLE FIBER CONCENTRATION DISTRIBUTION IN A TWO- DIMENSIONAL ANALYTICAL FLOW FIELD FOR DIFFERENT STOKES NUMBERS <i>Martinez Manuel, Pallares Jordi, Vernet Anton</i>	109/5
	DEVIATORIC STRESSES IN SUSPENSIONS OF FLEXIBLE OR CURVED FIBERS <i>Lindstrom Stefan, Andric Jelena, Sasic Srdjan, Nilsson Hakan</i>	109/6

## PARALLEL SESSIONS XIV

Thu 11:30-12:50	<b>Colloidal and Suspension Dynamics</b> <i>Room: Affari-3.B</i> BASIC EQUATIONS OF SEPARATION PROCESSES IN SUSPENSIONS WITH GRAVITY, CENTRIFUGAL AND CORIOLIS FORCES <i>Schneider Wilhelm</i>	110/1
	SIMULATIONS TO EXPLORE THE STABILITY OF NANOPARTICLE SUSPENSIONS UNDER HYDRODYNAMIC SHEAR <i>Vo Minh, Papavassiliou Dimitrios</i>	110/2
	SEDIMENTATION OF DILUTE SUSPENSIONS OF SPHERES AT FINITE REYNOLDS NUMBERS <i>Bergougnoux Laurence, Guazzelli Elisabeth</i>	110/3
	A NUMERICAL STUDY FOR SUSPENSION PARTICLES MIXED WITH DROP FLOWS <i>Chen Ming, Zhang Baili, Lou Jing</i>	110/4
Thu 11:30-12:50	<b>Computational Techniques for Multiphase Flows</b> <i>Room: Villa-Auditorium</i> APPLICATION OF A LEVEL-SET /VOLUME-OF-FLUID METHOD AND AN IMMERSED BOUNDARY METHOD TO THE ATOMIZATION OF A LIQUID JET EJECTED FROM A COMPLEX- SHAPED INJECTOR <i>Vu Trung-Thanh, Menard Thibaut, Dumouchel Christophe</i>	110/5
	DEVELOPMENT OF A MULTI-PHASE VOF DYNAMIC SOLVER IN THE OPENFOAM® TECHNOLOGY: AN APPLICATION TO THE SIMULATION OF THE OPENING AND CLOSURE EVENTS IN HIGH PRESSURE GDI INJECTORS <i>Piscaglia Federico, Montorfano Andrea, Helie Jerome, Demoulin Francois-Xavier</i>	110/6
	A LES-DEM-VOF METHOD FOR TURBULENT THREE PHASE FLOWS <i>Pozzetti Gabriele, Peters Bernhard</i>	111/1
	3D, GPU-ACCELERATED AND MPI-PARALLEL SIMULATIONS OF TWO-FLUID FLOWS INTERACTING WITH MOVING RIGID BODIES – APPLICATION IN RENEWABLE ENERGY SYSTEMS <i>Raessi Mehdi, Pathak Ashish</i>	111/2
Thu 11:30-12:50	<b>Droplet Flows</b> <i>Room: Affari-3.D</i> COALESCENCE AND BREAK-UP OF LARGE, DEFORMABLE DROPLETS WITH DIFFERENT VISCOSITIES IN A TURBULENT CHANNEL FLOW <i>Roccon Alessio, Zonta Francesco, Soldati Alfredo</i>	111/3



	EFFECTS OF INTERFACIAL TENSION AND VISCOSITY ON DROPLETS FORMATION BY BREAKING WAVES IMPINGING ON AN OIL SLICK <i>Li Cheng, Katz Joseph</i>	111/4
	HIGH FIDELITY SIMULATION OF IMPACT OF HIGH TEMPERATURE AND PRESSURE ON LIQUID JET IN CROSSFLOW ATOMIZATION <i>Soteriou Marios, Li Xiaoyi</i>	111/5
	RECIRCULATION, ENTRAINMENT AND CLUSTER FORMATION IN BOUNDED SPRAYS <i>Lampa Aljoscha, Sander Soren, Schwenck Daniel, Fritsching Udo</i>	111/6
Thu 11:30-12:50	<b>Fluidization</b> <b>Room: Affari-3.A</b> NUMERICAL ANALYSIS OF DENSE FLUIDIZATION: A BENCHMARK FOR PARTICLE RESOLVED SIMULATIONS OF LIQUID-SOLID AND GAS-SOLID FLUIDIZED BEDS <i>Esteghamatian Amir, Bernard Manuel, Hammouti Abdelkader, Wachs Anthony, Uhlmann Markus, Lance Michel</i>	112/1
	NUMERICAL SIMULATION OF PERIODIC CIRCULATING FLUIDIZED BED OF BINARY MIXTURE OF PARTICLES – BUDGET ANALYSIS <i>Chevrier Solene, Fede Pascal, Simonin Olivier</i>	112/2
	A NEW DRAG CORRELATION FROM PARTICLE-RESOLVED DIRECT NUMERICAL SIMULATION OF A GAS-FLUIDIZED BED <i>Tan Junhua, Luo Kun, Fan Jianren</i>	112/3
	DRAG FORCE MODELLING IN DILUTE TO DENSE PARTICLE-LADEN FLOWS WITH MONO-DISPERSE OR BINARY MIXTURE OF SOLID PARTICLES <i>Simonin Olivier, Chevrier Solene, Audard Francois, Fede Pascal</i>	112/4
Thu 11:30-12:50	<b>Instabilities</b> <b>Room: Affari-3.C</b> INSTABILITY DUE TO INTERFACIAL TENSION IN STRATIFIED AND CORE-ANNULAR FLOW <i>Hernandez Rodriguez Oscar Mauricio, Rocha Douglas Martins</i>	112/5
	A NUMERICAL INVESTIGATION OF FLOW BOILING INSTABILITY IN HORIZONTAL TUBE <i>Wang Yufei, Zhang Liang, Yu Zi-Tao</i>	112/6
	TIME-DOMAIN STUDY ON THE EFFECT OF CHANNEL LENGTH AND CHANNEL NUMBER ON DENSITY-WAVE INSTABILITY OF SUPERCRITICAL FLUID FLOWS IN SINGLE CHANNEL AND PARALLEL MULTIPLE-CHANNELS SYSTEM <i>Liu Jialun, Li Hui-Xiong, Li Liangxing, Zhang Qian</i>	113/1
	VISCOUS VORTEX RECONNECTION IN THE CAVITATING SWIRL FLOW <i>Tsoy Mikhail, Shtork Sergey, Alekseenko Sergey, Kuibin Pavel</i>	113/2
Thu 11:30-12:50	<b>Modelling of Multiphase Flows</b> <b>Room: Affari-2.D</b> DROPLET CONDENSATION/EVAPORATION IN A TURBULENT FREE SHEAR FLOW: MIXING-INDUCED SUPERSATURATION AND INERTIAL DROPLET DYNAMICS AT THE TURBULENT/NON-TURBULENT INTERFACE <i>Keedy Ryan, Aliseda Alberto</i>	113/3
	MULTIPLE HOLDUP PROBLEM IN DEVELOPING LAMINAR-LAMINAR STRATIFIED FLOW IN AN INCLINED CHANNEL <i>Thibault Daniel, Line Alain, Belt Roel</i>	113/4
	A PRESSURE DISCONTINUOUS CONTROL VOLUME FINITE ELEMENT METHOD FOR MULTIPHASE POROUS MEDIA FLOW WITH DYNAMIC MESH OPTIMIZATION AND PHASE CHANGE <i>Salinas Pablo, Pavlidis Dimitrios, Xie Zhihua, Pain Christopher, Jackson Matthew</i>	113/5
	STATISTICAL MODELLING OF TURBULENCE INTERFACE INTERACTIONS WITH THE CONSERVATIVE LEVEL-SET METHOD <i>Waclawczyk Tomasz</i>	113/6
Thu 11:30-12:50	<b>Reactive Multiphase Flows</b> <b>Room: Affari-2.C</b> NUMERICAL SIMULATION OF BUBBLY FLOWS IN AN AERATION TANK WITH BIOCHEMICAL REACTIONS <i>Khateeb Noor Ul Huda, Takagi Shu, Gong Xiaobo, Matsumoto Yoichiro</i>	114/1
	MODELLING OF MASS TRANSFER AND CHEMICAL REACTIONS IN BUBBLE COLUMN REACTORS USING OPENFOAM <i>Hlawitschka Mark W., Drefenstedt Sebastian, Schafer Jan, Bart Hans-Jorg</i>	114/2
	MODELLING OF SLURRY STAGED BUBBLE COLUMN FOR CO <sub>2</sub> CAPTURE IN CARBONATES SOLUTION WITH ENZYME CATALYSIS <i>Marzocchella Antonio, Russo Maria Elena, Bareschino Piero, Olivieri Giuseppe, Chirone Riccardo, Salatino Piero</i>	114/3
	INFLUENCE OF BUBBLE BOUNCING ON MASS TRANSFER AND CHEMICAL REACTION <i>Timmermann Jens, Hoffmann Marko, Schlueter Michael</i>	114/4

Thu 11:30-12:50	<b>Turbulence in Multiphase Flows</b> <i>Room: Villa-Sala Verde</i> BUBBLE GENERATED TURBULENCE IN TWO FLUID SIMULATION OF BUBBLE FLOW <i>Schwarz Mark Philip, Feng Yuqing</i> 114/5 THE WAKE EFFECT OF SOLID PARTICLES AND COMBUSTING DROPLETS ON TURBULENCE <i>Zhou Lixing, Li Ke, Zeng Zhuoxiong</i> 114/6 DRAG REDUCTION IN DISPERSED MULTIPHASE TAYLOR-COUPETTE TURBULENCE: A NUMERICAL APPROACH <i>Spandan Vamsi Arza, Verzicco Roberto, Lohse Detlef</i> 115/1 DRAG REDUCTION IN TURBULENT PIPE FLOW INDUCED BY SUPERHYDROPHOBIC SURFACE <i>Costantini Roberta, Battista Francesco, Casciola Carlo Massimo</i> 115/2
Thu 11:30-12:50	<b>CFD modelling of multiphase flow</b> <i>Room: Affari-2.B</i> EULERIAN MULTI-FLUID SIMULATIONS OF LOCAL LIQUID SPREADING IN A TRICKLE BED <i>Buwa Vivek, Jindal Arpit</i> 115/3 A NOVEL MULTISCALE MODELLING APPROACH FOR FLOW THROUGH A CYLINDRICAL PACKED-BED REACTOR FILLED WITH POROUS NON-SPHERICAL PARTICLES <i>Das Saurish, Kuipers J. A. M (hans), Deen Niels G.</i> 115/4 FREE SURFACE FLOW IMPROVEMENTS FOR POOR MESH <i>Gupta Vinay Kumar, Puneekar Hemant, Kvss Srikanth</i> 115/5 EULERIAN MODELLING OF LIQUID FLOW REDISTRIBUTION IN A RANDOM PACKED BED <i>Asendrych Dariusz, Niegodajew Pawel</i> 115/6
Thu 11:30-12:50	<b>Fiber suspension flow (focus session)</b> <i>Room: Affari-2.A</i> SIMULATION OF FIBER SUSPENSION IN A TURBULENT MIXING LAYER WITH AN EULER-LAGRANGIAN METHOD <i>Zhou Kun, Yang Wei</i> 116/1 TURBULENT DISPERSION OF ELLIPSOIDAL PARTICLES IN A WALL-BOUNDED TURBULENT FLOW <i>Ouchene Rafik, Hamidouche Souria, Arcen Boris, Khalij Mohammed, Taniere Anne</i> 116/2 MODELLING OF FIBRE SUSPENSIONS IN TURBULENT FLOW: A PSEUDO-HOMOGENEOUS APPROACH <i>Rasteiro Maria Graca, Cotas Carla, Gracia Fernando, Asendrych Dariusz</i> 116/3 MRI OF CELLULOSE FIBRE SUSPENSIONS IN A STRAIGHT PIPE: A NEW METHODOLOGY FOR MEASURING THE REYNOLDS STRESS COMPONENTS <i>Mackenzie Jordan, Soderberg Daniel, Lundell Fredrik, Swerin Agne</i> 116/4
Thu 11:30-12:50	<b>Finite size particles/droplets</b> <i>Room: Affari-1.A</i> FINITE-SIZE PARTICLE TRANSPORT BY TURBULENT CHANNEL FLOW <i>Vinkovic Ivana, Yu Wenchao, Buffat Marc</i> 116/5 EFFECT OF REYNOLDS NUMBER AND CONCENTRATION ON MODULATION OF TURBULENCE BY FINITE SIZE NEUTRALLY BUOYANT PARTICLES <i>Wang Guquan, Abbas Micheline, Climent Eric</i> 116/6 PARTICLE RESOLVED SIMULATION OF A 3D PERIODIC COUPETTE DENSE FLUID-PARTICLE FLOW <i>Scorsim Oliver, Fede Pascal, Simonin Olivier</i> 117/1 ANALYZING THE RELATION BETWEEN FINITE-SIZE RESOLVED PARTICLES AND COHERENT STRUCTURES IN FORCED HOMOGENEOUS-ISOTROPIC TURBULENCE <i>Uhlmann Markus, Chouippe Agathe</i> 117/2

## PARALLEL SESSIONS XV

Thu 15:10-16:30	<b>Colloidal and Suspension Dynamics</b> <i>Room: Affari-3.B</i> DYNAMICS OF SHEAR-INDUCED MIGRATION OF SPHERICAL PARTICLES IN PIPE FLOW <i>Guazzelli Elisabeth, Snook Braden, Butler Jason</i> 117/3 SHEAR FLOW-INDUCED ENHANCEMENT IN EFFECTIVE THERMAL CONDUCTIVITY OF NANOPARTICLE SUSPENSIONS <i>Sun Chengzhen, Lu Wenqiang, Bai Bofeng</i> 117/4 EVIDENCE OF SHEAR-INDUCED MIGRATION IN A PIPE FLOW OF A DENSE EMULSION <i>Abbas Micheline, Pouplin Amelie, Masbernat Olivier, Line Alain, Decarre Sandrine</i> 117/5 DYNAMICS OF A SHEAR-INDUCED AGGREGATION PROCESS BY A COMBINED MONTE CARLO - STOKESIAN DYNAMICS APPROACH <i>Frungieri Graziano, Vanni Marco</i> 117/6
-----------------	---

Thu 15:10-16:30	<b>Computational Techniques for Multiphase Flows</b> <b>Room: Villa-Auditorium</b> EVALUATION OF GENTOP CONCEPT FOR THE BUOYANCY-DRIVEN MOTION OF A SINGLE BUBBLE <i>Schlegel Fabian, Gauss Frederike, Krepper Eckhard, Lucas Dirk</i> 118/1 AN IMPLICIT HIGH ORDER DISCONTINUOUS GALERKIN LEVEL SET METHOD FOR TWO-PHASE FLOW PROBLEMS <i>Ta Thi Thanh Mai, Pigeonneau Franck, Saramito Pierre</i> 118/2 DETAILED NUMERICAL SIMULATION OF SWIRLING PRIMARY ATOMIZATION USING LEVEL SET METHOD <i>Shao Changxiao, Luo Kun, Fan Jianren</i> 118/3 A MULTI-SCALE EULERIAN-LAGRANGIAN METHOD FOR EFFICIENT DNS SIMULATION OF THE ASSISTED ATOMIZATION OF A PLANAR LIQUID SHEET <i>Estivalezes Jean-Luc, Zuzio Davide</i> 118/4
Thu 15:10-16:30	<b>Fluidization</b> <b>Room: Affari-3.A</b> DIRECT NUMERICAL SIMULATION OF FLUIDIZATION OF SOLID PARTICLES DUE TO LIFT FORCES <i>Maitri Rohit, Peters Frank, Padding Johan, Kuipers J. A. M (hans)</i> 118/5 WALL BOUNDARY CONDITIONS FOR NUMERICAL SIMULATION OF BUBBLING FLUIDIZED BEDS <i>Haghighi Mohammad Reza, Bergstrom Donald J., Spiteri Raymond</i> 118/6 LAGRANGIAN MEASUREMENT OF FORCE ON COARSE OBJECTS IN GAS-SOLID FLUIDIZED BED <i>Yoshimori Wataru, Ikegai Tomoki, Higashida Kyohei, Rai Kenta, Tsuji Takuya, Oshitani Jun, Harada Shusaku</i> 119/1 EVALUATION OF FLOWABILITY OF COHESIVE POWDER USING POWDER DISCHARGE METHOD BY AIR FLOW <i>Ogata Koichiro, Ouchi Koki, Fujita Seiji</i> 119/2
Thu 15:10-16:30	<b>Fluid-Structure Interactions</b> <b>Room: Affari-3.D</b> COMPARATIVE STUDIES OF SHOCK INTERACTION WITH STATIC AND MOVING CYLINDERS THROUGH IMMERSED BOUNDARY METHOD <i>Luo Yujuan, Luo Kun, Jin Tai, Fan Jianren</i> 119/3 INVESTIGATION OF GAS-LIQUID TWO-PHASE FLOW INDUCED VIBRATION CHARACTERISTICS <i>Miwa Shuichiro, Liu Yang, Takashi Hibiki, Mori Michitsugu</i> 119/4 A FLUID-STRUCTURE INTERACTION SOLVER COUPLED TO A VOLUME OF FLUID METHOD <i>Cerroni Daniele, Fancellu Luca, Manservigi Sandro, Menghini Filippo, Scardovelli Ruben</i> 119/5 MULTIPHASE FORCES ON BEND STRUCTURES <i>Nennie Erik, Belfroid Stefan</i> 119/6
Thu 15:10-16:30	<b>Industrial Applications</b> <b>Room: Affari-1.A</b> LAY DOWN SIMULATION OF VISCOELASTIC FLUIDS USING THE HYBRID IMMERSED-BOUNDARY METHOD <i>Mark Andreas, Ingelsten Simon, Edelvik Fredrik</i> 120/1 EFFECTS OF AZIMUTHAL ANGLE OF AERATION HOLES ON FLOWS INSIDE AND OUTSIDE A BUBBLE DIFFUSER PIPE <i>Sato Ryo, Miyayoshi Tatsuya, Hayashi Kosuke, Tomiyama Akio</i> 120/2 INVESTIGATION OF AIR INJECTION PARAMETERS TOWARD OPTIMUM FUEL SAVING EFFECT FOR SHIPS <i>Lee Inwon, Park Seong Hyeon, Chun Ho Hwan</i> 120/3 VALIDATION OF CFD PREDICTION OF SLOSHING EFFECTS FOR NUCLEAR REACTOR SEISMIC SAFETY <i>Myrillas Konstantinos, Planquart Philippe, Buchlin Jean-Marie, Schyns Marc</i> 120/4
Thu 15:10-16:30	<b>Instabilities</b> <b>Room: Affari-3.C</b> RECONNECTION OF CAVITATING VORTEX TUBES WITH FORMATION OF VORTEX RINGS IN A SWIRL FLOW <i>Skripkin Sergey, Kuibin Pavel, Shtork Sergey, Alekseenko Sergey</i> 120/5 LIGAMENTS FORMATION IN CO-FLOWING JETS WITH VISCOSITY STRATIFICATION <i>Loiseau Jean-Christophe, Tammisola Outi, Brandt Luca</i> 120/6 JUMPING HOOPS ON LIQUID <i>Yang Eunjin, Kim Ho-Young</i> 121/1 SIMULATIONS OF A SHOCK WAVE - PARTICLE CLOUD INTERACTION IN A SHOCK TUBE <i>Rollin Bertrand, Annamalai Subramanian, Ouellet Frederick, Koneru Rahul</i> 121/2

Thu 15:10-16:30	<b>Micro- and Nano-Scale Multiphase Flows</b> <b>Room: Affari-2.D</b> INFLUENCE OF DC ELECTRIC FIELD UPON THE PRODUCTION OF OIL-IN-WATER-IN-OIL DOUBLE EMULSIONS IN MM-SCALE CHANNELS. <i>Alberini Federico, Enjalbert Romain, Van Crombrugge Yann, Dapelo Davide, Simmons Mark</i> 121/3 OIL-WATER PLUG FLOW FORMATION IN SMALL CHANNELS <i>Chinaud Maxime, Roumpea Evangelia, Kahouadji Lyes, Matar Omar K., Angeli Panagiota</i> 121/4 EFFECT OF SOLID FOAM ON SEGMENTATION FOR CO <sub>2</sub> DISOLUTION FOR MILLI-CHANNEL TAYLOR FLOW <i>Simoens Serge, Zhou Guanyu, Philippe Regis, Zanotta Marie-Line, Valiorgue Pierre</i> 121/5 EXPERIMENTAL INVESTIGATION OF AIR WATER HYDRODYNAMICS IN PEMFC MICRO GAS CHANNELS <i>Malhotra Sneha, Ghosh Sumana</i> 121/6
Thu 15:10-16:30	<b>Reactive Multiphase Flows</b> <b>Room: Affari-2.C</b> DIRECT NUMERICAL SIMULATION OF A SINGLE RISING BUBBLE WITH INTERFACIAL SPECIES TRANSFER AND CHEMISORPTION IN THE LIQUID PHASE <i>Bezecny Daniel, Deising Daniel, Eller Sebastian, Marschall Holger, Schlueter Michael, Bothe Dieter</i> 122/1 WHAT IS THE EFFECT OF TURBULENCE ON THE HETEROGENEOUS REACTION RATES IN THE HIGH AND LOW DAMKÖHLER NUMBER REGIMES? <i>Krueger Jonas, Haugen Nils Erland L., Lovas Terese</i> 122/2 LARGE-EDDY SIMULATION OF SODIUM RELEASE FROM PULVERIZED-COAL JET FLAMES <i>Wan Kaidi, Xia Jun, Wang Zhihua, Wrobel Luiz, Cen Kefa</i> 122/3 FLOW PATTERN VISUALIZATION OF HEATED KEROSENE AT SUB- AND SUPER-CRITICAL PRESSURES <i>Liu Zhaohui, Pan Hui, Feng Song, Yang Zhuqiang, Wang Fei, Bi Qincheng</i> 122/4
Thu 15:10-16:30	<b>Turbulence in Multiphase Flows</b> <b>Room: Villa-Sala Verde</b> EULERIAN MULTIPHASE PREDICTIONS OF TURBULENT BUBBLY FLOW IN A STEP-EXPANSION CHANNEL <i>Papoulias Dimitrios, Splawski Andrew, Vichansky Alexander, Lo Simon</i> 122/5 DISPERSION OF FINITE SIZE DROPLETS AND SOLID PARTICLES IN ISOTROPIC TURBULENCE <i>Rosso Michele, Wang Hengjie, Elghobashi Said</i> 122/6 LARGE-EDDY SIMULATION OF PARTICLE TRANSPORT IN A VOLCANIC PLUME <i>Suto Hitoshi, Hattori Yasuo, Toshida Kiyoshi</i> 123/1 DNS OF A TURBULENT STEAM/WATER BUBBLY FLOW IN A VERTICAL CHANNEL <i>Bois Guillaume, Mathieu Benoit, Fauchet Gauthier, Toutant Adrien</i> 123/2
Thu 15:10-16:30	<b>CFD modelling of multiphase flow</b> <b>Room: Affari-2.B</b> CFD MODELLING OF LIQUID-LIQUID SLUG FLOW IN CAPILLARIES <i>Figuroa Rosette Julieth, Burns Alan</i> 123/3 A MULTI-FLUID MODELLING APPROACH FOR THE AIR ENTRAINMENT OF VENTILATED PARTIAL CAVITY <i>Xiang Min, Tu Jiyuan, Liu Longbin</i> 123/4 CFD MODELLING OF DOWNWARD TWO PHASE PIPE FLOW <i>Krepper Eckhard, Lucas Dirk, Rzehak Roland</i> 123/5 SIMULATION OF TWO PHASE FLOW INSIDE AIR CONDITIONING EVAPORATORS <i>Schlotke Jan, Kuhnelt-Wolf</i> 123/6
Thu 15:10-16:30	<b>Fiber suspension flow (focus session)</b> <b>Room: Affari-2.A</b> RHEOLOGICAL CHARACTERIZATION OF SUSPENSIONS OF RIGID FIBERS: THE STRONG EFFECT OF CONFINEMENT <i>Butler Jason, Snook Braden, Guazzelli Elisabeth, Pouliquen Olivier</i> 124/1 FLOW RHEOLOGY OF FIBER-LADEN AQUEOUS FOAMS <i>Koponen Antti, Selenius Pasi, Jasberg Ari</i> 124/2 DYNAMICS AND RHEOLOGY OF A DILUTE VISCOELASTIC SUSPENSION OF SPHEROIDS IN AN UNBOUNDED SHEAR FLOW <i>D'Avino Gaetano, Greco Francesco, Maffettone Pier Luca</i> 124/3 BROWNIAN DIFFUSION OF NANO-FIBERS <i>Tian Lin, Ahmadi Goodarz, Tu Jiyuan</i> 124/4

## PARALLEL SESSIONS XVI

Thu 16:50-18:10	<b>Colloidal and Suspension Dynamics</b>	
	<i>Room: Affari-3.B</i>	
	INERTIA-DRIVEN MIGRATION AND MIXING OF PARTICLES IN A SUSPENSION CHANNEL FLOW	
	<i>Loisel Vincent, Abbas Micheline, Masbernat Olivier, Climent Eric</i>	124/5
	SETTLING VELOCITY AND MICROSTRUCTURE OF SOLID PARTICLES IN SUSPENSIONS SIMULATIONS WITH LATTICE-BOLTZMANN METHOD	
Thu 16:50-18:10	<i>Huang Yi, Li Jing, Zha Lu, Liu Zhaohui, Zheng Chuguang</i>	124/6
	THE MOTION OF A CLOUD OF SOLID SPHERICAL PARTICLES FALLING IN A CELLULAR FLOW FIELD AT LOW STOKES NUMBER	
	<i>Marchetti Benjamin, Bergougnoux Laurence, Bouchet Gilles, Guazzelli Elisabeth</i>	125/1
	TUMBLING DYNAMICS OF NON-SPHERICAL PARTICLES FALLING IN A TURBULENT FLOW	
	<i>Gustavsson Kristian, Mehlig Bernhard</i>	125/2
Thu 16:50-18:10	<b>Computational Techniques for Multiphase Flows</b>	
	<i>Room: Villa-Auditorium</i>	
	FIDELITY REQUIREMENTS FOR THE LARGE EDDY SIMULATION OF DIESEL SPRAY	
	<i>Doisneau Francois, Arienti Marco, Oefelein Joseph</i>	125/3
	SIMULATION OF EVAPORATING DROPLETS WITHIN A DISCONTINUOUS GALERKIN MULTI-SCALE FRAMEWORK	
Thu 16:50-18:10	<i>Hitz Timon, Fechter Stefan, Munz Claus-Dieter</i>	125/4
	THERMOCAPILLARY DROPLET ACTUATION ON A WALL	
	<i>Fath Anja, Bothe Dieter</i>	125/5
	IMPLEMENTATION OF THE IMPLICIT-CONTINUOUS-EULERIAN SCHEME FOR MULTIPHASE FLOWS IN THE PARALLEL DUNE FRAMEWORK	
	<i>Fochesato Christophe, Zabiego Magali</i>	125/6
Thu 16:50-18:10	<b>Fluidization</b>	
	<i>Room: Affari-3.A</i>	
	EFFECT OF FRICTIONAL PACKING LIMIT ON NUMERICAL SIMULATION OF HYDRODYNAMICS OF DENSE GAS-SOLID FLOWS	
	<i>Sahu Akhilesh Kumar, Vasudevan Raghavan, Bhamidi Prasad</i>	126/1
	THE DENSE DISCRETE PHASE MODEL FOR SIMULATION OF BUBBLING FLUIDIZED BEDS: VALIDATION AND VERIFICATION	
Thu 16:50-18:10	<i>Cloete Schalk, Amini Shahriar</i>	126/2
	FLUID PERMEABILITY THROUGH FIBROUS BED WITH TORTUOUS STRUCTURE	
	<i>Otomo Ryoko, Kazuki Mori, Sho Sakai</i>	126/3
	MACROSCOPIC MODELLING OF PARTICLE FLUCTUANT KINETIC ENERGY IN A LIQUID FLUIDIZED BED	
	<i>Ozel Ali, Brandle De Motta Jorge Cesar, Masbernat Olivier, Simonin Olivier</i>	126/4
Thu 16:50-18:10	<b>Fluid-Structure Interactions</b>	
	<i>Room: Affari-3.D</i>	
	MEASUREMENT OF LIQUID FILM THICKNESS IN TWO PHASE FLOW OF HFC134A GAS-ETHANOL LIQUID IN A VERTICAL TUBE	
	<i>Mori Shoji, Tanaka Yuta, Okuyama Kunito</i>	126/5
	DIRECT NUMERICAL SIMULATIONS OF TURBULENT BOUNDARY LAYER OVER HEMISPHERICAL ROUGH WALL	
Thu 16:50-18:10	<i>Liu Xiaofei, Zhao Hui, Wei Anyang, Luo Kun, Fan Jianren</i>	126/6
	LIQUID-GAS DYNAMIC SYSTEM UNDER VIBRATION	
	<i>O Hern Timothy, Torczynski John, Clausen Jonathan</i>	127/1
	THE SWIRLING ANNULAR TWO-PHASE FLOW INSIDE A CIRCULAR PIPE - INTERFACIAL PHENOMENA, FLOW MECHANISM AND MODELLING	
	<i>Bai Bofeng, Liu Wen, Liu Li</i>	127/2
Thu 16:50-18:10	<b>Micro- and Nano-Scale Multiphase Flows</b>	
	<i>Room: Affari-2.D</i>	
	STEADY FILM FLOW OVER 2D TOPOGRAPHY WITH AIR INCLUSION	
	<i>Varchanis Stylianos, Pettas Dionisios, Dimakopoulos Yiannis, Tsamopoulos John</i>	127/3
	ON THE EFFECTS OF INERTIA IN TAYLOR FLOW IN RECTANGULAR MILLI-CHANNELS OF MODERATE ASPECT RATIO	
Thu 16:50-18:10	<i>Legendre Dominique, Abadie Thomas, Aubin Joelle</i>	127/4
	THE INFLUENCE OF FLUID VISCOSITY ON SIZE-BASED PARTICLE SEPARATION IN THE MICROCHANNEL WITH SYMMETRIC SHARP CORNER STRUCTURES	
	<i>Fan Liang-Liang, He Xukun, Niu Xiao-Juan, Zhe Jiang, Zhao Liang, Yan Qing</i>	127/5

	DYNAMICS OF NEUTRALLY BUOYANT SPHERES IN OBLIQUE MICRO-BIFURCATION CHANNELS <i>Medhi Bhaskar, Agrawal Vipin, Singh Anugrah</i>	127/6
Thu 16:50-18:10	<b>Non-Newtonian Multiphase Flows</b> <i>Room: Affari-3.C</i> YIELDING THE YIELD-STRESS ANALYSIS: A STUDY FOCUSED ON THE EFFECTS OF ELASTICITY ON THE SETTLING OF A SINGLE PARTICLE <i>Fraggedakis Dimitrios, Dimakopoulos Yiannis, Tsamopoulos John</i>	128/1
	SIMULATION OF DROPLET JETTING OF A NON-NEWTONIAN MIXED SUSPENSION <i>Svensson Martin, Mark Andreas, Martensson Gustaf, Karawajczyk Andrzej, Edelvik Fredrik</i>	128/2
	REMOVING DRILLING MUD IN CEMENTING OPERATIONS: A FAMILY OF CHALLENGING DISPLACEMENT FLOWS <i>Frigaard Ian, Roustaei Ali, Jeon Jaewoo, Zare Marjan, Taghavi Mohammad, Kamran Alba</i>	128/3
Thu 16:50-18:10	<b>Reactive Multiphase Flows</b> <i>Room: Affari-2.C</i> DEVELOPMENT OF A NEW OPENFOAM VOF-BASED SOLVER COUPLED WITH POPULATION BALANCES FOR THE SIMULATION OF EXPANDING POLYURETHANE FOAMS <i>Karimi Mohsen, Marchisio Daniele</i>	128/4
	DIRECT NUMERICAL SIMULATION OF SUPERSONIC TURBULENT SPRAY JET FLAME IN HEATED COFLOW <i>Jin Tai, Luo Kun, Fan Jianren</i>	128/5
	DIRECT NUMERICAL SIMULATION OF N-HEPTANE DROPLETS IGNITING IN A TURBULENT FLOW <i>Abdelsami Abouelmagd, Thevenin Dominique</i>	128/6
	A CFD-VOF BASED MODEL TO ADDRESS INTENSIVE PHOTOBIOREACTOR DESIGN <i>Olivieri Giuseppe, Salatino Piero, Marzocchella Antonio, Mazzei Luca, Lettieri Paola</i>	129/1
Thu 16:50-18:10	<b>Turbulence in Multiphase Flows</b> <i>Room: Villa-Sala Verde</i> THE EFFECT OF TURBULENCE ON PARTICLE IMPACTION ON A CYLINDER IN A CROSS FLOW <i>Aarnes Jorgen Roysland, Haugen Nils Erland L., Andersson Helge</i>	129/2
	A COMBINED LES-PBE-PDF METHOD FOR MODELLING POLYDISPERSED PARTICLE FORMATION IN TURBULENT REACTING FLOWS <i>Sewerin Fabian, Rigopoulos Stelios</i>	129/3
	TURBULENT THERMAL CONVECTION INDUCED BY THE HEATING OF A PARTICLE SUSPENSION <i>Zamansky Remi, Coletti Filippo, Massot Marc, Mani Ali</i>	129/4
	DNS OF CO-CURRENT TURBULENT STRATIFIED TWO-PHASE PIPE FLOW <i>Breugem Wim-Paul, Niazi Ardekani Mehdi, Birvalski Milos, Tummers Mark, Henkes Ruud</i>	129/5
Thu 16:50-18:10	<b>CFD modelling of multiphase flow</b> <i>Room: Affari-2.B</i> EVAPORATION IN SUPERSONIC CO <sub>2</sub> EJECTORS: ANALYSIS OF THEORETICAL AND NUMERICAL MODELS <i>Giacomelli Francesco, Mazzelli Federico, Milazzo Adriano</i>	129/6
	A PHYSICALLY CONSISTENT SUBGRID SCALE BOILING APPROACH FOR MULTIPHASE CFD <i>Baglietto Emilio</i>	130/1
	CFD STUDY OF WATER DROPLETS IN A TOP ROTATING WALL DOUGHNUT CELL <i>Verdin Patrick, Brown Lloyd</i>	130/2
	TOWARDS A MORE GENERAL CFD MODELING FOR ALL FLOW REGIMES <i>Mimouni Stephane, Fleau Solene, Vincent Stephane</i>	130/3
Thu 16:50-18:10	<b>Fiber suspension flow (focus session)</b> <i>Room: Affari-2.A</i> ENERGY EFFICIENT HIGH SOLIDS PROCESSING OF FIBER SUSPENSIONS USING MRI <i>Cardona Maria, Tozzi Emilio, Karuna Nardrapee, Jeoh Tina, Powell Robert, Mccarthy Michael</i>	130/4
	FIBERS SETTLING IN CELLULAR FLOW FIELD <i>Lopez Diego, Bergougnoux Laurence, Bouchet Gilles, Guazzelli Elisabeth</i>	130/5
	EXPERIMENTAL STUDY ON DISPERSION OF SPHERICAL PARTICLES AND FIBRES IN A JET PENETRATING A CROSSFLOW <i>Pasternak Lars, Sommerfeld Martin</i>	130/6
	MODELLING OF GAS-SOLID FLOWS WITH NON-SPHERICAL PARTICLES <i>Van Wachem Berend, Curran Thomas, Evrard Fabien, Wilson Robert</i>	131/1

Thu 16:50-18:10

**Finite size particles/droplets**

*Room: Affari-1.A*

MODELLING THE MOTION OF FINITE-SIZE PARTICLES NEAR A MOVING WALL BY A TWO-WAY COUPLING APPROACH

*Romano Francesco, Kuhlmann Hendrik*

131/2

NUMERICAL SIMULATION OF COLLISION DYNAMICS FOR FINITE SIZE PARTICLES IN TURBULENT SUSPENSION

*Vincent Stephane, Brandle De Motta Jorge Cesar, Climent Eric, Estivalezes Jean-Luc*

131/3

TRANSLATIONAL AND ROTATIONAL DYNAMICS OF FINITE-SIZED LIGHT SPHERES IN TURBULENCE

*Mathai Varghese, Sun Chao, Lohse Detlef*

131/4

MODELING HEAT TRANSFER IN REACTING GAS-SOLID FLOW USING PARTICLE-RESOLVED DIRECT NUMERICAL SIMULATION

*Subramaniam Shankar, Sun Bo, Tenneti Sudheer*

131/5

## FRIDAY, May 27, 2016

### PARALLEL SESSIONS XVII

Fri 9:50-11:10	<b>Colloidal and Suspension Dynamics</b>	
	<i>Room: Affari-3.B</i>	
	SETTLING OF FINITE-SIZE PARTICLES IN TURBULENT ENVIRONMENTS AT DIFFERENT GALILEO NUMBERS	
	<i>Fornari Walter, Picano Francesco, Brandt Luca</i>	131/6
	SCALE INDEPENDENCE OF COLLECTIVE PARTICLE SETTLING IN FRACTAL CHANNELS	132/1
Fri 9:50-11:10	<b>Fluidization</b>	
	<i>Room: Affari-3.A</i>	
	VALIDATION OF THE TWOPHASEEULERFOAM SOLVER FOR JET INLET FLUIDIZED BEDS	132/3
	<i>Tretola Giovanni, Marra Francesco Saverio</i>	
	A DISCRETE PARTICLE MODEL FOR MEMBRANE ASSISTED FLUIDIZED BED REACTORS	132/4
Fri 9:50-11:10	<b>Micro- and Nano-Scale Multiphase Flows</b>	
	<i>Room: Affari-2.D</i>	
	AEROSOL SAMPLING OF SUB-MICRON PARTICLES USING A PORTABLE ELECTROSTATIC IONIZER: EFFECT OF BULK FLOW VELOCITY ON COLLECTION EFFICIENCY	133/1
	<i>Ramin Imani Jajarmi, Etienne Robert, Laila Ladhani, Van Der Wijngaart Wouter</i>	
	SIMULATION OF TWO-PHASE FLOWS IN MICROSCOPIC NETWORKS OF INTERCONNECTED PORES TO IMPROVE MANUFACTURING PROCESS OF CERAMIC MATRIX COMPOSITES (CMCS).	133/2
Fri 9:50-11:10	<b>Non-Newtonian Multiphase Flows</b>	
	<i>Room: Affari-3.C</i>	
	HYDRODYNAMIC NON-NEWTONIAN LIQUID-SOLID FLOW IN COMPOUND TAYLOR-COUETTE FLOW	133/5
	<i>Aleksandrov Anatoly, Devisilov Vladimir, Sharay Elena</i>	
	DYNAMICS OF A RISING BUBBLE IN A VISCOELASTIC LIQUID	133/6
Fri 9:50-11:10	<b>Turbulence in Multiphase Flows</b>	
	<i>Room: Affari-2.A</i>	
	TWO-FLUID MODEL PREDICTION OF GAS-LIQUID FLOW IN A VERTICAL PIPE USING AN ALGEBRAIC SECOND MOMENT CLOSURE	134/3
	<i>Adoo N.A., Islam A. S. M. Atiqul, Bergstrom Donald J.</i>	
	PREDICTION OF GAS VOLUME FRACTION FOR POLY-DISPERSE GAS-LIQUID TURBULENT FLOW IN A VERTICAL PIPE	134/4
	<i>Islam A. S. M. Atiqul, Adoo N.A., Bergstrom Donald J.</i>	



	EXPERIMENTAL STUDY ON STRUCTURE OF A DILUTE GAS-PARTICLE AXISYMMETRIC OPPOSED JETS FLOW <i>Li Jing, Wang Hanfeng, Liu Zhaohui, Zheng Chuguang</i>	134/5
	TURBULENT STRUCTURES IN LEAD-BISMUTH FLOWS MEASURED BY USING INTRUSIVE PROBE METHODS <i>Ariyoshi Gen, Ito Daisuke, Saito Yasushi, Mishima Kaichiro</i>	134/6
<b>Fri 9:50-11:10</b>	<b>CFD modelling of multiphase flow</b> <i>Room: Affari-2.B</i>	
	CFD MODELLING OF POLYDISPERSE BUBBLY FLOWS <i>Vichansky Alexander</i>	135/1
	HIGH-ORDER MOMENT-CONSERVING METHOD OF CLASSES IN CFD CODE. <i>Jama Mohamed Ali, Buffo Antonio, Ville Alopaeus</i>	135/2
	CFD MODELLING OF PHASE SEPARATION AND COALESCENCE IN A MICELLAR THREE-PHASE SYSTEM <i>Wagner Lisa-Maria, Hohl Lena, Medeiros De Souza Luis, Thevenin Dominique, Kraume Matthias</i>	135/3
	CFD STUDY ON AGGLOMERATION IN A COUNTER-CURRENT SPRAY DRYING TOWER DUE TO PARTICLE-WALL INTERACTIONS <i>Ali Muzammil, Mahmud Tariq, Heggs Peter, Ghadiri Mojtaba, Bayly Andrew, Martindejuan Luis</i>	135/4
<b>Fri 9:50-11:10</b>	<b>Finite size particles/droplets</b> <i>Room: Affari-2.C</i>	
	FULLY RESOLVED SIMULATION OF FLUID FLOWS WITH SUSPENDED PARTICLES <i>Prosperetti Andrea, Sierakowski A., Willen D., Wang Y.</i>	135/5
	FULLY RESOLVED NUMERICAL SIMULATION OF INTER-PHASE HEAT TRANSFER IN GAS-SOLID TURBULENT FLOW <i>Xia Junjie, Luo Kun, Fan Jianren</i>	135/6
	DNS ANALYSIS OF WEAKLY-COMPRESSIBLE VAPORIZING TURBULENT TWO-PHASE FLOWS <i>Duret Benjamin, Menard Thibaut, Reveillon Julien, Demoulin Francois-Xavier</i>	136/1
	DIRECT NUMERICAL SIMULATION WITH HEAT TRANSFER OF GAS-PARTICLE-MULTIPHASE FLOW CONTAINING SHOCK WAVES BY IMMERSED BOUNDARY METHOD <i>Mizuno Yusuke, Takahashi Shun, Nonomura Taku, Nagata Takayuki, Fukuda Kota</i>	136/2

## PARALLEL SESSIONS XVIII

<b>Fri 11:30-12:50</b>	<b>Computational Techniques for Multiphase Flows</b> <i>Room: Affari-2.B</i>	
	SHOCK-INDUCED COLLAPSE OF A SINGLE CAVITATION BUBBLE ATTACHED TO AN ELASTIC-PLASTIC SOLID <i>Turangan Cary</i>	136/3
	NUMERICAL INVESTIGATIONS OF SHOCK-INDUCED FLOW PATTERNS IN INHOMOGENEOUS MEDIA <i>Nowakowski Andrzej, Nicolleau Franck</i>	136/4
	A COMPUTATIONAL MODELLING APPROACH TO MINIMIZE PARAMETER SPACE AND GEOMETRY ADAPTABILITY WHEN INVESTIGATING CLEANROOM PARTICULATE LEVELS <i>Baker Nazar, O Sullivan Paul, Kelly Ger</i>	136/5
<b>Fri 11:30-12:50</b>	<b>Droplet Flows</b> <i>Room: Affari-3.B</i>	
	EFFECTS OF ANISOKINETIC SAMPLING IN STEAM QUALITY MEASUREMENTS FOR STERILIZATION PROCESSES <i>Serra Sonia</i>	136/6
	MODULATION OF SPRAY DROPLET NUMBER DENSITY AND SIZE DISTRIBUTION BY AN ACOUSTIC FIELD <i>Achury Javier, Polifke Wolfgang</i>	137/1
	DIRECT NUMERICAL SIMULATION OF MULTI-COMPONENT DROPLET INTERACTIONS <i>Schlotke Karin, Meister Christian, Karch Grzegorz, Sadlo Filip, Ertl Thomas, Weigand Bernhard</i>	137/2
<b>Fri 11:30-12:50</b>	<b>Fluid-Structure Interactions</b> <i>Room: Affari-3.A</i>	
	MORPHODYNAMICS AROUND A SLENDER CYLINDER UNDER A WAVY BOUNDARY LAYER <i>Corvaro Sara, Brocchini Maurizio, Miozzi Massimo, Pereira Francisco Alves, Lorenzoni Carlo, Mancinelli Alessandro, Tondi Cristina</i>	137/3
	THREE-PHASE ROUGHNESS EFFECTS ON SUPERHYDROPHOBIC SURFACES FOR TURBULENT DRAG REDUCTION <i>Golovin Kevin, Gose James W., Perlin Marc, Ceccio Steven L., Tuteja Anish</i>	137/4

	FLUID DYNAMICS OF SELF-PROPELLED DROPLET JUMPING ON SUPERHYDROPHOBIC SURFACES <i>Figueiredo Patric, Wasserfall Joram, Pischke Philipp, Kneer Reinhold</i>	137/5
	ON THE INTERACTION BETWEEN WAVES AND A MODULAR OSCILLATING WAVE SURGE CONVERTER <i>Abadie Thomas, Dias Frederic</i>	137/6
<b>Fri 11:30-12:50</b>	<b>Instabilities</b> <b>Room: Affari-3.C</b> ENERGY-BASED MODEL FOR THE PREDICTION OF MINI-CHANNELS DRY-OUT CONDITION <i>Giannetti Niccolo, Saito Kiyoshi, Yamaguchi Seiichi</i>	138/1
	TOWARDS A 3D GLOBAL LINEAR STABILITY ANALYSIS FOR BUOYANT FLUID PARTICLES <i>Kromer Johannes, Bothe Dieter</i>	138/2
	THE ROLE OF VISCOSITY STRATIFICATION ON THE GLOBAL STABILITY OF TWO-PHASE JETS <i>Tammisola Outi, Loiseau Jean-Christophe, Brandt Luca</i>	138/3
<b>Fri 9:50-11:10</b>	<b>Micro- and Nano-Scale Multiphase Flows</b> <b>Room: Affari-2.D</b> NUMERICAL STUDY ON EFFECTS OF ORIENTATION AND GRAVITY ON TWO-PHASE FLOW OF MINICHANNEL OF PEM FUEL CELLS <i>Moosa Ashrafi, Mehrzad Shams, Bozorgnezhad Ali, Ahmadi Goodarz</i>	138/4
	MODELLING OF GAS-LIQUID, TWO-PHASE FLOW IN POROUS MEDIA AND CHANNELS OF A PEM WATER ELECTROLYSIS CELL USING THE EULER-EULER FRAMEWORK OF OPENFOAM <i>Olesen Anders Christian, Kaer Soren Knudsen</i>	138/5
	DYNAMICS OF GAS-LIQUID SEGMENTED FLOW IN POROUS MEDIA: MONOLITHIC OPEN CELL SOLID FOAMS VERSUS DENSE PACKED BED <i>Serres Marion, Zannota Marie-Line, Bornette Frederic, Philippe Regis, Vidal Valerie</i>	138/6
<b>Fri 11:30-12:50</b>	<b>Reactive Multiphase Flows</b> <b>Room: Affari-2.A</b> LES OF A REACTIVE, MULTI-PHASE FLOW IN THE INJECTOR NEAR-FIELD OF AN ENTRAINED-FLOW GASIFIER <i>Eckel Georg, Saenger Alexander, Hotz Christian, Kathrotia Trupti, Rachner Michael, Le Clercq Patrick, Aigner Manfred</i>	139/1
	INTERACTION OF FLUID DYNAMICS AND CHEMICAL KINETICS IN A THREE-PHASE DEHYDROGENATION REACTION <i>Heublein Norbert, Sattelmayer Thomas</i>	139/2
	NUMERICAL STUDY ON DROPLET EVAPORATION AND AUTO-IGNITION OF HYPERGOLIC PROPELLANTS <i>Tani Hiroumi, Umemura Yutaka, Terashima Hiroshi, Kanno Nozomu, Koshi Mitsuo</i>	139/3
<b>Fri 11:30-12:50</b>	<b>Finite size particles/droplets</b> <b>Room: Affari-2.C</b> FLOW TOPOLOGY AND ATTRACTORS FOR THE MOTION OF FINITE-SIZE PARTICLES IN A THREE-DIMENSIONAL STEADY CAVITY FLOW <i>Kuhlmann Hendrik, Romano Francesco, Albensoeder Stefan</i>	139/4
	TURBULENT CHANNEL FLOW WITH SUSPENDED BI-DISPERSE FINITE SIZE NEUTRALLY BUOYANT SPHERES <i>Picano Francesco, Lashgari Iman, Costa Pedro, Breugem Wim-Paul, Brandt Luca</i>	139/5
	DNS OF TURBULENT BOUNDARY LAYER WITH FULLY RESOLVED PARTICLES AT LOW VOLUME FRACTION <i>Wu Fan, Luo Kun, Fan Jianren</i>	139/6
	FULLY RESOLVED SIMULATIONS OF SUSPENSION OF SPHERICAL PARTICLES IN SHEAR FLOWS <i>Rahmani Mona, Wachs Anthony</i>	140/1



## POSTERS

### **Bubbly Flows**

- EFFECT OF A CONCENTRATION GAS BUBBLES IN THE NEAR-WALL LIQUID FLOW ON THE DRAG REDUCTION  
*Evseev Aleksei, Maltzev Leonid* 140/2
- NUMERICAL STUDY OF MASS TRANSFER OF A RISING BUBBLE WITH BOUNDARY LAYER APPROACH  
*Islam Shafiul, Kuipers J. A. M (hans), Deen Niels G.* 140/3
- MANIPULATING OF MICRO-BUBBLES USING ULTRASONIC WAVE PAIRS  
*Bao Fubing, Xiang Sujuan, Wang Yuebing* 140/4
- EFFECT OF GAS PROPERTIES ON THE CHARACTERISTICS OF A BUBBLE COLUMN EQUIPPED WITH FINE POROUS SPARGER.  
*Chatzidafni Ariadni, Passos Agathoklis, Paras Spiros, Mouza Aikaterini* 140/5
- MEASUREMENT OF BUBBLE VELOCITY, SHAPE, AND SIZE DISTRIBUTION IN AN UNDIVIDED ELECTROCHLORINATION CELL BY TELECENTRIC DIRECT IMAGE METHOD  
*Andersen Nikolaj, Hedensted Lau, Stroe Rodica-Elisabeta* 140/6
- THE FORMATION OF BUBBLES FROM THE CAPILLARY IN A LIQUID FLOW  
*Lobanov Pavel, Kashinsky Oleg, Chinak Aleksandr, Vorobyev Maxim* 141/1
- NUMERICAL INVESTIGATION ON MICROBUBBLE DYNAMIC EVOLUTION AND TURBULENCE MODULATION IN A HORIZONTAL CHANNEL BUBBLY FLOW  
*Pang Mingjun, Wei Jinjia* 141/2
- BENCHMARKING CFD AND NEW MECHANISTIC APPROACH FOR ENABLING SCALE-UP OF MULTIPHASE REACTORS VIA SOPHISTICATED MEASUREMENT AND COMPUTING TECHNIQUES  
*Al-Dahhan Muthanna* 141/3
- EXPERIMENTAL INVESTIGATION OF PRESSURE DROP OF COUNTER CURRENT AIR-WATER FLOW IN A VERTICAL COLUMN  
*Mashayekhpour Maryam, Lay Ebrahim Nemati* 141/4
- EFFECTS OF PHYSICAL & GEOMETRICAL PARAMETERS ON COALESCENCE OF TWO BUBBLES  
*Yadav Sumedh, Pradhan Aniruddhe, Das Prasanta Kumar* 141/5

### **Boiling, Condensation, Evaporation**

- MEASUREMENTS OF LOCAL HEAT TRANSFER IN SUBCOOLED POOL BOILING USING INFRARED THERMOMETRY  
*Jung Jaehoon, Kim Sung Jin, Kim Jungho, Lee Hyeseung* 141/6
- MEASUREMENTS OF 3-PENTANONE LASER INDUCED FLUORESCENCE FOR IMAGING CONCENTRATION OF AN EVAPORATE N-HEPTANE LIQUID FROM A POROUS MEDIA  
*Mouret Quentin, Kuhni Manuel, Vena Patrizio, Galizzi Cedric, Escudie Dany* 142/1
- A NEW TWO-PHASE FRICTIONAL PRESSURE DROP FOR ADIABATIC AND CONDENSING MINI/MICRO-CHANNEL FLOWS  
*Mashayekhpour Maryam, Lay Ebrahim Nemati* 142/2

### **Cavitation/Nucleation**

- MICROBUBBLE GENERATION FROM ULTRAFINE BUBBLE WATER BY ULTRASONIC STIMULATION  
*Terasaka Koichi, Harasawa Nozomi, Yamazaki Isako, Fujioka Satoko* 142/3
- APPLICATION OF A COMPRESSIBLE TWO-PHASE MIXTURE FORMULATION TO TURBULENT CAVITATING FLOWS  
*Kim Sung-Eun* 142/4

### **Collision, Agglomeration and Breakup**

- NUMERICAL SIMULATION OF IMPINGING JETS ATOMIZATION  
*Zhang Peiyu, Wang Bing* 142/5
- NUMERICAL STUDY ON HEAD-ON COLLISION OF WATER DROPLETS AT HIGH WEBER NUMBERS  
*Liu Muyuan, Bothe Dieter* 142/6
- INFLUENCE OF PARTICLE-WALL COLLISIONS AND PARTICLE AGGLOMERATION ON CYCLONE PERFORMANCE  
*Sgrott Junior Oscar, Novelletto Ricardo Guilherme A., Sommerfeld Martin* 143/1
- INFLUENCE OF MODEL ENERGY SPECTRUM AND STRUCTURE FUNCTION ON THE FLUID PARTICLE BREAKUP MODELS  
*Solsvik Jannike, Jakobsen Hugo A.* 143/2

## **Computational Techniques for Multiphase Flows**

OPTIMAL FREQUENCY OF ELECTRICAL-IMPEDANCE MEASUREMENT IN THREE-PHASE MIXTURES <i>Hernandez Cely Marlon, Gomez Bernal Sergio, Velasco Pena Hugo Fernando, Hernandez Rodriguez Oscar Mauricio</i>	143/3
REMAINING SWIRLING MOTION ON THE GAS OUTLET DUCT OF CYCLONE SEPARATORS: ALTERNATIVES TO INCREASE EFFICIENCY <i>Salvo Ricardo, De Souza Francisco Jose, Martins Diego</i>	143/4
FINITE VOLUME / FINITE AREA INTERFACE-TRACKING METHOD FOR TWO-PHASE FLOWS WITH SOLUBLE SURFACTANTS AND MASS TRANSFER <i>Pesci Chiara, Weber Paul Stephan, Falcone Manuel, Marschall Holger, Bothe Dieter</i>	143/5
A CONSERVATIVE LEVEL SET METHOD FOR TWO-PHASE FLOWS <i>Moghaddam Amin Mahmoudi, Shafieefar Mehdi, Panahi Roozbeh</i>	143/6

## **Experimental Methods for Multiphase Flows**

AN EXPERIMENTAL STUDY OF THE FLOW CHARACTERISTIC OF AIR-WATER TWO PHASE FLOW IN A SINGLE SCREW CHANNEL <i>Cai Bo, Xia Guodong, Liu Xianfei, Yang Guang</i>	144/1
EFFECT OF DISPERSED MODALITY ON TURBULENCE CHARACTERISTICS OF LIQUID-LIQUID DISPERSION IN STIRRED TANK <i>Han Jiacheng, Liu Nannan, Liu Fangyuan, Zhang Meng, Wang Wei, Gong Jing</i>	144/2
ELECTRICAL AND ULTRASONIC DUAL-MODALITY MEASUREMENT FOR OIL-WATER FLOW BASED ON KALMAN FUSION <i>Li Pengfei, Wu Hao, Tan Chao, Dong Feng</i>	144/3
MULTIPHASE FLOW MEASUREMENT BY MULTIELECTRODE ELECTROMAGNETIC FLOW METER BASED ON REGION WEIGHT FUNCTION <i>Zhao Yuyang, Li Zheng, Liang Wei</i>	144/4
GENERATOR FOR WIDE RANGE OF BUBBLE SIZE USING ACOUSTIC PRESSURE WAVE AND ELASTIC TUBE <i>Kasai Yuma, Sanada Toshiyuki</i>	144/5

## **Fluidization**

IBD – INTENSIFIED BY DESIGN <i>Mihajlovic Milan, Roghair Ivo, Van Sint Annaland Martin</i>	144/6
---	-------

## **FluidStructure Interactions**

CHARACTERISTICS OF TURBULENT BOUNDARY LAYER AFFECTED BY A SPHERE NEXT TO A WALL <i>Zhao Hui, Liu Xiaofei, Li Dong, Wu Fan, Luo Kun, Fan Jianren</i>	145/1
--	-------

## **Granular Media & Granular flow**

THE PROPAGATION OF SHOCK WAVES IN GAS-WATER-SATURATION BULK MEDIA <i>Zhurov Andrei, Akmetov Alfir, Urmancheev Said</i>	145/2
DENSITY WAVES IN THE GRAVITATIONAL FLOW OF GRANULAR MATERIAL IN NARROW PIPES <i>Alvarez Carlos, De Moraes Franklin Erick</i>	145/3

## **Industrial Applications**

UPWARD-VERTICAL TWO-PHASE FLOW IN A LARGE-DIAMETER ANNULAR DUCT <i>Almeida Colmanetti Alex Roger, Souza De Castro Marcelo, Cavallini Barbosa Marcel, Kjeldby Tor Kindsbekken, Derks Peter W. J., Hernandez Rodriguez Oscar Mauricio</i>	145/4
FOUR-WAY COUPLED SIMULATIONS OF THE GAS-SOLID FLOW IN A CYCLONE SEPARATOR <i>Martins Diego, De Souza Francisco Jose, Salvo Ricardo</i>	145/5
ANALYSIS OF TWO-PHASE FLOW IN AN CENTRIFUGAL PUMP IMPELLER <i>Sabino Renzo, Stel Henrique, Bertoldi Dalton, Morales Rigoberto</i>	145/6
A STUDY OF TWO-PHASE FLOW MALDISTRIBUTION BETWEEN CELLS OF A HIGH PRESSURE PEM WATER ELECTROLYSIS STACK VIA COMPUTATIONAL FLUID DYNAMICS <i>Gao Xin, Olesen Anders Christian, Romer Carsten, Kaer Soren Knudsen</i>	146/1
LIQUID MALDISTRIBUTION IN STRUCTURED PACKING IN AN INCLINED COLUMN: CFD SIMULATION AND VALIDATION <i>Beard Philippe, Fourati Manel, Alix Pascal, Maubert Thomas, Carlier Vincent, Courtial Xavier, Salais Clement</i>	146/2

## **Micro and NanoScale Multiphase Flows**

LIQUID INFILTRATION CHARACTERISTICS INTO CLOSED END STRUCTURES UNDER THE EXTERNAL PRESSURE <i>Yamaguchi Eri, Nozaki Hiroshi, Sanada Toshiyuki</i>	146/3
--	-------

ADSORPTION OF ALCOHOLIC ADDITIVES IN WATER/LIBR AQUEOUS SOLUTION AT LOW PRESSURE <i>Lonardi Federico, Luke Andrea</i>	146/4
--	-------

### **Mixing**

EXPERIMENT STUDY OF THE INSITU DROP SIZE DISTRIBUTIONS AT DIFFERENT SPATIAL REGIONS IN STIRRED TANK <i>Zhang Meng, Liu Nannan, Han Jiacheng, Wang Wei, Gong Jing</i>	146/5
NUMERICAL STUDY ON MIXING CHARACTERISTICS IN SPHERICAL FLASK WITH MAGNETIC STIR BAR <i>Lee Sang Hyuk, Cho Wonsam, Kim Dae-Hwan, Lee Taehyun, Ryu Kyungha</i>	146/6
INFLUENCE OF PARTICLE FORCES ON PARTICLES IN FLUID PHASE RESONANCE MIXERS <i>Schmalfuss Silvio, Sommerfeld Martin</i>	147/1

### **Modelling of Multiphase Flows**

MODELING OF SCATTERING INTENSITY OF MINERAL AEROSOLS WITH A GAUSSIAN BEAM <i>Tang Hong, Lin Jianzhong</i>	147/2
SIMULATION OF TURBULENT LIQUID-LIQUID DISPERSIONS: VALIDATION OF DIFFERENT BREAKAGE AND COALESCENCE KERNELS AND COMPARISON BETWEEN DIFFERENT MODELLING APPROACHES <i>De Bona Jeremias, Li Dongyue, Buffo Antonio, Vanni Marco, Podgorska Wioletta, Marchisio Daniele</i>	147/3
MODELING OF SCATTERING INTENSITY OF MINERAL AEROSOLS WITH A GAUSSIAN BEAM <i>Tang Hong, Lin Jianzhong</i>	147/4
A LATTICE BOLTZMANN MODEL TO SIMULATE LIQUID-LIQUID SLUG FLOW IN 90° BEND <i>Jain Shakendra, Ghosh Sumana</i>	147/5
DETERMINISTIC AND STATISTICAL MODELLING OF PARTICLE-PARTICLE COLLISIONS IN SUPERSONIC GAS-SOLID FLOWS PAST BODIES <i>Reviznikov Dmitry, Sposobin Andrey, Varaksin Aleksei</i>	147/6
ASSESSMENT OF THE EFFECT OF DYNAMIC PRESSURE CLOSURE ON THE SIMULATIONS OF VERTICAL ANNULAR GAS-LIQUID FLOW WITH THE 1D TWO FLUID MODEL <i>Gonzalez Eric, Carneiro Joao, Nieckele Angela</i>	148/1
A KINETIC WAX DEPOSITION MODEL FOR W/O EMULSION WITH THE INFLUENCE OF CRYSTALLIZED WAX ADHERING TO OIL-WATER INTERFACE <i>Song Pan, Wang Wei, Yang Juheng, Gong Jing</i>	148/2
OPTIMAL FILTER WIDTH FOR CONTINUUM MODELING OF DISCRETE PHASES <i>Bateman Samuel, Simeonov Julian, Calantoni Joseph</i>	148/3
NUMERICAL SIMULATION OF ANNULAR FLOW IN HORIZONTAL PIPES USING THE TWO FLUID MODEL <i>Souto Izabel, Nieckele Angela</i>	148/4
THE NUMERICAL MODELLING OF MULTIPHASE FLOW INSTABILITIES IN 3D CYLINDRICAL COORDINATES <i>Oud Guido</i>	148/5
MULTIPHASE SIMULATION FOR WATER COOLING TURBINE EXHAUST <i>Jiang Minyee, Vinh David</i>	148/6

### **Multiphase Flow in Heat and Mass Transfer**

A LAGRANGIAN MULTISTAGE SPRAY DRYING MODEL FOR MULTIPHASE CFD APPLICATIONS <i>Hribersek Matjaz, Sagadin Gregor, Zadravec Matej, Ravnik Jure</i>	149/1
THERMOPHORETIC DEPOSITION OF NANOPARTICLES IN A 90° SQUARE BEND TUBE <i>Yin Zhaoqin, Dai Zhongping</i>	149/2
EXPERIMENTAL STUDY AND MECHANISTIC MODELLING OF HEAT TRANSFER IN AIR-WATER SLUG FLOW UNDER COOLING PROCESS <i>Wang Xin, Dong Chuanshuai, Zhang Xiaoling, Yue Xiaoqing, Wang Zhaoting, He Limin</i>	149/3

### **NonNewtonian Multiphase Flows**

RHEOLOGICAL EFFECTS ON FLUID INVASION INTO CEMENTED ANNULI <i>Zare Marjan, Ward Matthew, Quentin Lindfield Roberts, Kamran Alba, Frigaard Ian</i>	149/4
A NOVEL VISCO-PLASTICALLY LUBRICATED METHOD FOR CORE-ANNULAR PIPELINE FLOW <i>Sarmadi Parisa, Frigaard Ian, Hormozi Sarah</i>	149/5

### **Particle and Drop Dynamics**

NUMERICAL SIMULATION OF INERTIAL MIGRATION OF ELASTIC PARTICLE IN POISEUILLE FLOWS <i>He Xukun, Fan Liang-Liang, Zhe Jiang, Zhao Liang</i>	149/6
EXPERIMENTAL STUDY ON THE COALESCENCE BEHAVIORS UNDER VARIOUS SURFACTANT CONCENTRATIONS BETWEEN DROP-INTERFACE <i>Ma Mengyu, Li Kai, Voulgaropoulos Victor, Jin Hang, Wang Wei, Gong Jing</i>	150/1

NUMERICAL INVESTIGATION OF THE GROUPING BEHAVIOUR WHEN THE PARTICLES ARE COAXIALLY SETTLING <i>Nie Deming, Chen Rongqian</i>	150/2
NUMERICAL INVESTIGATION OF HEAT TRANSFER AND FLUID FLOW BEHAVIOR AROUND AN OBLATE SPHEROID <i>Razgallah Intissar, Ouchene Rafik, Khalij Mohammed, Arcen Boris, Taniere Anne</i>	150/3
ABILITY OF ANSYS FLUENT® V 16.0 TO COMPUTE THE DRAG FORCE ACTING ON NON-SPHERICAL PARTICLES LIKE CYLINDRICAL DISKS <i>Sierra Del Rio Jorge, Taniere Anne, Khalij Mohammed, Hincapie Diego, Velez Sebastian</i>	150/4
DRIFT AND MIXING BY AIR BUBBLES CROSSING AN INTERFACE OF A STRATIFIED MEDIUM AT MODERATE RE <i>Diaz Lamberto, Ruiz Angulo Angel, Zenit Roberto</i>	150/5
NUMERICAL ANALYSIS OF MICROBUBBLE DEFORMATION NEAR A HIGH-VISCOUS LIQUID IN ULTRASOUND FIELD <i>Mori Kengo, Sugiura Toshihiko</i>	150/6
 <b>ParticleLaden Flows</b>	
A PRELIMINARY STUDY ON THE STABILITY OF PARTICLE LADEN JETS THROUGH A FULLY COUPLED CFD-DEM SOLVER <i>Peters Bernhard, Pozzetti Gabriele</i>	151/1
NANOPARTICLE COAGULATION AND DISPERSION IN A TURBULENT PLANAR JET WITH CONSTRAINTS <i>Tu Chengxu, Liu Song, Bao Fubing, Yin Zhaoqin</i>	151/2
ON THE SEGREGATION OF PARTICLE SIZE CLASSES IN BIDISPERSE GAS-SOLID FLOW: DEPENDENCE ON GAS-SOLID DRAG AND INFLUENCE ON INTERPHASE ENERGY TRANSFER DYNAMICS <i>Mehrabadi Mohammad, Subramaniam Shankar</i>	151/3
ANALYSIS OF THE EFFECT OF PARTICLE-WALL AND INTER-PARTICLE COLLISION ON EROSION USING OPENFOAM® <i>Novelletto Ricardo Guilherme A., Sgrott Junior Oscar, Lain Santiago, Sommerfeld Martin</i>	151/4
LARGE-EDDY SIMULATION TO STUDY PARTICLE DEPOSITION IN A CURVED PASSAGE <i>Sinha Nityanand, Ghodke Chaitanya, Khot Ambarish, Apte Sourabh</i>	151/5
 <b>Bubble dynamics</b>	
SIMULATION OF SINGLE RISING BUBBLES IN A LINEAR SHEAR FLOW WITH THE COLOR-GRADIENT LATTICE BOLTZMANN METHOD <i>Malli Phillip, Sommerfeld Martin</i>	151/6

# Author Index

Aarnes Jorgen Roysland	129/2, 129/2	Amini Parsa	8/2, 8/2
Abadie Thomas	127/4, 137/6, 127/4, 137/6	Amini Shahriar	16/1, 126/2, 16/1, 126/2
Abba Antonella	92/6, 92/6	Amiri Delouei Amin	4/1, 4/1
Abbagoni Baba Musa	67/2, 67/2	An Jae Sik	5/3, 5/3
Abbas Micheline	116/6, 117/5, 124/5, 132/5, 116/6, 117/5, 124/5, 132/5	Andayi Samwel	35/6, 54/2, 35/6, 54/2
Abdelsami Abouelmagd	128/6, 128/6	Andersen Nikolaj	140/6, 140/6
Abdibekov Ualikhan	99/2, 99/2	Andersson Helge	8/4, 94/6, 102/4, 129/2, 8/4, 94/6, 102/4, 129/2
Abdol Azis Mohd Hazmil Syahidy	3/4, 3/4	Andersson Ronnie	3/1, 3/1
Abdollahi Mofakham Amir	13/5, 13/5	Ando Toshitake	73/2, 96/3, 73/2, 96/3
Abdulkareem Lokman A.	82/3, 82/3	Andrade Paul	65/5, 65/5
Abe Yutaka	34/4, 44/1, 95/4, 34/4, 44/1, 95/4	Andric Jelena	109/6, 109/6
Abouali Omid	28/5, 28/5	Angeli Panagiota	24/6, 26/1, 77/2, 121/4, 24/6, 26/1, 77/2, 121/4
Aboulhasanzadeh Bahman	66/3, 66/3	Angilella Jean-Regis	102/1
Acher Thomas	41/3, 83/2, 41/3, 83/2	Anglart Henryk	88/6, 88/6
Achury Javier	137/1, 137/1	Aniszewski Wojciek	33/3, 98/2, 33/3, 98/2
Adams Nikolaus	6/4, 49/2, 6/4, 49/2	Anjos Gustavo	99/6, 99/6
Adamsen Per	102/3, 102/3	Annamalai Subramanian	121/2, 121/2
Addy Joseph	37/5, 37/5	Ansart Renaud	16/2, 100/5, 16/2, 100/5
Adebayo Idris	7/4, 7/4	Antkowiak Arnaud	28/4, 28/4
Adoo N.A.	134/3, 134/4, 134/3, 134/4	Antonyuk Sergiy	104/4, 104/4
Afkhami Shahriar	10/6, 10/6	Anupriya Sethu	45/2, 45/2
Agbaglah Gilou	97/3, 97/3	Aoki Jiro	52/5, 60/6, 52/5, 60/6
Aglave Ravindra	91/2, 91/2	Aouadi Aroua	42/6, 42/6
Agrawal Vipin	127/6, 127/6	Aoyama Shohei	55/1, 55/1
Agrawal Yogesh	8/2, 8/2	Apte Sourabh	13/4, 151/5, 13/4, 151/5
Agunlejika Ezekiel	9/6	Arai Takahiro	38/2, 38/2
Ahmad.S Thariq	74/6, 74/6	Arai Yoshihiro	95/4, 95/4
Ahmadi Goodarz	4/1, 8/5, 13/5, 28/5, 62/6, 124/4, 138/4	Arcen Boris	13/3, 76/4, 116/2, 150/3, 13/3, 76/4, 116/2, 150/3
Ahmadloo Ebrahim	99/3, 99/3	Arienti Marco	36/3, 125/3, 36/3, 125/3
Ahmed Wael	90/2, 90/2	Ariyoshi Gen	23/6, 134/6, 23/6, 134/6
Ahn Taehwan	45/3, 45/3	Arjmand Mehdi	37/3, 37/3
Aidun Cyrus K.	86/4, 86/4	Arquis Eric	133/2, 133/2
Aigner Manfred	139/1, 139/1	Arroxellas Andre	5/5, 5/5
Akesjo Anders	51/2, 51/2	Asano Kei	100/6, 100/6
Akiki Georges	71/6, 71/6	Asendrych Dariusz	115/6, 116/3, 115/6, 116/3
Akmetov Alfir	145/2, 145/2	Ashok Anand	91/4, 91/4
Akselsen Andreas	9/3, 9/3	Assylbekuly Aidar	99/2, 99/2
Aktershev Sergey	51/3, 51/3	Atta Arnab	84/5, 84/5
Al Safran Eissa	105/1, 105/1	Aubert Stephane	14/6, 14/6
Alahmadi Yaser	100/4, 100/4	Aubin Joelle	127/4, 127/4
Alahyari Abbas	56/5, 56/5	Audard Francois	73/4, 90/6, 112/4, 73/4, 90/6, 112/4
Alahyari Beig Shahab	66/3, 66/3	Augier Frederic	19/5, 75/5, 19/5, 75/5
Alaoui Yousuf	40/6, 59/3	Aussal Matthieu	58/2, 58/2
Al-Behadili Mustapha	40/2	Aussillous Pascale	73/6, 82/5, 73/6, 82/5
Albensoeder Stefan	139/4, 139/4	Avellan Francois	21/2, 21/2
Alberini Federico	89/6, 121/3, 89/6, 121/3	Avila Ricardo	5/5, 5/5
Al-Dahhan Muthanna	141/3, 141/3	Awad Mohamed	47/2, 99/3, 47/2, 99/3
Aleksandrov Anatoly	133/5, 133/5	Ayati Anis Awal	98/3, 98/3
Alekseenko Sergey	51/3, 105/4, 113/2, 120/5, 51/3, 105/4, 113/2, 120/5	Azizi Sadra	99/3, 99/3
Alexiadis Alessio	89/6, 89/6	Azizi Salar	32/2, 32/2
Ali Muzammil	135/4, 135/4	Azzi Abdelwahid	49/5
Alimoradi Hasan	25/3, 25/3	Azzolin Marco	73/3, 73/3
Alin Niklas	11/4, 11/4	Azzopardi Barry James	4/6, 9/6, 15/5, 19/3, 27/4, 27/5, 27/6, 28/2, 40/2, 40/6, 49/4, 49/5, 54/4, 58/6, 59/3, 82/3, 93/5, 104/2
Aliseda Alberto	21/5, 30/5, 44/5, 70/6, 113/3, 21/5, 30/5, 44/5, 70/6, 113/3	Baba Misaki	104/6, 104/6
Alix Pascal	39/5, 146/2, 39/5, 146/2	Babler Matthaus	64/1, 64/1
Aliyu Aliyu	36/2, 36/2	Bache Elliott	133/2, 133/2
Almeida Colmanetti Alex Roger	145/4, 145/4	Baczysmaliski Dominik	48/2, 48/2
Almeras Elise	46/2, 92/4, 46/2, 92/4	Bae Choongsik	24/3, 24/3
Almohammed Naser	39/4, 66/1, 39/4, 66/1	Bae Kyung Jin	29/4, 29/4
Alouges Francois	58/2, 58/2	Baglietto Emilio	130/1, 130/1
Alzheimer Marco	40/4, 82/2	Bai Bofeng	20/2, 78/5, 89/5, 117/4, 127/2, 20/2, 78/5, 89/5, 117/4, 127/2
Alvarez Carlos	145/3, 145/3	Bai Wei	64/4, 64/4
Alzuguir Azevedo Luis Fernando	98/3, 98/3		
Amalfi Raffaele	52/2, 52/2		
Amberg Gustav	78/2, 103/4, 78/2, 103/4		



Bai Xiao	41/5	Bharadwaj Rahul	82/4, 82/4
Baker Lucia	35/4, 35/4	Bi Qincheng	25/4, 33/6, 43/3, 122/4, 25/4, 33/6, 43/3, 122/4
Baker Nazar	136/5, 136/5	Biance Anne-Laure	15/4, 15/4
Balachandar S	71/6, 71/6	Bianchi Pedro	106/1, 106/1
Balakin Boris V.	12/4, 69/3, 74/4, 12/4, 69/3, 74/4	Biazussi Jorge Luiz	18/4, 74/3, 18/4, 74/3
Baltussen Maike	132/5, 132/5	Bieberle Andre	46/3, 46/3
Bamidele Olufemi Eytape	90/2, 90/2	Billard Jean Yves	55/4, 55/4
Banerjee Jyotirmay	12/2, 66/6	Birvalski Milos	129/5, 129/5
Banko Andrew	27/2, 35/2, 27/2, 35/2	Blanchard Ghislain	24/1, 24/1
Bannwart Antonio Carlos	18/4, 74/3, 18/4, 74/3	Blanco Stephane	17/5, 17/5
Bansal Akshay	50/3, 50/3	Blondeaux Paolo	44/2, 44/2
Bansal Hemant	6/6, 6/6	Boblest Sebastian	104/5, 104/5
Bao Fubing	140/4, 151/2, 140/4, 151/2	Bobylev Aleksey	51/3, 51/3
Baptiste Heiles	77/3, 77/3	Boden Stephan	52/4, 52/4
Bar Kohany Tali	108/1	Boden Wiebke	14/6, 14/6
Barbosa Christophe	55/2, 55/2	Bogey Christophe	58/5, 58/5
Barbuto Fausto	9/1, 9/1	Boileau Matthieu	79/1, 102/6, 79/1, 102/6
Bareschino Piero	114/3, 114/3	Bois Guillaume	123/2, 123/2
Barmak Ilya	105/3	Boivin Pierre	46/6, 46/6
Barnea Dvora	34/2, 79/4, 34/2, 79/4	Bokkers Albert	37/3, 37/3
Barnett Alejandro Oyarce	64/4, 64/4	Bolanos Jimenez Rocio	28/1, 63/3, 28/1, 63/3
Baroud Yousef	23/3	Bon Alexandre	55/4, 55/4
Barre Stephane	57/3, 57/3	Bonaventura Luca	92/6, 92/6
Bart Hans-Jorg	43/6, 114/2	Bonetti Darlan	74/3, 74/3
Barthel Frank	46/3, 80/4, 46/3, 80/4	Bonilla Riano Adriana	18/2
Bartholomew Paul	12/6, 12/6	Bonjour Jocelyn	65/1, 65/1
Barton Geoff	41/2	Bontozoglou Vasileios	6/3, 6/3
Bartosiewicz Yann	12/5, 12/5	Bonzanini Arianna	8/6, 54/3, 8/6, 54/3
Bassani Carlos	9/1, 9/1	Booty Michael	6/2
Bateman Samuel	148/3, 148/3	Bordoloi Nimish	94/5, 94/5
Bateson Colin	44/5, 44/5	Bornette Frederic	138/6, 138/6
Battista Francesco	35/1, 71/3, 109/1, 115/2, 35/1, 71/3, 109/1, 115/2	Bortolin Stefano	73/3, 73/3
Battistella Alessandro	3/3, 3/3	Bossy Mireille	65/6, 65/6
Baudin Nicolas	29/2, 29/2	Bothe Dieter	6/5, 41/3, 60/6, 122/1, 125/5, 133/6, 138/2, 142/6, 143/5, 6/5, 41/3, 60/6, 122/1, 125/5, 133/6, 138/2, 142/6, 143/5
Baudry Cyril	90/3, 90/3	Bouchet Gilles	125/1, 130/5, 125/1, 130/5
Baumann Ingo	1/1, 1/1	Bourdon Benoit	65/2, 65/2
Bayer Ilker	75/2, 75/2	Bourgoin Mickael	21/5, 21/5
Bayly Andrew	135/4, 135/4	Bozorgnezhad Ali	138/4, 138/4
Bazan Carlos Martinez	28/1, 63/3, 28/1, 63/3	Braga De Paula Igor	98/3, 98/3
Bazile Rudy	33/2, 33/2	Bragelli Leonardo	43/4, 43/4
Beard Philippe	146/2, 146/2	Brandl De Motta Jorge Cesar	79/1, 126/4, 131/3, 79/1, 126/4, 131/3
Becker Dustin	82/2, 82/2	Brandner Juergen J.	133/3, 133/3
Behr Marek	50/2	Brandt Luca	23/1, 62/3, 79/3, 102/2, 120/6, 131/6, 138/3, 139/5, 23/1, 62/3, 79/3, 102/2, 120/6, 131/6, 138/3, 139/5
Bel Fdhila Rebei	88/6, 88/6	Brata Enggar Diwangkara	91/1, 91/1
Belfroid Stefan	119/6	Brauner Neima	105/3
Bellakhal Ghazi	42/6, 51/5, 54/5, 42/6, 51/5, 54/5	Bravo Luis	69/1, 69/1
Bellot Jean-Pierre	50/3, 50/3	Breakey David	8/2, 8/2
Belt Roel	113/4, 113/4	Bredmose Henrik	103/2, 103/2
Belut Emmanuel	12/1, 52/6, 73/4, 90/6	Breitenbach Jan	7/6, 7/6
Beneventi Davide	19/5, 19/5	Brenn Guenter	2/6, 65/4, 2/6, 65/4
Benguigui William	29/5, 29/5	Bretones Alvaro	20/5, 20/5
Bennani Lokman	90/3, 90/3	Breuer Michael	37/2, 39/4, 66/1, 37/2, 39/4, 66/1
Benoit Hadrien	16/2, 16/2	Breugem Wim-Paul	79/3, 102/2, 129/5, 139/5, 79/3, 102/2, 129/5, 139/5
Berg Niclas	29/1, 29/1	Brezgin Dmitry	107/2, 107/2
Bergeles Georgios	88/1, 88/1	Brito Ronei	9/4, 40/5
Bergeles Konstantinos	30/6, 70/4, 30/6, 70/4	Brocchini Maurizio	4/4, 137/3, 4/4, 137/3
Berger Martina	75/6, 75/6	Bromhal Grant	13/5, 13/5
Bergougnoux Laurence	110/3, 125/1, 130/5, 110/3, 125/1, 130/5	Brown Lloyd	130/2, 130/2
Bergstrom Donald J.	118/6, 134/3, 134/4, 118/6, 134/3, 134/4	Brucato Alberto	85/2, 85/2
Berlemont Alain	33/3, 33/3	Bruder Moritz	34/1, 34/1
Bernard Manuel	112/1, 112/1	Bucci Matteo	20/1, 47/3, 20/1, 47/3
Bertola Francesco	75/4, 75/4	Buchanan John	92/5, 92/5
Bertoldi Dalton	145/6, 145/6	Buchlin Jean-Marie	120/4, 120/4
Bertrand Emilie	65/2, 65/2	Bueck Andreas	104/4, 104/4
Bertrand Frederic	107/1, 107/1		
Bertrandias Aude	70/3, 70/3		
Besagni Giorgio	19/4, 81/6, 19/4, 81/6		
Bezecny Daniel	68/5, 122/1, 68/5, 122/1		
Bhagwat Swanand	108/5, 108/5		
Bhamidi Prasad	126/1, 126/1		

Buffat Marc	116/5	Chen Nana	2/1, 2/1
Buffo Antonio	85/5, 135/2, 147/3	Chen Rongqian	150/2, 150/2
Bulusu Kartik V.	28/6, 28/6	Chen Rui	1/3, 1/3
Buongiorno Jacopo	10/6, 20/1, 37/6, 47/3, 105/1, 10/6, 20/1, 37/6, 47/3, 105/1	Chen Tianming	28/3, 28/3
Burlutskii Evgenii	81/2	Chen Weixiong	77/1, 77/1
Burns Alan	123/3, 123/3	Cherdantsev Andrey	4/6, 5/3, 15/5, 28/2, 105/4, 4/6, 5/3, 15/5, 28/2, 105/4
Burns Peter	71/2, 71/2	Cherdantsev Mikhail	105/4, 105/4
Busciglio Antonio	85/3, 85/3	Chergui Jalel	60/3, 66/2, 60/3, 66/2
Buss Lizoel	108/4, 108/4	Cherrier Gael	52/6, 52/6
Butler Jason	117/3, 124/1, 117/3, 124/1	Cheung Sherman	56/1, 56/1
Buwa Vivek	5/2, 50/1, 115/3	Chevrier Solene	112/2, 112/4, 112/2, 112/4
Caballina Ophelie	70/2, 70/2	Chibbaro Sergio	69/4, 69/4
Cai Bo	144/1, 144/1	Chien Nguyen Ba	1/4, 1/4
Cai Xuan	63/4, 68/4, 63/4, 68/4	Chinak Aleksandr	141/1, 141/1
Calantoni Joseph	4/4, 4/5, 54/1, 148/3, 4/4, 4/5, 54/1, 148/3	Chinaud Maxime	24/6, 26/1, 121/4, 24/6, 26/1, 121/4
Camarri Simone	98/1, 98/1	Chirone Riccardo	114/3, 114/3
Candelier Fabien	102/1	Chiva Sergio	72/3, 72/3
Capecelatro Jesse	44/3, 44/3	Chivot Guillaume	58/3, 58/3
Caposciutti Gianluca	108/3, 108/3	Cho Hyun-Jin	50/5, 68/1, 50/5, 68/1
Capuano Francesco	60/1, 60/1	Cho Wonsam	146/6, 146/6
Capuano Marion	58/5, 58/5	Choi Chang-Hwan	46/1, 61/6, 62/1, 46/1, 61/6, 62/1
Cardona Maria	130/4, 130/4	Choi Haecheon	10/2, 10/2
Carlier Vincent	146/2, 146/2	Choi Hyoung	31/5
Carneiro Joao	10/3, 148/1, 10/3, 148/1	Choi Kwang-Il	1/4, 1/4
Carstensen Jens Michael	74/5, 74/5	Choi Sung Mok	90/5, 90/5
Cartellier Alain	19/5, 21/5, 40/3, 46/6, 19/5, 21/5, 40/3, 46/6	Choi Sungmin	68/1, 68/1
Carter Douglas	35/4, 35/4	Chong Daotong	77/1, 77/1
Carter William	9/6	Chou Shih-Hao	96/6, 96/6
Casciola Carlo Massimo	35/1, 71/3, 109/1, 115/2, 35/1, 71/3, 109/1, 115/2	Chouippe Agathe	76/6, 117/2, 76/6, 117/2
Castanet Guillaume	70/2, 70/2	Chowdhury Rajesh	106/2, 106/2
Castrillon-Escobar Sebastian	32/3, 98/5, 32/3, 98/5	Christophe Theo	12/1
Cavallini Barbosa Marcel	86/5, 145/4, 86/5, 145/4	Chun Ho Hwan	120/3, 120/3
Cavicchio Caio	18/4, 18/4	Cid Emmanuel	18/1, 18/1
Caviezel Daniel	72/1, 72/1	Cierpka Christian	48/2, 48/2
Cazin Sebastien	55/4, 55/4	Cifani Paolo	103/1, 103/1
Ceccio Steven L.	30/1, 47/6, 64/5, 66/3, 92/5, 137/4, 30/1, 47/6, 64/5, 66/3, 92/5, 137/4	Ciofalo Michele	85/2, 85/2
Cen Kefa	122/3, 122/3	Cipollina Andrea	85/2, 85/2
Cerqueira Rafael	9/4, 40/5	Clark Alicia	70/6, 70/6
Cerroni Daniele	119/5	Clausen Jonathan	127/1, 127/1
Cha Dong An	29/4, 29/4	Climent Eric	109/3, 116/6, 124/5, 131/3, 109/3, 116/6, 124/5, 131/3
Chahed Jamel	42/6, 51/5, 54/5, 42/6, 51/5, 54/5	Cloete Schalk	16/1, 126/2, 16/1, 126/2
Chaib Ina Fathia	51/5, 51/5	Cockx Arnaud	76/1
Chakraborty Nilanjan	69/2, 69/2	Cohen Asaf	95/1, 95/1
Challabotla Niranjan Reddy	102/4, 102/4	Coletti Filippo	35/4, 39/3, 129/4, 35/4, 39/3, 129/4
Chan Derek	18/6, 18/6	Colin Catherine	1/5, 19/1, 29/2, 55/4, 1/5, 19/1, 29/2, 55/4
Chandra Pankaj	81/3	Colonus Tim	11/5, 11/5
Chandra Sanjeev	18/5	Connington Kevin	61/6, 61/6
Chang Choon-Soo	105/2, 105/2	Coppola Gennaro	60/1, 60/1
Chang Hongli	34/3, 34/3	Corvaro Sara	137/3, 137/3
Channiwala Salim	47/1, 47/1	Cossali Gianpietro Elvio	38/1, 60/5, 38/1, 60/5
Chanson Hubert	14/3, 95/3	Costa Pedro	79/3, 102/2, 139/5, 79/3, 102/2, 139/5
Chapelle Pierre	50/3, 50/3	Costantini Roberta	115/2, 115/2
Charalampous Georgios	30/6, 57/4, 65/5, 30/6, 57/4, 65/5	Cotas Carla	116/3, 116/3
Charogiannis Alexandros	59/5, 77/3, 59/5, 77/3	Coufort Saudejaud Carole	48/3, 48/3
Chatelain Philippe	12/5, 12/5	Courtial Xavier	146/2, 146/2
Chatzidafni Ariadni	140/5, 140/5	Covello Vanessa	92/6, 92/6
Che Zhizhao	7/4, 22/1, 7/4, 22/1	Coyle Carolyn	37/6, 37/6
Chen Bin	1/2, 39/1, 53/2	Crandall Dustin	13/5, 13/5
Chen Boxiong	2/2, 2/2	Cui Xinguang	1/1, 1/1
Chen Chunying	1/3, 1/3	Curran Thomas	131/1, 131/1
Chen Hong	10/4, 24/5, 88/2, 10/4, 24/5, 88/2	D Avino Gaetano	124/3, 124/3
Chen Jincai	21/6, 21/6	Da Silva Marco	67/1, 67/1
Chen Li	56/1, 56/1	Dabiri Sadegh	103/3, 103/3
Chen Luming	42/5, 42/5	Dai Zhongping	149/2
Chen Ming	30/4, 110/4, 30/4, 110/4	Dairay Thibault	1/6, 1/6
		Dalmazzone Christine	39/5, 39/5
		Damir Garaev	54/4, 54/4
		Dapelo Davide	121/3, 121/3

Daraio Domenico	89/6, 89/6	Duan Yifei	70/5, 70/5
Darvan Ali	89/2, 89/2	Duignan Barry	93/4
Das Arup Kumar	15/6, 33/4, 65/3, 89/1, 103/6	Dumont Nicolas	102/5, 102/5
Das Gargi	67/6, 67/6	Dumouchel Christophe	98/2, 110/5, 98/2, 110/5
Das Prasanta Kumar	15/6, 36/6, 38/3, 61/1, 65/3, 67/6, 74/6, 89/1, 103/6, 141/5	Dunn Dennis	87/3, 87/3
		Dupays Joel	13/1, 107/6, 13/1, 107/6
		Dupif Valentin	13/1, 102/6, 107/6, 13/1, 102/6, 107/6
Das Saurish	83/4, 115/4, 83/4, 115/4	Dupuy Magali	78/3, 78/3
Dashliborun Amir Motamed	61/2, 61/2	Duret Benjamin	136/1, 136/1
Datta Saikat	74/6, 89/1, 74/6, 89/1	Dusek Jan	76/6, 76/6
Datta Santanu Prasad	74/6, 74/6	Duval Herve	70/3, 78/3, 70/3, 78/3
De Bona Jeremias	147/3	Dwivedi Prateek	48/1, 48/1
De Chaisemartin Stephane	66/5, 66/5	Dymowski Konrad	64/2
De Guido Giorgia	81/6, 81/6	Eaton John	35/2, 35/2
De Luca Luigi	60/1, 60/1	Eckel Georg	139/1, 139/1
De Malmazet Erik	68/6, 68/6	Eckert Kerstin	48/2, 48/2
De Monaco Gaetano	98/6, 98/6	Eckert Sven	32/1, 39/2, 46/5, 32/1, 39/2, 46/5
De Moraes Franklin Erick	62/4, 145/3, 62/4, 145/3	Edelvik Fredrik	120/1, 128/2, 120/1, 128/2
De Paoli Marco	22/6, 22/6	Eghtesad Mohammad	28/5, 28/5
De Shauvik	77/4, 77/4	Ehara Hideharu	15/1
De Souza Francisco Jose	6/1, 20/5, 143/4, 145/5	Ein Mozaffari Farhad	92/2
De Vries Jaap	24/2, 24/2	Einarsson Jonas	102/1
Decarre Sandrine	117/5, 117/5	Eisma Jerke	79/5, 79/5
Deen Niels G.	3/2, 80/4, 115/4, 140/3, 3/2, 80/4, 115/4, 140/3	Elbing Brian. R	72/5
Deendarlianto Deendarlianto	72/6, 89/4, 91/1, 72/6, 89/4, 91/1	Elgeti Stefanie	50/2
Dehbi Abdel	100/2, 100/2	Elghobashi Said	122/6, 122/6
Deising Daniel	41/3, 122/1, 41/3, 122/1	Elias Jebin	25/1, 25/1
Del Col Davide	73/3, 73/3	Elkins Christopher	35/2, 35/2
Delannoy Yves	50/3, 50/3	Eller Sebastian	122/1, 122/1
Delmotte Blaise	109/3, 109/3	Elperin Tov	71/5, 71/5
Delon Antoine	40/3, 40/3	Elsinga Gerrit	79/5, 79/5
Demaziere Christophe	46/4, 66/4, 46/4, 66/4	Eluk Tal	108/1
Demoulin Francois-Xavier	99/5, 110/6, 136/1, 99/5, 110/6, 136/1	Engesser Philipp	65/4, 65/4
Deng Xiaolong	41/5, 84/1	Enjalbert Romain	121/3, 121/3
Denis Heidi	59/2, 59/2	Eppinger Thomas	91/2, 91/2
Denner Fabian	12/6, 49/1, 59/5, 12/6, 49/1, 59/5	Erkan Nejdet	53/3, 53/3
Derks Peter W. J.	145/4, 145/4	Ern Patricia	10/1, 45/6, 10/1, 45/6
Derksen Jos	61/5, 61/5	Ertl Moritz	134/2, 134/2
Descamps Cedric	133/2, 133/2	Ertl Thomas	134/2, 137/2, 134/2, 137/2
Desjardins Olivier	7/3, 8/3, 44/3, 97/3, 7/3, 8/3, 44/3, 97/3	Escrig Escrig Josep	19/3, 27/6, 54/4, 82/3, 104/2
		Escudie Dany	142/1, 142/1
Detcheverry Francois	15/4, 15/4	Eskerud Smith Ivar	5/6, 5/6
Deutschmann Olaf	63/4, 68/4, 63/4, 68/4	Esmaily Moghadam Mahdi	21/3, 21/3
Devisilov Vladimir	133/5, 133/5	Essadki Mohamed	66/5, 66/5
Dewangan Kush Kumar	61/1, 61/1	Esteghamatian Amir	112/1, 112/1
Di Felice Fabio	59/1, 59/1	Estevam Valdir	18/2, 18/3
Di Marco Paolo	80/1, 80/1	Estivalezes Jean-Luc	24/1, 118/4, 131/3, 24/1, 118/4, 131/3
Di Vitantonio Giuseppe	22/2, 22/2		
Dias Frederic	137/6, 137/6	Estrada Perez Carlos E.	23/4, 80/5, 23/4, 80/5
Diaz Lamberto	150/5, 150/5	Etienne Robert	133/1, 133/1
Dimakopoulos Yiannis	127/3, 128/1, 127/3, 128/1	Evrard Fabien	49/1, 131/1, 49/1, 131/1
Dimitrakis Georgios	27/4, 59/3	Evseev Aleksei	140/2
Ding Lin	12/3, 12/3	Fairweather Michael	26/6, 26/6
Ding Wei	11/1	Falcone Manuel	143/5, 143/5
Dittmeyer Roland	23/3	Falgout Zachary	2/2, 2/2
Djeridi Henda	46/6, 57/3, 46/6, 57/3	Fan Jianren	101/3, 112/3, 118/3, 119/3, 126/6, 128/5, 135/6, 139/6, 145/1
Do Quang Minh	78/2, 86/4, 78/2, 86/4		
Dodd Michael	109/2, 109/2	Fan Liang-Liang	88/4, 127/5, 149/6, 88/4, 127/5, 149/6
Doisneau Francois	125/3, 125/3		
Dong Chuanshuai	149/3, 149/3	Fan LiWu	29/3, 29/3
Dong Feng	67/3, 81/5, 144/3, 67/3, 81/5, 144/3	Fancellu Luca	119/5
		Farnoud Ali	1/1, 1/1
Dong Jingliang	1/3, 1/3	Fath Anja	125/5, 125/5
Dong Xiaoxiao	81/5, 81/5	Fauchet Gauthier	123/2, 123/2
Donghyeon Lee	52/2, 52/2	Febres Soria Mijail	106/5, 106/5
Dooleweerd Karin	83/5, 83/5	Fechter Stefan	125/4, 125/4
Dos Santos Vinicius Vagundes	6/1	Fede Pascal	17/5, 73/4, 87/5, 90/3, 90/6, 100/5, 109/3, 112/2, 112/4, 117/1, 17/5, 73/4, 87/5, 90/3, 90/6, 100/5, 109/3, 112/2, 112/4, 117/1
Drefenstedt Sebastian	114/2		
Dru Florence	60/2, 60/2		
Du Xiaozhe	20/6, 20/6		

Feng Song	122/4, 122/4	Geurts Bernard	93/1, 103/1, 93/1, 103/1
Feng Yuqing	114/5, 114/5	Ghabache Elisabeth	28/4, 28/4
Feng Zhi Wei	31/3, 31/3	Ghadiri Mojtaba	135/4, 135/4
Feng Zhi-Gang	70/5, 70/5	Ghaemi Sina	8/2, 8/2
Ferkl Pavel	86/6, 86/6	Ghajar Afshin	108/5, 108/5
Ferrante Antonino	109/2, 109/2	Ghodke Chaitanya	13/4, 151/5, 13/4, 151/5
Ferrari Marco	8/6, 54/3, 8/6, 54/3	Ghosh Sumana	48/1, 106/4, 121/6, 147/5,
Fershtman Adam	34/2, 34/2		48/1, 106/4, 121/6, 147/5
Figueiredo Patric	137/5, 137/5	Giacomelli Francesco	129/6, 129/6
Figueroa Rosette Julieth	123/3, 123/3	Giannetti Niccolo	138/1, 138/1
Filella Audrey	10/1, 45/6, 10/1, 45/6	Gicquel Olivier	102/5, 102/5
Finotello Giulia	3/2, 3/2	Giorgi Marie-Laurence	70/3, 70/3
Flamant Gilles	16/2, 16/2	Girardin Mathieu	58/3, 58/3
Fleau Solene	52/1, 130/3, 52/1, 130/3	Goda Atsushi	34/4, 44/1, 34/4, 44/1
Fletcher David	41/2	Goll Stephan	83/2, 83/2
Fochesato Christophe	31/4, 125/6, 31/4, 125/6	Golovin Kevin	64/5, 137/4, 64/5, 137/4
Fominykh Andrew	71/5, 71/5	Gomez Bernal Sergio	143/3, 143/3
Fonellosa Caro Aleix	88/6, 88/6	Gomez-Lopez Miguel	51/1, 51/1
Fong Kee Onn	39/3, 39/3	Gong Jing	12/3, 17/2, 34/3, 56/4, 99/1,
Fontaine Jean-Raymond	73/4, 90/6, 73/4, 90/6		144/2, 146/5, 148/2, 150/1,
Fornari Walter	62/3, 131/6, 62/3, 131/6		12/3, 17/2, 34/3, 56/4, 99/1,
Forret Ann	19/5, 75/5, 19/5, 75/5		144/2, 146/5, 148/2, 150/1
Forte Giuseppe	89/6, 89/6	Gong Xiaobo	114/1
Fourati Manel	146/2, 146/2	Goniva Christoph	104/1
Fournier Richard	17/5, 17/5	Gonzalez Eric	148/1, 148/1
Fox Rodney	44/3, 75/4, 85/4, 44/3, 75/4,	Gose James W.	64/5, 92/5, 137/4, 64/5,
	85/4		92/5, 137/4
Fraggedakis Dimitrios	128/1, 128/1	Gouarin Loic	58/3, 58/3
Frances Christine	48/3, 48/3	Gourdon Mathias	51/2, 51/2
Frank Donya	4/5, 4/5	Gracia Fernando	116/3, 116/3
Frederix Edo	93/1, 93/1	Gradeck Michel	32/3, 98/5, 32/3, 98/5
Frigaard Ian	128/3, 149/4, 149/5, 128/3,	Grana-Otero Jose	51/1, 51/1
	149/4, 149/5	Grann Helge	83/5, 83/5
Frings Markus	50/2	Greco Francesco	98/6, 124/3, 98/6, 124/3
Fritsching Udo	57/5, 69/6, 106/1, 108/4,	Gregorc Jurij	37/4, 37/4
	111/6, 57/5, 69/6, 106/1,	Grosjean Nathalie	20/3, 20/3
	108/4, 111/6	Gu Jingchen	7/2, 7/2
Frohlich Jochen	3/5, 3/6, 29/6, 39/2, 95/5	Guala Michele	31/1, 31/1
Frungieri Graziano	117/6, 117/6	Gualtieri Paolo	35/1, 71/3, 109/1, 35/1,
Fuchs Laszlo	28/6, 29/1, 28/6, 29/1		71/3, 109/1
Fujii Hiroyuki	2/5, 11/3, 16/5, 16/6, 2/5,	Guazzelli Elisabeth	110/3, 117/3, 124/1, 125/1,
	11/3, 16/5, 16/6		130/5, 110/3, 117/3, 124/1,
			125/1, 130/5
Fujimoto Shuhei	77/6, 77/6		48/3, 48/3
Fujioka Satoko	142/3	Guerin Lea	74/3, 74/3
Fujishima Shun	63/5, 63/5	Guersoni Bizotto Vanessa	
Fujita Seiji	119/2, 119/2	Cristina	
Fukada Toshiaki	87/2, 87/2	Gugliermetti Luca	43/4, 43/4
Fukuda Kota	76/2, 136/2, 76/2, 136/2	Guhathakurta Jainabalkya	23/3
Fullana Jose-Maria	84/6, 84/6	Gui Nan	40/1, 83/1, 40/1, 83/1
Furlan John	32/4, 32/4	Guingo Mathieu	29/5, 29/5
Furuya Masahiro	38/2, 38/2	Guion Alexandre	10/6, 10/6
Fuster Daniel	33/1, 33/1	Guo Yajun	43/3, 43/3
Gabillet Celine	55/4, 55/4	Gupta Vinay Kumar	115/5, 115/5
Gada Vinesh H.	25/1, 108/6, 25/1, 108/6	Gurka Roi	63/6, 63/6
Galbiati Claudio	60/5, 60/5	Gurung Arati	31/6
Galizzi Cedric	142/1, 142/1	Gustavsson Kristian	125/2, 125/2
Galleni Francesco	5/4, 27/6	Gutheil Eva	1/1, 1/1
Galletti Chiara	22/2, 98/1, 108/3, 22/2,	Gutierrez Montes Candido	28/1, 28/1
	98/1, 108/3	Gvozdic Biljana	46/2, 46/2
Gamit Jigar	47/1, 47/1	Ha Hyun-Ho	29/4, 29/4
Ganesh Harish	30/1, 47/6, 66/3, 30/1, 47/6,	Haase Katharina	101/4, 101/4
	66/3	Hadj-Achour Miloud	32/3, 43/5, 98/5, 32/3, 43/5,
			98/5
Gao Xin	146/1, 146/1	Hafeli Richard	40/4, 40/4
Gao Yuhang	20/6, 20/6	Haghighi Mohammad Reza	118/6, 118/6
Gatewood Jason	70/5, 70/5	Haghighi Mohammad Reza	52/4, 52/4
Gauss Frederike	118/1, 118/1	Hagiwara Yoshimichi	61/4, 61/4
Gauthier Daniel	16/2, 16/2	Hagmeijer Rob	38/5, 106/6
Gavaises Manolis	2/4, 88/1, 2/4, 88/1	Haidn Oskar	47/4, 47/4
Ge Liang	83/1, 83/1	Haj Ali	62/6, 62/6
Gebauer Felix	43/6, 43/6	Hamamoto Shoichiro	92/1, 92/1
Gelfgat Alexander	105/3	Hamidouche Souria	13/3, 116/2, 13/3, 116/2
Gemello Luca	75/5, 75/5	Hamidouche Ziad	100/5, 100/5
Georgantaki Anna	6/3, 6/3	Hammouti Abdelkader	112/1, 112/1
Geraci Gianluca	27/2, 27/2	Hampel Uwe	11/1, 46/3, 52/4, 80/4
Gerardin Fabien	52/6, 52/6		

Han Jiacheng	99/1, 144/2, 146/5, 99/1, 144/2, 146/5	Hohl Lena	135/3, 135/3
Han Sang-Woo	50/5, 68/1, 50/5, 68/1	Hohlfeld Ivette	24/4, 24/4
Hann David	4/6, 15/5, 28/2, 93/5, 104/2	Hokari Ryohei	49/3, 49/3
Hao Chenyu	25/5, 25/5	Holland Daniel	14/4, 14/4
Harada Shusaku	119/1, 132/1, 119/1, 132/1	Holmgren Hanna	86/2, 86/2
Harasawa Nozomi	142/3	Holzinger Gerhard	22/4
Hardalupas Yannis	30/6, 57/4, 65/5, 70/4, 30/6, 57/4, 65/5, 70/4	Holzner Markus	64/1, 64/1
Hargreaves David	40/2	Hong Jiarong	31/1, 31/1
Haringa Cees	85/1, 86/1, 85/1, 86/1	Hoppe Dietrich	32/1, 32/1
Haroun Yacine	39/5, 39/5	Hoppe Felix	37/2, 37/2
Hasan Abbas	27/4, 40/6, 59/3	Hori Yohei	52/5, 52/5
Hasegawa Koji	34/4, 64/6, 34/4, 64/6	Hori Yuusaku	16/5, 16/5
Hassan Yassin A.	23/4, 80/5, 23/4, 80/5	Hormozi Sarah	149/5, 149/5
Hata Kazuaki	32/6, 32/6	Horwitz Jeremy	27/2, 27/2
Hattori Yasuo	123/1, 123/1	Hosokawa Shigeo	26/2, 26/3, 52/5, 55/1, 26/2, 26/3, 52/5, 55/1
Haugen Nils Erland L.	79/2, 122/2, 129/2, 79/2, 122/2, 129/2	Hospers Jacco	91/4, 97/4, 91/4, 97/4
Hayasaka Keisuke	47/5, 47/5	Hotz Christian	139/1, 139/1
Hayashi Kosuke	26/2, 26/3, 52/5, 55/1, 60/6, 120/2, 26/2, 26/3, 52/5, 55/1, 60/6, 120/2	Howard Amanda	132/2, 132/2
Hazuku Tatsuya	35/6, 54/2, 35/6, 54/2	Hribersek Matjaz	149/1
He Denghui	89/5, 89/5	Hsiau Shu-San	96/6, 96/6
He Guo-Wei	21/6, 21/6	Hu Jun	19/2, 19/2
He Limin	149/3, 149/3	Hu Xiangyu	49/2, 49/2
He Xukun	127/5, 149/6, 127/5, 149/6	Hu Xiaofei	75/4, 75/4
Hecht Nicolas	99/5, 99/5	Hua Meng	29/3, 29/3
Hedensted Lau	140/6, 140/6	Huang Jian	59/4, 59/4
Heggs Peter	135/4, 135/4	Huang Meng	39/1
Hei Ng Chin	30/5, 30/5	Huang Yi	124/6, 124/6
Heiniger Bruno	42/4	Huber Gregory	1/5, 1/5
Heinz Johannes	6/4, 6/4	Hur Nahmkeon	105/2, 105/2
Helgans Brian	94/1	Hussainov Medhat	69/3, 69/3
Helie Jerome	90/1, 110/6, 90/1, 110/6	Iavarone Salvatore	108/3, 108/3
Hemati Mehrdji	100/5, 100/5	Ibarra Roberto	27/3, 99/3, 27/3, 99/3
Hempert Fabian	104/5, 104/5	Iben Uwe	104/5, 104/5
Hendrix Maurice	5/6, 5/6	Ibrahim Abubakr	58/6, 58/6
Henkes Ruud	26/5, 62/5, 83/6, 129/5, 26/5, 62/5, 83/6, 129/5	Icardi Matteo	104/3, 104/3
Henry De Frahan Marc	66/3, 66/3	Ichihara Mie	22/5, 22/5
Hernandez Cely Marlon	143/3, 143/3	Idoughi Schehinez	18/1, 18/1
Hernandez Freddy	44/6, 44/6	Ihara Tomonori	82/1, 82/1
Hernandez Rodriguez Iara	81/4, 81/4	Ike Masatoshi	95/4, 95/4
Hernandez Rodriguez Oscar	5/5, 18/2, 18/3, 81/4, 86/5, 112/5, 143/3, 145/4	Ikeda Ryosuke	59/4, 59/4
Mauricio	61/2, 61/2	Ikegai Tomoki	119/1, 119/1
Herrmann Heber Robert	94/3, 94/3	Imai Hiroaki	34/4, 34/4
Herrmann Marcus	34/1, 139/2, 34/1, 139/2	Indarto Indarto	72/6, 91/1, 72/6, 91/1
Heublein Norbert	4/6, 15/5, 19/3, 27/6, 28/2, 40/2, 40/6, 49/4, 54/4, 58/6, 59/3, 82/3, 84/4, 93/5	Ingelsten Simon	120/1, 120/1
Hewakandamby Buddhikha	16/4, 16/4	Ingen Housz Eduard	83/6, 83/6
Hewitt Geoffrey	85/4, 85/4	Innings Fredrik	3/2, 3/2
Heylmun Jeffrey	72/6, 72/6	Innocenti Alessio	69/4, 69/4
Hidaka Sumitomo	119/1, 119/1	Inoue Yuzo	95/4, 95/4
Higashida Kyohei	41/3, 41/3	Inthavong Kiao	1/3, 1/3
Hill Simon	150/4, 150/4	Inzoli Fabio	19/4, 81/6, 19/4, 81/6
Hincapie Diego	2/6, 2/6	Isaenkov Sergey	105/4, 105/4
Hinterbichler Hannes	32/6, 32/6	Ishii Eiji	15/1
Hirose Yusuke	42/2	Ishima Tsuneaki	63/5, 63/5
Hirsch Tobias	125/4, 125/4	Ishizawa Kanami	132/1, 132/1
Hitz Timon	114/2	Islam A. S. M. Atiquil	134/3, 134/4, 134/3, 134/4
Hlawitschka Mark W.	85/1, 85/1	Islam Shafiu	140/3, 140/3
Hoang Duong	75/3, 75/3	Ismail Renad	7/4, 7/4
Hoehne Thomas	86/3, 91/4, 97/4, 86/3, 91/4, 97/4	Iso Yoshiyuki	59/4, 59/4
Hoeijmakers Harry	74/4, 74/4	Isono Toshihisa	16/6, 16/6
Hoffmann Alex Christian	42/2	Issa Raad	5/4, 27/6
Hoffmann Alexander	104/5, 104/5	Ito Daisuke	23/6, 134/6, 23/6, 134/6
Hoffmann Malte	68/5, 96/1, 114/4, 68/5, 96/1, 114/4	Ito Takahiro	25/2, 84/2, 25/2, 84/2
Hoffmann Marko	41/3, 41/4, 41/3, 41/4	Ito Yui	57/1, 57/1
Hoffmann Rainer	92/5, 92/5	Iwata Keisuke	80/6, 80/6
Hogan Kevin		Iwata Shuichi	134/1, 134/1
		Jackson Matthew	113/5, 113/5
		Jackson Thomas L.	71/6, 71/6
		Jafari Saeed	8/5
		Jain Ekta	5/2, 5/2
		Jain Shakendra	147/5, 147/5
		Jakirlic Suad	17/1, 17/1
		Jakobsen Hugo A.	143/2, 143/2
		Jama Mohamed Ali	135/2, 135/2
		Jamison Kristin	24/2, 24/2

Jang Dong Soo	52/3, 52/3	Kartushinsky Alexander	12/4, 69/3, 74/4, 12/4, 69/3, 74/4
Janiga Gábor	107/5, 107/5	Karuna Nardrapee	130/4, 130/4
Jareteg Klas	46/4, 66/4, 82/6, 46/4, 66/4, 82/6	Kasai Yuma	144/5, 144/5
Jasak Hrvoje	103/2, 103/2	Kasbaoui Mohamed Housseem	8/3, 8/3
Jasberg Ari	124/2, 124/2	Kashinsky Oleg	141/1, 141/1
Jay Stephane	66/5, 66/5	Kassar Bruno	10/3, 10/3
Jayanti Sreenivas	45/2, 45/2	Kassinov Stavros	107/4, 107/4
Jensen Anna Lyhne	102/3, 102/3	Kastens Sven	60/6, 60/6
Jeoh Tina	130/4, 130/4	Kataoka Yutaka	16/5, 16/5
Jeon Byoungjin	31/5	Kathrotia Trupti	139/1, 139/1
Jeon Jaewoo	128/3, 128/3	Kato Kenji	25/2, 32/5, 25/2, 32/5
Jeon Joon Goo	33/5, 33/5	Katz Joseph	23/2, 111/4, 23/2, 111/4
Jeong Eunju	92/3, 92/3	Kavagnah John	41/2
Jeong Jaejun	45/3, 45/3	Kawahara Akimaro	91/5, 91/5
Jeong Uiju	25/6, 33/5, 25/6, 33/5	Kawaji Masahiro	44/6, 44/6
Jeun Gyoodong	25/6, 25/6	Kawashima Hisanobu	63/5, 63/5
Jiang Minyee	148/6, 148/6	Kayakol Nuray	2/3, 2/3
Jiang Chen	20/2, 20/2	Kazuki Mori	126/3, 126/3
Jiang Mingjer	97/2, 97/2	Ke Renjie	32/4, 32/4
Jiang Youhua	61/6, 61/6	Keedy Ryan	113/3, 113/3
Jin Chunyu	27/1, 27/1	Kekesi Timea	103/4, 103/4
Jin Guodong	21/6, 21/6	Kelly Ger	136/5, 136/5
Jin Hang	17/2, 26/4, 34/3, 150/1, 17/2, 26/4, 34/3, 150/1	Kelz Marcus	24/4, 24/4
Jin Seonyong	68/1, 68/1	Kempe Tobias	3/5, 3/5
Jin Tai	119/3, 128/5, 119/3, 128/5	Kenjeres Sasa	86/1, 86/1
Jinbo Yoshinori	11/3, 11/3	Kerber Christiane	83/2, 83/2
Jindal Arpit	115/3	Kerst Kristin	107/5, 107/5
Johansen Stein Tore	45/1, 73/1, 101/6, 45/1, 73/1, 101/6	Kerstein Alan	69/1, 69/1
Johnsen Eric	66/3, 66/3	Ketterl Sebastian	99/4, 99/4
Johnsen Sverre Gullikstad	73/1, 73/1	Khalij Mohammed	116/2, 150/3, 150/4, 116/2, 150/3, 150/4
Jong Conrad	85/1, 85/1	Khalilitehrani Mohammad	67/5
Jongsma Alfred	3/2, 3/2	Khan Shadman	106/4, 106/4
Josserand Christophe	7/5, 28/4, 84/6, 7/5, 28/4, 84/6	Kharlamov Sergey	51/3, 51/3
Jung Jaehoon	141/6, 141/6	Kharlashkin Andrey	87/1, 87/1
Jung Jinyoung	24/3, 24/3	Khateeb Noor Ul Huda	114/1
Jung Wonjong	90/5, 90/5	Khodakov Konstantin	20/4, 20/4
Juric Damir	60/3, 66/2, 60/3, 66/2	Khoo Boo Cheong	10/4, 24/5, 31/3, 88/2, 10/4, 24/5, 31/3, 88/2
Kadambi Jaikrishnan	32/4, 32/4	Khosronejad Ali	90/4, 90/4
Kaehler Christian	101/4, 101/4	Khot Ambarish	151/5, 151/5
Kaer Soren Knudsen	108/2, 138/5, 146/1, 108/2, 138/5, 146/1	Kibar Ali	46/1, 46/1
Kahouadi Lyes	48/4, 60/3, 66/2, 121/4, 48/4, 60/3, 66/2, 121/4	Kida Hitoshi	69/5, 69/5
Kajishima Takeo	7/2, 61/3, 87/2, 7/2, 61/3, 87/2	Kidanemariam Aman	44/2, 62/2, 44/2, 62/2
Kalaga Dinesh	44/6, 44/6	Kidess Anton	83/6, 83/6
Kalashnikov Alexander	59/3	Kikuchi Masataka	38/4, 38/4
Kalikmanov Vitaly	38/5, 38/5	Kikura Hiroshige	82/1, 82/1
Kalilainen Jarmo	100/2, 100/2	Kim Dae-Hwan	146/6, 146/6
Kalliadasis Serafim	59/5, 59/5	Kim Dongwoo	52/3, 52/3
Kalman Haim	71/4, 71/4	Kim Ho-Young	89/3, 90/5, 121/1, 89/3, 90/5, 121/1
Kameda Masaharu	22/5, 47/5, 69/5, 22/5, 47/5, 69/5	Kim Jungho	52/2, 141/6, 52/2, 141/6
Kamlade Jens	24/4, 24/4	Kim Jungwoo	10/2, 10/2
Kamran Alba	128/3, 149/4, 128/3, 149/4	Kim Kiyong	10/2, 10/2
Kamranian Marnani Abbas	104/4, 104/4	Kim Min Soo	29/4, 29/4
Kanagawa Tetsuya	44/1, 95/4	Kim Minjae	30/2, 30/2
Kanani Homayoon	14/2, 14/2	Kim Minki	19/6, 19/6
Kaneko Akiko	44/1, 95/4, 44/1, 95/4	Kim Sangkyu (calvin)	103/3, 103/3
Kang Nam-Won	105/2, 105/2	Kim Seunggho	89/3, 89/3
Kang Youn-Bae	50/5, 50/5	Kim Sung Jin	141/6, 141/6
Kanizawa Fabio Toshio	41/1	Kim Sung Joong	25/6, 33/5, 25/6, 33/5
Kannan Ananda Subramani	74/5, 74/5	Kim Sung-Eun	142/4, 142/4
Kanno Nozomu	139/3, 139/3	Kim Wonjung	90/5, 90/5
Kar Abhimanyu	36/6, 36/6	Kim Yongchan	52/3, 52/3
Karapetsas George	6/3, 6/3	Kimura Ryota	61/4, 61/4
Karathanassis Ioannis	2/4, 2/4	Kireev Victor	74/2, 93/3, 105/6, 74/2, 93/3, 105/6
Karawajczyk Andrzej	128/2, 128/2	Kiselev Alexei	76/6, 76/6
Karch Grzegorz	134/2, 137/2, 134/2, 137/2	Kitahara Tatsuya	45/4, 45/4
Karimi Mohsen	86/6, 128/4	Kitano Tomoaki	101/1, 101/1
Karnbach Franziska	48/2, 48/2	Kittel Hannah	8/1, 8/1
		Kjeldby Tor Kindsbekken	145/4, 145/4
		Klaseboer Evert	18/6, 18/6
		Kleijn Chris	85/1, 85/1

Klein Harald	41/3, 41/3	Lain Santiago	109/4, 151/4, 109/4, 151/4
Klein Markus	99/4, 99/4	Lakehal Djamel	16/4, 42/4, 72/1, 105/1
Kleinbart Simon	44/6, 44/6	Lakshmanan Susithra	14/4, 14/4
Kljenak Ivo	36/1, 36/1	Lalanne Benjamin	36/4, 54/6, 36/4, 54/6
Knaup Manuel	83/2, 83/2	Lamarque Nicolas	90/1, 90/1
Kneer Reinhold	137/5, 137/5	Lamorgese Andrea	22/2, 75/1, 22/2, 75/1
Knikker Ronnie	9/5, 9/5	Lampa Aljoscha	111/6, 111/6
Knoop Claas	57/5, 57/5	Lance Michel	20/3, 39/5, 107/1, 112/1
Knorps Maria	78/6, 78/6	Lancien Thea	102/5, 102/5
Kobayashi Kazumichi	2/5, 11/3, 16/5, 16/6, 2/5, 11/3, 16/5, 16/6	Laneryd Tor	88/6, 88/6
Koch Donald	8/3, 8/3	Langsholt Morten	59/2, 59/2
Koguna Aminu	50/6, 50/6	Lanotte Alessandra Sabina	53/4, 53/4
Kohno Masamichi	72/6, 89/4, 72/6, 89/4	Lao Liyun	36/2, 50/6, 36/2, 50/6
Koizumi Yasuo	81/1, 81/1	Larat Adam	60/2, 66/5, 87/4, 60/2, 66/5, 87/4
Kokh Samuel	60/2, 60/2	Lashgari Iman	62/3, 139/5, 62/3, 139/5
Komori Satoru	101/1, 101/1	Lau Timothy	58/1, 58/1
Kon Misaki	2/5, 2/5	Lau Yuk Man	32/2, 61/2, 80/4, 32/2, 61/2, 80/4
Koncar Bostjan	36/1, 36/1	Lauk Peep	69/3, 69/3
Kondo Katsuya	7/2, 7/2	Laupsien David	76/1
Koneru Rahul	121/2, 121/2	Laurent Frederique	13/1, 66/5, 102/6, 107/6, 13/1, 66/5, 102/6, 107/6
Kono Kengo	34/4, 34/4	Lavalle Gianluca	59/6, 59/6
Koponen Antti	124/2, 124/2	Lavieville Jerome	29/5, 37/1, 90/3, 29/5, 37/1, 90/3
Korytnyi Efim	108/1	Lavrenteva Olga	34/5
Kosek Juraj	86/6, 86/6	Law Wen Zhe	91/5, 91/5
Koshi Mitsuo	139/3, 139/3	Lay Ebrahim Nemati	141/4, 142/2, 141/4, 142/2
Kosinski Pawel	12/4, 69/3, 74/4, 12/4, 69/3, 74/4	Laysac Thibaut	56/3, 56/3
Kotsubo Yusuke	86/4, 86/4	Le Bot Cedric	133/2, 133/2
Koukouvinis Foivos C.	2/4, 2/4	Le Clercq Patrick	139/1, 139/1
Koullapis Pantelis	107/4, 107/4	Le Men Claude	76/1
Kovalchuk Nina	78/4	Le Merrer Marie	15/4, 15/4
Kovaleva Liana	93/3, 93/3	Le Sauze Nathalie	18/1, 18/1
Krasovitev Boris	71/5, 71/5	Le Touze Clement	13/1, 13/1
Kraume Matthias	43/6, 135/3, 43/6, 135/3	Lebon Michel	19/1, 19/1
Kreider Wayne	11/5, 11/5	Lebrun Pierre	50/3, 50/3
Kreiss Gunilla	86/2, 86/2	Lee Hyeseung	141/6, 141/6
Krepper Eckhard	11/1, 51/4, 118/1, 123/5	Lee Inwon	120/3, 120/3
Kreutzer Michiel	85/1, 85/1	Lee Joo Seong	52/3, 52/3
Kriebitzsch Sebastian	51/4, 51/4	Lee Joohwa Sarah	5/1, 5/1
Krishnan Ghoku	96/5, 96/5	Lee Juhun	24/3
Kroll-Rabotin Jean-Sebastien	48/5, 48/5	Lee Jun Ho	19/6, 19/6
Kromer Heiko	88/6, 88/6	Lee Keng Lin	73/5, 73/5
Kromer Johannes	138/2, 138/2	Lee Kurnchul	30/2, 30/2
Kronbichler Martin	86/2, 86/2	Lee Sang Hyuk	146/6, 146/6
Krueger Jonas	122/2, 122/2	Lee Sang-Beom	92/3, 92/3
Krull Benjamin	3/6, 3/6	Lee Taehyun	146/6, 146/6
Kuck Ulf	101/4, 101/4	Lefebvre Lepot Aline	58/2, 58/3, 58/2, 58/3
Kuczaj Arkadiusz	93/1, 93/1	Legendre Dominique	37/1, 55/2, 106/5, 127/4, 37/1, 55/2, 106/5, 127/4
Kuerten Hans	57/6, 103/1	Leisner Thomas	76/6, 76/6
Kuesters Christof	133/3, 133/3	Lemoine Fabrice	70/2, 70/2
Kuhlmann Hendrik	131/2, 139/4, 131/2, 139/4	Lenz Stefan	83/2, 83/2
Kuhnel Wolfram	123/6, 123/6	Lepilliez Mathieu	39/6, 39/6
Kuhni Manuel	142/1, 142/1	Lepoutere Guillaume	17/5, 17/5
Kuibin Pavel	113/2, 120/5, 113/2, 120/5	Lester Edward	84/4, 84/4
Kuipers J. A. M (hans)	3/2, 7/1, 48/6, 77/4, 80/4, 115/4, 118/5, 140/3, 3/2, 7/1, 48/6, 77/4, 80/4, 115/4, 118/5, 140/3	Letan Ruth	79/6, 79/6
Kumagai Ichiro	55/6, 77/6, 55/6, 77/6	Lettieri Paola	129/1, 129/1
Kumar Kiran	81/2	Levy Avi	108/1
Kumar Manoj	65/3, 65/3	Li Cheng	111/4, 111/4
Kunkelmann Christian	133/6, 133/6	Li Dong	101/3, 145/1, 101/3, 145/1
Kuntoro Hadiyan	89/4, 89/4	Li Dong	1/2, 1/2
Kunugi Tomoaki	56/2, 56/2	Li Dongyue	147/3
Kurihara Kazuma	49/3, 49/3	Li Hang	43/3, 43/3
Kurose Ryoichi	101/1, 101/1	Li Hui-Xiong	11/2, 25/5, 113/1, 11/2, 25/5, 113/1
Kurose Yudai	132/1, 132/1	Li Jason	2/4, 2/4
Kusuno Hiroaki	36/5	Li Jing	44/4, 100/1, 124/6, 134/5, 44/4, 100/1, 124/6, 134/5
Kvss Srikanth	115/5, 115/5	Li Kai	11/2, 17/2, 26/4, 34/3, 150/1, 11/2, 17/2, 26/4, 34/3, 150/1
Kwak Minhee	105/2, 105/2	Li Ke	114/6, 114/6
Kwon Oh Kyung	29/4, 29/4		
Labois Mathieu	58/4, 58/4		
Lagarde Julien	102/6, 102/6		
Laila Ladhani	133/1, 133/1		

Li Liangxing	113/1, 113/1	Lucquiaud Mathieu	59/6, 59/6
Li Pengfei	144/3, 144/3	Luke Andrea	37/5, 57/2, 63/1, 146/4, 37/5, 57/2, 63/1, 146/4
Li Qi	77/2, 77/2		
Li Weiyu	53/2, 53/2	Lund Jacob	74/5, 74/5
Li Xia	42/5, 104/2	Lundell Fredrik	86/4, 102/1, 116/4
Li Xiaoyi	111/5, 111/5	Luo Kun	101/3, 112/3, 118/3, 119/3, 126/6, 128/5, 135/6, 139/6, 145/1
Li Yong	2/1, 2/1		
Li Zheng	144/4, 144/4	Luo Meng	47/4, 47/4
Li Zhenlin	40/1, 83/1, 40/1, 83/1	Luo Shuaishuai	88/4, 88/4
Liang Wei	144/4, 144/4	Luo Yujuan	119/3, 119/3
Liberzon Alex	64/1, 64/1	Luo Zhengyuan	78/5, 78/5
Lichtenegger Thomas	22/4, 67/4	Luthi Beat	64/1, 64/1
Lif Johan	37/3, 37/3	Lv Xiaofang	12/3, 12/3
Lim Eldin Wee Chuan	68/3, 68/3	Lykholt Ustrup Flemming	102/3, 102/3
Lin Cheng Chuan	76/3, 76/3	Lyu Haicai	33/6, 43/3, 33/6, 43/3
Lin Jianzhong	96/2, 147/2, 147/4	Ma Mengyu	17/2, 26/4, 150/1, 17/2, 26/4, 150/1
Lin Ping	13/2		
Lindstrom Stefan	109/6, 109/6	Ma Ming	51/6, 51/6
Line Alain	48/3, 76/1, 113/4, 117/5	Ma Qianli	56/4, 56/4
Line Hakon	65/4, 65/4	Ma Tian	95/5
Ling Yue	33/1, 84/6, 33/1, 84/6	Mackenzie Jordan	116/4, 116/4
Linne Mark	2/2, 69/1, 2/2, 69/1	Madadi Kandjani Ehsan	85/4, 85/4
Lips Stephane	56/3, 56/3	Maeda Kazuki	11/5, 11/5
Liu Fangyuan	17/2, 34/3, 144/2, 17/2, 34/3, 144/2	Maffettone Pier Luca	98/6, 124/3, 98/6, 124/3
		Maftai Radu Eduard	65/6, 65/6
Liu Hongbo	28/3, 28/3	Mahmud Tariq	135/4, 135/4
Liu Jialun	113/1, 113/1	Maiti Biswajit	38/3, 38/3
Liu Li	127/2, 127/2	Maiti Ritwik	67/6, 67/6
Liu Longbin	123/4, 123/4	Maitri Rohit	118/5, 118/5
Liu Muyuan	142/6, 142/6	Majid Akmal Irfan	89/4, 91/1, 89/4, 91/1
Liu Nannan	26/4, 99/1, 144/2, 146/5, 26/4, 99/1, 144/2, 146/5	Makiharju Simo	92/5, 92/5
		Malavasi Ileana	65/2, 65/2
Liu Qiao	17/4, 17/4	Malavasi Stefano	14/5, 14/5
Liu Shang-Yu	96/6, 96/6	Malhotra Sneha	121/6, 121/6
Liu Song	151/2, 151/2	Maliska Clovis	9/4, 40/5
Liu Wen	127/2, 127/2	Malli Phillip	151/6, 151/6
Liu Xianfei	144/1, 144/1	Maltzev Leonid	140/2
Liu Xiaofei	126/6, 145/1, 126/6, 145/1	Manchon Xavier	107/1, 107/1
Liu Yang	12/3, 119/4	Mancinelli Alessandro	137/3, 137/3
Liu Zhaohui	44/4, 100/1, 122/4, 124/6, 134/5, 44/4, 100/1, 122/4, 124/6, 134/5	Mangiavacchi Norberto	99/6, 99/6
		Mani Ali	21/3, 27/2, 129/4, 21/3, 27/2, 129/4
Lizarraga-Garcia Enrique	105/1, 105/1		
Llor Antoine	31/4, 31/4	Manica Rogerio	18/6, 18/6
Lo Simon	25/1, 91/2, 122/5, 25/1, 91/2, 122/5	Manservigi Sandro	119/5
		Mansoori Zohreh	62/6, 62/6
Lobanov Pavel	141/1, 141/1	Mansour Mohamed H.	91/5, 91/5
Lohse Detlef	46/2, 92/4, 115/1, 131/4, 46/2, 92/4, 115/1, 131/4	Marcelino Neto Moises	67/1, 67/1
		Marchetti Benjamin	125/1, 125/1
Loiseau Jean-Christophe	120/6, 138/3, 120/6, 138/3	Marchioli Cristian	23/1, 69/4, 71/1, 94/6, 23/1, 69/4, 71/1, 94/6
Loisel Vincent	124/5, 124/5		
Loisy Aurore	87/6, 94/2, 87/6, 94/2	Marchisio Daniele	75/5, 86/6, 128/4, 147/3
Loizou Katerina	84/4, 84/4	Marefatallah Maedeh	49/6, 49/6
Lombardi Davide	98/6, 98/6	Marengo Marco	65/2, 65/2
Lomboy Gilson	77/5, 77/5	Marie Jean-Louis	20/3, 20/3
Lonardi Federico	146/4, 146/4	Marie Nathalie	107/1, 107/1
Longhitano Marco	97/5	Mark Andreas	120/1, 128/2, 120/1, 128/2
Lopez Diego	130/5, 130/5	Markides Christos N.	5/3, 23/5, 27/3, 59/5, 77/3, 99/3, 5/3, 23/5, 27/3, 59/5, 77/3, 99/3
Lorenzoni Carlo	137/3, 137/3		
Lorieul Gael	12/5, 12/5	Markovich Dmitriy	11/6, 51/3, 80/3, 105/4, 11/6, 51/3, 80/3, 105/4
Losi Gianluca	9/2, 9/2		
Loth Eric	43/2, 75/2, 96/5, 43/2, 75/2, 96/5	Marongiu Jean-Christophe	14/6, 14/6
		Marquis Andrew	12/6, 12/6
Lou Jing	30/4, 110/4, 30/4, 110/4	Marra Francesco Saverio	91/3, 132/3, 91/3, 132/3
Lovas Terese	122/2, 122/2	Marschall Holger	6/5, 41/3, 60/6, 63/4, 122/1, 133/6, 143/5, 6/5, 41/3, 60/6, 63/4, 122/1, 133/6, 143/5
Lovecchio Salvatore	23/1, 23/1		
Lowder Sean	37/6, 37/6	Martensson Gustaf	128/2, 128/2
Lowe Kirk	92/5, 92/5	Martin Charles-Etienne	33/2, 33/2
Loyola Francisco	43/1, 43/1	Martindejuan Luis	135/4, 135/4
Lu Jiakai	51/6, 51/6	Martinez Cuenca Raul	72/3, 72/3
Lu Jiangtao	7/1, 7/1	Martinez Manuel	109/5, 109/5
Lu Wenqiang	117/4, 117/4	Martins Diego	143/4, 145/5, 143/4, 145/5
Lu Yu	106/3, 106/3		
Lucas Dirk	19/4, 72/2, 95/5, 118/1, 123/5		



Maru Wes	14/4, 14/4	Mikielewicz Jaroslaw	95/6, 95/6
Marzocchella Antonio	114/3, 129/1, 114/3, 129/1	Milazzo Adriano	107/2, 129/6, 107/2, 129/6
Masbernat Olivier	18/1, 36/4, 54/6, 68/6, 117/5, 124/5, 126/4, 132/5, 18/1, 36/4, 54/6, 68/6, 117/5, 124/5, 126/4, 132/5 141/4, 142/2, 141/4, 142/2	Mimouni Stephane	29/5, 52/1, 130/3, 29/5, 52/1, 130/3 65/6, 65/6 137/3, 137/3 74/2, 74/2 134/6, 134/6 92/2 53/5, 53/5 79/2, 79/2 50/4, 119/4 61/3, 61/3 120/2, 120/2 76/2, 136/2, 76/2, 136/2 45/1, 45/1 46/3, 46/3 143/6, 143/6 72/5 4/1, 4/1 27/4, 40/6 83/4, 83/4 4/1, 4/1 83/5, 108/2, 83/5, 108/2 91/3, 91/3 85/3, 85/3 74/3, 74/3 18/2, 18/3
Mashayekhpour Maryam	87/5, 100/5, 87/5, 100/5	Minier Jean-Pierre	
Masi Enrica	13/1, 60/2, 66/5, 79/1, 87/4, 102/6, 107/6, 129/4, 13/1, 60/2, 66/5, 79/1, 87/4, 102/6, 107/6, 129/4	Miozzi Massimo	
Massot Marc	26/2, 26/3, 26/2, 26/3 7/4, 22/1, 27/3, 48/4, 60/3, 66/2, 121/4, 7/4, 22/1, 27/3, 48/4, 60/3, 66/2, 121/4	Miroshnichenko Vladislav	
Masukura Yuya	40/3, 46/6, 40/3, 46/6 46/2, 92/4, 131/4, 46/2, 92/4, 131/4	Mishima Kaichiro	
Matar Omar K.	77/3, 77/3 123/2, 123/2 53/1, 53/1 49/3, 49/3 49/3, 49/3 50/4 114/1 56/6, 100/6, 56/6, 100/6 146/2, 146/2 22/2, 75/1, 22/2, 75/1 57/3, 57/3 132/2, 132/2 11/5, 11/5 129/1 107/2, 129/6, 107/2, 129/6 53/4, 53/4 44/2, 44/2 15/5, 15/5 130/4, 130/4 41/2 20/1, 37/6, 47/3, 20/1, 37/6, 47/3	Mishra Prakash	
Matas Jean-Philippe		Mito Yoichi	
Mathai Varghese		Mitra Dhrubaditya	
Mathie Richard		Miwa Shuichiro	
Mathieu Benoit		Miyauchi Suguru	
Matsuda Keigo		Miyayoshi Tatsuya	
Matsumoto Junichi		Mizuno Yusuke	
Matsumoto Sohei		Mo Sjur	
Matsumoto Tatsuki		Moeller Felix	
Matsumoto Yoichiro		Moghaddam Amin Mahmoudi	
Matsuo Akiko		Mohagheghian Shahrouz	
Maubert Thomas		Mohammad Hasan Kayhani	
Mauri Roberto		Mohammed Shara	
Maurice Guillaume		Mohan Srinivasa	
Maxey Martin		Mohsen Nazari	
Maxwell Adam		Momeni Maryam	
Mazzei Luca		Montagnaro Fabio	
Mazzelli Federico		Montante Giuseppina	
Mazzitelli Irene M.		Monte Verde William	
Mazzuoli Marco		Monteiro De Carvalho Carlos Henrique	
Mccarthy Ian		Montorfano Andrea	110/6, 110/6
Mccarthy Michael		Moon Hojun	41/6, 41/6
Mcclore Dale		Moon Myoung-Woon	89/3, 89/3
Mckrell Thomas		Moon Seoksu	21/1, 21/1
Mechitoua Namane		Moosa Ashrafi	138/4, 138/4
Medeiros De Souza Luis		Moradi Nour Zeinab	78/2, 78/2
Medhi Bhaskar		Morales Rigoberto	9/1, 67/1, 145/6, 9/1, 67/1, 145/6
Mees Loic		Morente Antoine	37/1, 37/1
Meese Ernst A.		Morgan Rhys	23/5, 23/5
Mehlig Bernhard		Morganti Nicolo	80/1, 80/1
Mehrabadi Mohammad		Mori Akifumi	91/5, 91/5
Mehrzad Shams		Mori Kengo	150/6, 150/6
Mehta Yash		Mori Michitsugu	50/4, 119/4
Meiburg Eckart		Mori Shoji	126/5, 126/5
Meier Henry França		Mori Yasushige	57/1, 80/2, 57/1, 80/2
Meierhofer Florian		Morikawa Shotaro	11/3, 11/3
Meignen Renaud		Morvan Herve	81/2
Meister Christian		Mouret Quentin	142/1, 142/1
Menard Thibaut		Mouza Aikaterini	78/1, 140/5, 78/1, 140/5
Mendes Penteado Marcos Roberto		Movaghar Amirreza	69/1, 69/1
Meneghini Cristiano		Mudde Robert	91/4, 91/4
Menghini Filippo		Mueller Bjoern	37/5, 37/5
Meredith Karl V. M.		Mukherjee Siddhartha	86/1, 86/1
Merigoux Nicolas		Mun Hanseok	52/3, 52/3
Merlet Benoit		Municchi Federico	104/1
Messa Gianandrea Vittorio		Muniz Marcelo	63/2, 63/2
Messilem Abdelkader		Munoz Cobo Gonzalez Jose Luis	72/3, 72/3
Messina Roberta		Munz Claus-Dieter	104/5, 125/4, 104/5, 125/4
Meyer Jan		Muradoglu Metin	15/3, 15/3
Micale Giorgio		Murai Yuichi	55/6, 64/2, 64/3, 88/3
Michaelides Efsthios		Murakawa Hideki	104/6, 104/6
Michaie Sandra		Muramatsu Hiroya	30/3
Michalek Wiktor		Murata Shigeru	14/1, 14/1
Mihajlovic Milan		Murmanskii Ilia	107/2, 107/2
Mikielewicz Dariusz		Murphy David	23/2, 23/2
		Murphy Eric	77/5, 77/5
		Murrenhoff Hubertus	97/5
		Musin Ayrat	74/2, 93/3, 74/2, 93/3
		Mutschke Gerd	48/2, 48/2
		Mychkovsky Alexander	92/5, 92/5
		Myosen Kenta	57/1, 57/1
		Myrillas Konstantinos	120/4, 120/4
		Nagata Takayuki	76/2, 136/2, 76/2, 136/2
		Nagayama Takahiro	2/5, 2/5
		Nakagawa Atsushi	32/5, 32/5

Nakagawa Hajime	42/3, 42/3	Onishi Ryo	53/1, 101/5, 53/1, 101/5
Nakagawa Kazuyuki	84/2, 84/2	Onorato Miguel	84/3, 84/3
Nakamura Shiko	59/4, 59/4	Onuki Hajime	42/1, 42/1
Nakanishi Hiroaki	10/5, 10/5	Ooms Gijbert	83/6, 83/6
Narayanan Chidambaram	16/4, 42/4, 58/4, 72/1	Ortiz Vidal Luis Enrique	86/5, 86/5
Narayanan Ranga	98/4, 98/4	Oshitani Jun	119/1, 119/1
Naso Aurore	1/6, 87/6, 94/2, 1/6, 87/6, 94/2	Ota Kazuyuki	42/3, 42/3
Nathan Graham	58/1, 58/1	Otomo Ryoko	25/2, 126/3, 25/2, 126/3
Nayak Aameya Kumar	133/4, 133/4	Ouchene Rafik	116/2, 150/3, 116/2, 150/3
Neal Christopher R.	71/6, 71/6	Ouchi Koki	119/2, 119/2
Neau Herve	90/3, 90/3	Oud Guido	148/5, 148/5
Nelson Brent	82/4, 82/4	Ouellet Frederick	121/2, 121/2
Nemes Andras	39/3, 39/3	Overmars Edwin	79/5, 79/5
Nennie Erik	35/5, 119/6	Ozaki Tsubasa	64/3, 64/3
Ngo Long	31/5	Ozbay Ridvan	46/1, 46/1
Ngoma Jeff	48/5, 48/5	Ozel Ali	126/4, 126/4
Niazi Ardekani Mehdi	102/2, 129/5, 102/2, 129/5	Pacha Sanjeevi Sathish	48/6, 48/6
Nicolaou Laura	17/3, 17/3	Padding Johan	3/2, 48/6, 77/4, 118/5, 3/2, 48/6, 77/4, 118/5
Nicolleau Franck	136/4, 136/4	Paglianti Alessandro	85/3, 85/3
Nie Deming	150/2, 150/2	Pain Christopher	22/1, 113/5, 22/1, 113/5
Nieckele Angela	10/3, 148/1, 148/4, 10/3, 148/1, 148/4	Pakhomov Maksim	16/3, 16/3
Niegodajew Pawel	115/6, 115/6	Paladino Emilio	9/4, 40/5
Niethammer Matthias	133/6, 133/6	Pallares Jordi	109/5, 109/5
Nihei Naoto	92/1, 92/1	Palmore John	7/3, 7/3
Nikolay Zhelonkin	107/2, 107/2	Pan Hui	33/6, 122/4, 33/6, 122/4
Nikookar Hamed	28/5, 28/5	Pan Liangming	28/3, 28/3
Nikrityuk Petr	6/6, 6/6	Pan Shucheng	49/2, 49/2
Nilsson Hakan	109/6, 109/6	Panahi Roozbeh	143/6, 143/6
Nir Avinoam	34/5	Pang Mingjun	141/2, 141/2
Niu Xiao-Juan	88/4, 127/5, 88/4, 127/5	Paoli Roberto	4/3, 4/3
Niwa Motonori	44/1, 44/1	Papavassiliou Dimitrios	110/2, 110/2
Nizamova Adelina	105/6, 105/6	Papoulias Dimitrios	122/5, 122/5
Nobes David	8/2, 8/2	Paras Spiros	78/1, 140/5, 78/1, 140/5
Nobili Clement	73/6, 73/6	Parente Alessandro	108/3, 108/3
Nomata Koji	30/3	Park Chan Woo	29/4, 29/4
Nonomura Taku	76/2, 136/2, 76/2, 136/2	Park Chansoo	24/3, 24/3
Norde Ellen	86/3, 86/3	Park Hyun Jin	88/3, 88/3
Nordlund Markus	93/1, 93/1	Park Hyungmin	19/6, 19/6
Norlier Dirceu	106/1, 108/4, 106/1, 108/4	Park Kyeong Hyeon	24/6, 24/6
Novelletto Ricardo Guilherme A.	143/1, 151/4, 143/1, 151/4	Park Seong Hyeon	120/3, 120/3
Nowak Emilia	48/4, 78/4	Passalacqua Alberto	75/4, 85/4, 75/4, 85/4
Nowakowski Andrzej	100/4, 136/4, 100/4, 136/4	Passos Agathoklis	78/1, 140/5, 78/1, 140/5
Nozaki Hiroshi	146/3, 146/3	Pasternak Lars	130/6, 130/6
Nydal Ole Jørgen	9/3, 81/3	Pathak Ashish	111/2, 111/2
O Hern Timothy	127/1, 127/1	Pauchard Vincent	68/6, 68/6
O Sullivan Paul	136/5, 136/5	Pavlidis Dimitrios	22/1, 113/5, 22/1, 113/5
Oefelein Joseph	125/3, 125/3	Pelanti Marica	31/2, 31/2
Oevermann Michael	2/2, 69/1, 2/2, 69/1	Pelin Kalle	37/3, 37/3
Offenhaeuser Philipp	104/5, 104/5	Pellegrini Laura	81/6, 81/6
Ogasawara Toshiyuki	45/4, 45/5, 45/4, 45/5	Pena Monferrer Carlos	72/3, 72/3
Ogata Koichiro	119/2, 119/2	Peng Cheng	35/3, 94/4, 35/3, 94/4
Ogata Masayuki	77/6, 77/6	Penko A.	54/1, 54/1
Ogawa Mao	134/1, 134/1	Pereira Francisco Alves	59/1, 137/3, 59/1, 137/3
Ogawa Soichiro	63/5, 63/5	Perkins Richard	14/6, 14/6
Ogawa Yuichi	92/1, 92/1	Perles Carlos Eduardo	74/3, 74/3
Ogura Takahito	50/4	Perlin Marc	64/5, 137/4, 64/5, 137/4
Oh Heechang	24/3	Perrier Hugo	13/6, 13/6
Oh Jong-Taek	1/4, 1/4	Pervunin Konstantin	11/6, 80/3, 11/6, 80/3
Ohta Haruhiko	80/6, 80/6	Pesci Chiara	6/5, 60/6, 143/5, 6/5, 60/6, 143/5
Ohta Mitsuhiro	134/1, 134/1	Peters Bernhard	111/1, 151/1
Ohtake Hiroyasu	34/4, 64/6, 34/4, 64/6	Peters E.A.J.F.	7/1, 7/1
Oike Mamoru	38/4, 38/4	Peters Frank	77/4, 118/5, 77/4, 118/5
Oikonomou George	78/1, 78/1	Petersen Alec	35/4, 35/4
Okamoto Koji	53/3, 53/3	Pettas Dionisios	127/3, 127/3
Okayama Satoshi	80/6, 80/6	Pfitzer Michael	47/4, 47/4
Okuno Kohei	56/6, 56/6	Philippe Regis	121/5, 138/6, 121/5, 138/6
Okuyama Kunito	126/5, 126/5	Phillips Bren	37/6, 37/6
Olejnik Michal	85/6, 85/6	Pialago Edward Joshua	29/4, 29/4
Olesen Anders Christian	138/5, 146/1, 138/5, 146/1	Picano Francesco	62/3, 79/3, 131/6, 139/5, 62/3, 79/3, 131/6, 139/5
Oliveira Gustavo	99/6, 99/6	Picchi Davide	97/6, 97/6
Olivieri Giuseppe	114/3, 129/1, 114/3, 129/1	Piccinni Leopardi Giacomo	90/1, 90/1
Olsen Jan Erik	101/6, 101/6	Piedfert Antoine	54/6, 54/6
Omar Rajab	49/4		

Pigeonneau Franck	22/3, 58/2, 118/2, 22/3, 58/2, 118/2	Reeks Michael	27/1, 27/1
Pillai Abhishek	101/1, 101/1	Reinecke Sebastian	82/3, 82/3
Piomelli Ugo	71/1, 71/1	Ren Zhaoxin	103/5, 103/5
Pirker Stefan	22/4, 60/4, 67/4, 93/2	Reveillon Julien	99/5, 136/1, 99/5, 136/1
Piscaglia Federico	110/6, 110/6	Revellin Remi	9/5, 56/3, 9/5, 56/3
Pischke Philipp	39/2, 137/5, 39/2, 137/5	Reviznikov Dmitry	147/6, 147/6
Pitz-Paal Robert	42/2, 42/2	Rezig Mariem	54/5, 54/5
Pivello Marcio	55/3, 55/3	Ribatski Gherhardt	41/1, 43/1
Plais Cecile	75/5, 75/5	Ribeiro Duarte Carlos Antonio	6/1
Planchette Carole	2/6, 2/6	Riber Eleonore	79/1, 79/1
Planquart Philippe	120/4, 120/4	Richenderfer Andrew	47/3, 47/3
Plesniak Michael W.	28/6, 28/6	Richter David	94/1
Plouraboue Franck	109/3, 109/3	Richter Steffen	68/5, 68/5
Podgorska Wioletta	147/3	Riedelbauch Stefan	21/2, 21/2
Poelma Christian	31/6	Rigopoulos Stelios	129/3, 129/3
Poesio Pietro	8/6, 9/2, 54/3, 97/6, 8/6, 9/2, 54/3, 97/6	Rimbert Nicolas	32/3, 43/5, 52/6, 98/5, 32/3, 43/5, 52/6, 98/5
Polifke Wolfgang	5/1, 137/1, 5/1, 137/1	Rinke Gunter	23/3
Pons Audrey	133/2, 133/2	Rioboo Romain	65/2, 65/2
Pontes Jose	99/6, 99/6	Risso Frederic	36/4, 54/6, 68/6, 36/4, 54/6, 68/6
Popinet Stephane	84/6, 84/6	Robbes Benjamin	79/1, 79/1
Portela Luis	26/5, 62/5, 85/1, 26/5, 62/5, 85/1	Roccon Alessio	111/3, 111/3
Postacchini Matteo	4/4, 4/4	Rocha Douglas Martins	18/3, 112/5, 18/3, 112/5
Potapov Alexander	82/4, 82/4	Rodrigues Romulo	67/1, 67/1
Potts Ian	27/1, 27/1	Roenby Johan	103/2, 103/2
Pouliquen Olivier	124/1, 124/1	Rogachevskii Igor	79/2, 79/2
Pouplin Amelie	117/5, 117/5	Roghair Ivo	3/3, 132/4, 144/6, 3/3, 132/4, 144/6
Pourquie Mathieu	83/6, 83/6	Roig Veronique	10/1, 45/6, 10/1, 45/6
Povolny Antonin	82/1, 82/1	Roisman Ilia V.	7/6, 8/1, 17/1, 21/4, 7/6, 8/1, 17/1, 21/4
Powell Robert	130/4, 130/4	Rollin Bertrand	121/2, 121/2
Pozorski Jacek	76/5, 78/6, 85/6, 76/5, 78/6, 85/6	Roma Alexandre	55/3, 55/3
Pozzetti Gabriele	111/1, 151/1	Romano Francesco	131/2, 139/4, 131/2, 139/4
Pradhan Aniruddhe	141/5, 141/5	Romano Giampaolo	59/1, 59/1
Prahl Wittberg Lisa	28/6, 29/1, 103/4, 28/6, 29/1, 103/4	Romer Carsten	146/1, 146/1
Prasetya Rubby	21/1, 21/1	Rong Ji Li	31/3, 31/3
Premnath Kannan	88/5, 88/5	Ropelato Karolline	82/4, 82/4
Priems Giel	103/1, 103/1	Rosa Bogdan	76/5, 101/5, 76/5, 101/5
Profeta Christophe	65/6, 65/6	Rosen Tomas	86/4, 86/4
Prosperetti Andrea	135/5, 135/5	Rosendahl Lasse	102/3, 102/3
Protasov Mikhail	107/3, 107/3	Roshchupkina Olga	32/1, 46/5, 32/1, 46/5
Pujara Nimish	94/5, 94/5	Rosseburg Annika	75/6, 75/6
Pulido Jorge	44/6, 44/6	Rosso Michele	122/6, 122/6
Punde Pranav	108/6, 108/6	Roumpea Evangelia	121/4, 121/4
Punekar Hemant	83/4, 115/5, 83/4, 115/5	Roustaie Ali	128/3, 128/3
Purwono Baskoro Adiputro	91/1, 91/1	Roy Shantanu	32/2, 32/2
Qian Lijuan	96/2, 96/2	Rubner Michael	37/6, 37/6
Qin Chao	43/2, 43/2	Rudek Arthur	93/4
Qiu Guizhi	132/6, 132/6	Rudiger Frank	29/6, 29/6
Qu Chen	19/2, 19/2	Rudolf von Rohr Philipp	40/4, 82/2
Qu Zhiguo	101/2, 101/2	Ruegg Oliver	40/4, 40/4
Quan Qing	56/4, 56/4	Ruiz Angulo Angel	150/5, 150/5
Quentin Lindfield Roberts	149/4, 149/4	Ruiz-Rus Javier	28/1, 28/1
Queteschiner Daniel	67/4, 67/4	Rulliere Romuald	65/1, 65/1
Quintero Brian	109/4, 109/4	Russ Gerald	93/4
Rachner Michael	139/1, 139/1	Russo Maria Elena	114/3, 114/3
Radcliff Thomas	56/5, 56/5	Ruyer Pierre	29/2, 82/5, 29/2, 82/5
Radl Stefan	16/1, 104/1	Ryu Kyungha	146/6, 146/6
Raessi Mehdi	111/2, 111/2	Rzehak Roland	51/4, 123/5, 51/4, 123/5
Rahmani Mona	27/2, 140/1, 27/2, 140/1	Sabat Macole	87/4, 87/4
Rai Kenta	119/1, 119/1	Sabban Lilach	95/1, 95/1
Raimundo Pedro Maximiano	19/5, 75/5, 19/5, 75/5	Sabino Renzo	145/6, 145/6
Ramcke Thomas	47/4, 47/4	Saccone Giacomo	80/1, 80/1
Ramin Imani Jajarmi	133/1, 133/1	Sadatomi Michio	91/5, 91/5
Ramoino Luca	42/4, 42/4	Sadlo Filip	134/2, 137/2, 134/2, 137/2
Rana Basanta Kumar	15/6	Saeedipour Mahdi	60/4, 60/4
Rapberger Richard	64/2	Saenen Tom	105/5, 105/5
Rarrek Andreas	41/3, 41/3	Saenger Alexander	139/1, 139/1
Rasmuson Anders	67/5	Saffar Avval Majid	62/6, 62/6
Rasteiro Maria Graca	116/3, 116/3	Saga Mariko	59/4, 59/4
Ravnik Jure	149/1	Sagadin Gregor	149/1
Razgallah Intissar	150/3, 150/3	Saha Debashish	64/1, 64/1
		Sahu Akhilesh Kumar	126/1, 126/1

Saidj Faiza	49/5	Serra Sonia	136/6, 136/6
Saito Kiyoshi	138/1, 138/1	Serres Marion	138/6, 138/6
Saito Takayuki	30/3, 96/4	Sevilla Alejandro	28/1, 63/3, 28/1, 63/3
Saito Yasushi	23/6, 134/6, 23/6, 134/6	Sewerin Fabian	129/3, 129/3
Sajjadi Hasan	8/5	Sgrott Junior Oscar	143/1, 151/4, 143/1, 151/4
Sakakibara Jun	10/5, 10/5	Shafieefar Mehdi	143/6, 143/6
Salais Clement	146/2, 146/2	Shakouchi Toshihiko	73/2, 96/3, 73/2, 96/3
Salatino Piero	91/3, 114/3, 129/1, 91/3, 114/3, 129/1	Shang Yidan	1/3, 1/3
Saliakellis Panagiotis	53/6, 53/6	Shao Changxiao	118/3
Salinas Pablo	22/1, 113/5, 22/1, 113/5	Sharaf Safa	19/3, 19/3
Salmanzadeh Mazyar	8/5	Sharay Elena	133/5, 133/5
Salveti Maria Vittoria	98/1, 98/1	Sharma Yagya	15/6
Salvo Ricardo	143/4, 145/5, 143/4, 145/5	Shcheglov Igor	69/3, 69/3
Sampath Kaushik	23/2, 23/2	Shemer Lev	34/2, 34/2
Sanada Toshiyuki	16/5, 16/6, 36/5, 144/5, 146/3	Shen Hongwei	17/4, 17/4
Sander Soren	111/6, 111/6	Sheremet Alexandru	4/4, 4/4
Sanders Sean	8/2, 49/6, 8/2, 49/6	Shevchenko Natalia	32/1, 46/5, 32/1, 46/5
Sanderse Benjamin	5/6, 5/6	Shi Bohui	12/3, 12/3
Sanjay Vatsal	33/4	Shimizu Hiromu	96/4
Saraceno Luca	43/4, 43/4	Shimomura Masaya	100/3, 100/3
Saramito Pierre	22/3, 118/2, 22/3, 118/2	Shin Seungwon	60/3, 66/2, 60/3, 66/2
Saranjam Nazli	18/5	Shingote Chinmay	32/4, 32/4
Sardina Gaetano	23/1, 23/1	Shimoto Yasuhisa	80/6, 80/6
Sarmadi Parisa	149/5, 149/5	Shirai Shota	45/5, 45/5
Sarshar Mohammad	62/1, 62/1	Shmueli Havatzelet	79/6, 79/6
Sasaki Koichi	11/3, 11/3	Sho Sakai	126/3, 126/3
Sasic Srdjan	46/4, 51/2, 66/4, 67/5, 74/5, 82/6, 109/6	Shoeibomrani Pejman	35/5, 35/5
Sato Ryo	120/2, 120/2	Shtork Sergey	113/2, 120/5, 113/2, 120/5
Sato Ryoichi	69/5, 69/5	Shukla Ratnesh	38/6, 38/6
Sato Takahiro	42/3, 42/3	Shum Kivan Francis	79/1, 79/1
Sattelmayer Thomas	34/1, 139/2, 34/1, 139/2	Siconolfi Lorenzo	98/1, 98/1
Scardovelli Ruben	33/1, 119/5	Siegel Michael	6/2
Scargiali Francesca	85/2, 85/2	Sierakowski A.	135/5, 135/5
Schaefer Jan Erik	75/6, 75/6	Sierra Del Rio Jorge	150/4, 150/4
Schafer Jan	114/2	Sikovskiy Dmitrii	17/6, 17/6
Schlegel Fabian	118/1, 118/1	Silveira Neto Aristeu	55/3, 55/3
Schleicher Eckhard	42/2	Simeonov Julian	4/5, 148/3, 4/5, 148/3
Schlottke Jan	123/6, 123/6	Simmons Mark	48/4, 78/4, 89/6, 106/3, 121/3
Schlottke Karin	137/2, 137/2	Simoens Serge	121/5, 121/5
Schlueter Michael	60/6, 68/5, 75/6, 96/1, 114/4, 122/1, 60/6, 68/5, 75/6, 96/1, 114/4, 122/1	Simon Sven	23/3
Schmalfuss Silvio	147/1, 147/1	Simonin Olivier	16/2, 17/5, 73/4, 87/5, 90/3, 90/6, 100/5, 112/2, 112/4, 117/1, 126/4, 16/2, 17/5, 73/4, 87/5, 90/3, 90/6, 100/5, 112/2, 112/4, 117/1, 126/4
Schmitt Christoffer	90/1, 90/1	Singh Anugrah	127/6, 127/6
Schmitt Damien	107/1, 107/1	Singh Brajesh Kumar	5/2, 5/2
Schneider Fabio	67/1, 67/1	Singh Chamkor	103/6, 103/6
Schneider Wilhelm	110/1	Singha Sanat Kumar	38/3, 74/6, 38/3, 74/6
Schneiderbauer Simon	60/4, 93/2, 60/4, 93/2	Singhal Arpit	16/1, 16/1
Schranner Felix	6/4, 6/4	Sinha Nityanand	151/5, 151/5
Schremb Markus	17/1, 17/1	Sinibaldi Giorgia	35/1, 35/1
Schubert Markus	32/2, 46/3, 61/2, 80/4, 32/2, 46/3, 61/2, 80/4	Sinkov Konstantin	87/1, 87/1
Schumann Heiner	81/3	Skjaeraasen Olaf	59/2, 59/2
Schumichen Michel	29/6, 29/6	Skjetne Paal	101/6, 101/6
Schurr Daniela	23/3	Skripkin Sergey	120/5, 120/5
Schutte Koen	26/5, 26/5	Sloan Nicholas	39/3, 39/3
Schwarz Mark Philip	114/5, 114/5	Smagin Irina	34/5
Schwenck Daniel	111/6, 111/6	Snape Collin	42/5, 42/5
Schyns Marc	120/4, 120/4	Snook Braden	117/3, 124/1, 117/3, 124/1
Scorsim Oliver	117/1, 117/1	Soderberg Daniel	116/4, 116/4
Sebilleau Julien	19/1, 29/2, 19/1, 29/2	Soemoto Kazuhiko	32/5, 32/5
Seden Martin	68/1, 68/1	Soerensen Henrik	102/3, 102/3
Sederman Andrew	14/4, 14/4	Sofonea Victor	17/5, 17/5
Seegel Karl Erik	69/3, 69/3	Soldati Alfredo	22/6, 23/1, 71/1, 84/3, 111/3, 22/6, 23/1, 71/1, 84/3, 111/3
Seidel-Morgenstern Andreas	107/5, 107/5	Soligo Giovanni	71/1, 71/1
Sekiguchi Yoshito	10/5, 10/5	Solimene Roberto	91/3, 91/3
Selenius Pasi	124/2, 124/2	Solomenko Zlatko	39/5, 39/5
Selle Laurent	33/2, 33/2	Solotych Valentin	52/2, 52/2
Sellier Antoine	58/2, 58/2	Solsvik Jannike	143/2, 143/2
Seo Gwang Hyeok	25/6, 33/5, 25/6, 33/5	Soltner Helmut	39/2, 39/2
Seon Thomas	28/4, 28/4		
Serfaty Ricardo	49/1, 55/3, 49/1, 55/3		

Sommerfeld Martin	63/2, 89/2, 109/4, 130/6, 143/1, 147/1, 151/4, 151/6, 63/2, 89/2, 109/4, 130/6, 143/1, 147/1, 151/4, 151/6	Takahira Hiroyuki	2/5, 45/4, 45/5, 2/5, 45/4, 45/5
Son Hong Hyun	25/6, 33/5, 25/6, 33/5	Takamasa Tomoji	35/6, 54/2, 35/6, 54/2
Song Kyusang	25/6, 33/5, 25/6, 33/5	Takano Kenji	59/4, 59/4
Song Pan	56/4, 148/2, 56/4, 148/2	Takano Tomoyuki	38/4, 38/4
Sontti Somasekhara Goud	84/5, 84/5	Takashi Hibiki	119/4
Soos Miroslav	64/1, 64/1	Takata Yasuyuki	72/6, 89/4, 72/6, 89/4
Soteriou Marios	56/5, 111/5, 56/5, 111/5	Takei Masahiro	32/6, 32/6
Sotiropoulos Fotis	90/4, 90/4	Takemura Fumio	95/4, 95/4
Sou Akira	21/1, 21/1	Takenaka Nobuyuki	104/6, 104/6
Souto Izabel	148/4, 148/4	Takesue Syuichi	100/3, 100/3
Souza De Castro Marcelo	5/5, 18/4, 145/4, 5/5, 18/4, 145/4	Takeuchi Shintaro	7/2, 61/3, 87/2, 7/2, 61/3, 87/2
Spandan Vamsi Arza	115/1, 115/1	Tamburini Alessandro	85/2, 85/2
Spasova Berta	133/3, 133/3	Tammisola Outi	120/6, 138/3, 120/6, 138/3
Spelt Peter	1/6, 15/4, 39/5, 58/5, 87/6, 94/2, 1/6, 15/4, 39/5, 58/5, 87/6, 94/2	Tan Chao	67/3, 81/5, 144/3, 67/3, 81/5, 144/3
Spesivtsev Pavel	87/1, 87/1	Tan Junhua	112/3, 112/3
Spiteri Raymond	118/6, 118/6	Tan Lianghai	132/4, 132/4
Splawski Andrew	122/5, 122/5	Tanaka Mitsuru	72/4, 72/4
Sposobin Andrey	147/6, 147/6	Tanaka Yohsuke	14/1, 14/1
Squires Kyle	87/3, 87/3	Tanaka Yuta	126/5, 126/5
Stafford Christopher	69/2, 69/2	Tandon Mohit P.	25/1, 108/6, 25/1, 108/6
Steiner Helfried	65/4, 65/4	Tang Hong	147/2, 147/4
Stel Henrique	145/6, 145/6	Tanguy Sebastien	1/5, 39/6, 1/5, 39/6
Stengel Bruno	133/3, 133/3	Tani Hiroumi	139/3, 139/3
Sticko Simon	86/2, 86/2	Tani Shunsuke	14/1, 14/1
Stofer Cordeiro De Farias Paula	98/3, 98/3	Taniere Anne	13/3, 52/6, 76/4, 116/2, 150/3, 150/4, 13/3, 52/6, 76/4, 116/2, 150/3, 150/4
Stroe Rodica-Elisabeta	140/6, 140/6	Tanimoto Takuma	73/2, 73/2
Strom Henrik	46/4, 82/6, 46/4, 82/6	Tasaka Yuji	64/2, 64/3, 88/3
Strumpf Erik	32/1, 46/5, 32/1, 46/5	Tatekura Yuki	11/3, 11/3
Stumpf Michael	76/6, 76/6	Taylor Alex M.K. P.	30/6, 70/4, 30/6, 70/4
Su Guan-Yu	20/1, 20/1	Tekavcic Matej	36/1, 36/1
Subramaniam Shankar	77/5, 131/5, 151/3, 77/5, 131/5, 151/3	Tenneti Sudheer	131/5, 131/5
Subudhi Sudhakar	106/2, 106/2	Terasaka Koichi	142/3
Sudo Takayuki	38/4, 38/4	Terashima Hiroshi	139/3, 139/3
Sugimoto Katsumi	104/6, 104/6	Terekhov Viktor	16/3, 16/3
Sugitani Yosuke	96/3, 96/3	Thaker Jignesh	12/2, 66/6
Sugiura Toshihiko	150/6, 150/6	Thakur Siddarth	71/6, 71/6
Sugiyama Akifumi	92/1, 92/1	Thevenin Dominique	15/2, 128/6, 135/3, 15/2, 128/6, 135/3
Sulaiman Mostafa	109/3, 109/3	Thevenin Dominique	104/4, 104/4
Sum Amadeu	9/1, 9/1	Thibault Daniel	113/4, 113/4
Sumbekova Sholpan	21/5, 21/5	Thiruvalluvan Sujatha	80/4, 80/4
Sun Bo	131/5, 131/5	Krushnathej	
Sun Chao	46/2, 92/4, 131/4, 46/2, 92/4, 131/4	Thomassen Magnus	64/4, 64/4
Sun Chengzhen	117/4, 117/4	Thome John	52/2, 99/6, 105/5, 52/2, 99/6, 105/5
Sun Tiezhi	30/1, 30/1	Thoming Jorg	101/4, 101/4
Sun Yunkai	68/2, 68/2	Thommes Markus	24/4, 24/4
Sundararajan Sriram	77/5, 77/5	Tian Lin	124/4, 124/4
Sunden Bengt	47/2, 47/2	Tian Yunya	99/1, 99/1
Sussman Mark	36/3, 36/3	Timmermann Jens	114/4, 114/4
Suto Hitoshi	123/1, 123/1	Timoshevskiy Mikhail	11/6, 80/3, 11/6, 80/3
Svensson Martin	128/2, 128/2	Tingyen Chiu	74/1, 74/1
Swailles David	69/2, 69/2	Tinon Emmanuelle	33/2, 33/2
Swerin Agne	116/4, 116/4	Tisler Sergei	69/3, 69/3
Swiderski Karol	42/4, 72/1	Titta Andrea	15/4, 15/4
Szeliga Nicolai	68/5, 68/5	Tognotti Leonardo	108/3, 108/3
Szewc Kamil	85/6, 97/1, 85/6, 97/1	Tokuda Yomei	92/1, 92/1
Ta Thi Thanh Mai	118/2, 118/2	Tomas Jurgen	104/4, 104/4
Tagawa Yoshiyuki	42/1, 47/5, 69/5, 42/1, 47/5, 69/5	Tomiyama Akio	26/2, 26/3, 52/5, 55/1, 60/6, 120/2, 26/2, 26/3, 52/5, 55/1, 60/6, 120/2
Taghavi Mohammad	128/3, 128/3	Tondi Cristina	137/3, 137/3
Taitel Yehuda	79/4, 79/4	Tonini Simona	38/1, 60/5, 38/1, 60/5
Takada Hitoshi	38/4, 38/4	Tontowi Alva Edy	91/1, 91/1
Takada Naoki	49/3, 49/3	Torabi Soroosh	51/1, 51/1
Takagi Shu	10/5, 114/1	Torczynski John	127/1, 127/1
Takahashi Keiko	53/1, 53/1	Toschi Federico	53/4, 53/4
Takahashi Shun	76/2, 76/2	Toshida Kiyoshi	123/1, 123/1
Takahashi Shun	136/2, 136/2	Toutant Adrien	123/2, 123/2
		Tozzi Emilio	130/4, 130/4

Tretola Giovanni	91/3, 132/3, 91/3, 132/3	Vasiliev Nikolay	20/4, 107/3, 20/4, 107/3
Trinanes Pablo Garcia	16/2, 16/2	Vasques Joao	4/6, 4/6
Trinkle Christine	51/1, 51/1	Vasudevan Raghavan	126/1, 126/1
Troiano Maurizio	91/3, 91/3	Vaudor Geoffroy	33/3, 33/3
Tronchin Thibaut	20/3, 20/3	Vazquez Gonzalez Thibaud	31/4, 31/4
Tropea Cameron	7/6, 8/1, 17/1, 21/4, 7/6, 8/1, 17/1, 21/4	Velasco Pena Hugo Fernando	18/2, 143/3
Tryggvason Greta	15/3, 33/1, 51/6, 15/3, 33/1, 51/6	Velez Sebastian	150/4, 150/4
Tsamopoulos John	127/3, 128/1, 127/3, 128/1	Vena Patrizio	142/1, 142/1
Tsaoulidis Dimitrios	77/2, 77/2	Vercauteren Frank	35/5, 35/5
Tschisgale Silvio	3/5, 3/6, 3/5, 3/6	Verdin Patrick	130/2, 130/2
Tsoy Mikhail	113/2, 113/2	Verhille Gautier	109/3, 109/3
Tsubone Hiroaki	100/3, 100/3	Vernet Anton	109/5, 109/5
Tsuchiya Katsumi	57/1, 80/2, 57/1, 80/2	Verzicco Roberto	115/1, 115/1
Tsuji Takuya	119/1, 119/1	Vichansky Alexander	55/5, 122/5, 135/1
Tsuji Yoshiyuki	84/2, 84/2	Vicquelin Ronan	102/5, 102/5
Tsujimoto Koichi	73/2, 96/3, 73/2, 96/3	Vidal Valerie	138/6, 138/6
Tsukahara Takahiro	55/6, 55/6	Vie Aymeric	87/4, 87/4
Tu Chengxu	151/2, 151/2	Vikulov Roman	108/6, 108/6
Tu Jiuyan	1/3, 56/1, 123/4, 124/4, 1/3, 56/1, 123/4, 124/4	Villa Fabio	65/2, 65/2
Tu Qiuya	68/2, 68/2	Villafane Laura	35/2, 35/2
Tummers Mark	129/5, 129/5	Villar Millena Martins	55/3, 55/3
Turangan Cary	136/3, 136/3	Ville Alopaeus	85/5, 135/2, 85/5, 135/2
Turney Damon	44/6, 44/6	Villvock Jorn	43/6, 43/6
Tuteja Anish	64/5, 137/4, 64/5, 137/4	Vinai Paolo	66/4, 66/4
Twerda Aris	26/5, 26/5	Vincent Stephane	52/1, 130/3, 131/3, 52/1, 130/3, 131/3
Tyagi Parul	50/1, 50/1	Vines Chris	90/2, 90/2
Ueda Yoshikatsu	92/1, 92/1	Vinh David	148/6, 148/6
Ueno Masahiro	64/6, 64/6	Vinkovic Ivana	116/5
Uesawa Shinichiro	81/1, 81/1	Violato Daniele	5/1, 5/1
Uhlemann Margitta	48/2, 48/2	Visintainer Robert	32/4, 32/4
Uhlmann Markus	44/2, 62/2, 76/6, 112/1, 117/2, 44/2, 62/2, 76/6, 112/1, 117/2	Vittori Giovanna	44/2, 44/2
Ullmann Amos	105/3	Vlachogiannis Michalis	6/3, 6/3
Umemura Naoki	80/2, 80/2	Vo Minh	110/2, 110/2
Umemura Yutaka	139/3, 139/3	Vorobyev Maxim	141/1, 141/1
Urban Christian	96/1, 96/1	Voth Greg	95/2, 95/2
Urmancheev Said	105/6, 145/2, 105/6, 145/2	Voulgaropoulos Victor	26/1, 150/1, 26/1, 150/1
Uttinger Walter	42/4, 42/4	Vreman Bert	37/3, 57/6, 93/6
Vahaji Sara	56/1, 56/1	Vu Trung-Thanh	98/2, 110/5, 98/2, 110/5
Valat Marc	133/2, 133/2	Wachs Anthony	112/1, 140/1, 112/1, 140/1
Valdettaro Lorenzo	92/6, 92/6	Wack Jonas	21/2, 21/2
Valencia Castillo Cesar Manuel	43/4, 43/4	Waclawczyk Tomasz	113/6, 113/6
Valiorgue Pierre	121/5, 121/5	Wada Yoshitaka	21/1, 21/1
Valizadeh Ziba	25/3, 25/3	Wagner Lisa-Maria	135/3, 135/3
Valluri Prashant	59/6, 59/6	Wagner Stefan	57/2, 57/2
Vamling Lennart	51/2, 51/2	Wakimoto Tatsuro	25/2, 32/5, 25/2, 32/5
Van Boven Gijs	62/5, 62/5	Wakui Takaaki	72/6, 72/6
Van Crombrugge Yann	121/3, 121/3	Waldeck Steffen	69/6, 69/6
Van Den Akker Harry	86/1, 86/1	Walther Jens	4/2, 4/2
Van Der Geld Cees	103/1, 103/1	Walzel Peter	24/4, 34/6, 24/4, 34/6
Van Der Hoef Martin	132/5, 132/5	Wan Kaidi	122/3, 122/3
Van Der Weide Edwin	38/5, 86/3, 38/5, 86/3	Wang Bing	70/1, 103/5, 142/5, 70/1, 103/5, 142/5
Van Der Wijngaart Wouter	133/1, 133/1	Wang Fei	122/4, 122/4
Van Hout Rene	79/5, 95/1, 79/5, 95/1	Wang Guiquan	116/6, 116/6
Van Nimwegen Dries	62/5, 62/5	Wang Guo-Xiang	53/2, 53/2
Van Putten Dennis	106/6	Wang Haigang	68/2, 132/6, 68/2, 132/6
Van Rijsbergen Martijn	59/1, 59/1	Wang Hanfeng	134/5, 134/5
Van Sint Annaland Martin	3/3, 132/4, 144/6, 3/3, 132/4, 144/6	Wang Hang	14/3
Van Steijn Volkert	85/1, 85/1	Wang Hao	17/4, 17/4
Van Wachem Berend	3/4, 12/6, 49/1, 55/3, 59/5, 131/1, 3/4, 12/6, 49/1, 55/3, 59/5, 131/1	Wang Hengjie	122/6, 122/6
Van Wyk Stevin	28/6, 28/6	Wang Jinshi	2/1, 2/1
Van Zwieten Joost	5/6, 5/6	Wang Kejin	77/5, 77/5
Vani Manoj	83/4, 83/4	Wang Lian-Ping	35/3, 94/4, 101/5, 35/3, 94/4, 101/5
Vanni Marco	117/6, 147/3	Wang Lifeng	2/4, 2/4
Varaksin Aleksei	20/4, 107/3, 147/6, 20/4, 107/3, 147/6	Wang Qiong	101/2, 101/2
Varchanis Stylianos	127/3, 127/3	Wang Sheng	97/3, 97/3
Variano Evan	94/5, 94/5	Wang Shuli	83/3, 83/3
		Wang Wei	12/3, 17/2, 26/4, 34/3, 56/4, 99/1, 144/2, 146/5, 148/2, 150/1, 12/3, 17/2, 26/4, 34/3, 56/4, 99/1, 144/2, 146/5, 148/2, 150/1
		Wang Xin	149/3, 149/3

Wang Y.	135/5, 135/5	Yamamoto Shota	47/5, 47/5
Wang Yi	24/2, 24/2	Yamamoto Yasufumi	25/2, 25/2
Wang Yingchun	77/1, 77/1	Yamashita Masatoshi	84/2, 84/2
Wang Yuchen	35/6, 54/2, 35/6, 54/2	Yamazaki Isako	142/3
Wang Yuebing	140/4, 140/4	Yan Jianguo	33/6, 33/6
Wang Yufei	112/6, 112/6	Yan Junjie	2/1, 77/1, 2/1, 77/1
Wang Zhaoting	149/3, 149/3	Yan Qing	127/5, 127/5
Wang Zhihua	122/3, 122/3	Yanai Sayuri	30/3
Ward Kevin	98/4, 98/4	Yang Eunjin	121/1, 121/1
Ward Matthew	149/4, 149/4	Yang Fuling	73/5, 74/1, 76/3, 97/2, 73/5, 74/1, 76/3, 97/2
Wasserfall Joram	137/5, 137/5	Yang Guang	144/1, 144/1
Watahiki Soma	44/1, 44/1	Yang Hongkun	39/2, 39/2
Watanabe Ayumu	44/1, 44/1	Yang Juheng	56/4, 148/2, 56/4, 148/2
Watanabe Hiroki	77/6, 77/6	Yang Junfeng	16/4, 16/4
Watanabe Masao	2/5, 11/3, 16/5, 16/6, 2/5, 11/3, 16/5, 16/6	Yang Wei	116/1
Watson Nicholas	40/6, 59/3	Yang Wuqiang	132/6, 132/6
Waz Emmanuel	50/3, 78/3, 50/3, 78/3	Yang Xiaogang	42/5, 42/5
Weber Manfred	46/3, 46/3	Yang Xuegeng	48/2, 48/2
Weber Markus	46/3, 46/3	Yang Yan	4/2, 83/3, 4/2, 83/3
Weber Paul S.	6/5, 6/5	Yang Zhuqiang	122/4, 122/4
Weber Paul Stephan	143/5, 143/5	Yang Zonglin	20/2, 20/2
Wedberg Rasmus	11/4, 11/4	Yao Chao	25/5, 25/5
Wehliye Hashi Wehliye	26/1, 26/1	Yao Jun	26/6, 68/3, 26/6, 68/3
Wei Anyang	126/6, 126/6	Yasukawa Yoshihito	15/1
Wei Jinjia	141/2, 141/2	Yates Matthew	93/5, 93/5
Weickgenannt Christina	21/4, 21/4	Yazdani Miad	56/5, 56/5
Weigand Bernhard	60/5, 134/2, 137/2, 60/5, 134/2, 137/2	Ye Jiamin	132/6, 132/6
Welch Samuel	88/5, 88/5	Yeo Kyongmin	132/2, 132/2
Wen Chuang	4/2, 83/3, 4/2, 83/3	Yeung Hoi	36/2, 50/6, 67/2, 36/2, 50/6, 67/2
Westende Jos	35/5, 35/5	Yiantsios Stergios	53/6, 53/6
Westerweel Jerry	79/5, 79/5	Yin Chungen	108/2, 108/2
Willen D.	135/5, 135/5	Yin Zhaoqin	149/2, 151/2
Wilson Robert	131/1, 131/1	Yokohata Hideaki	21/1, 21/1
Wiratni Wiratni	91/1, 91/1	Yonemoto Yukihiro	56/2, 56/2
Wittgens Bernd	73/1, 73/1	Yoo Junsoo	23/4, 80/5, 23/4, 80/5
Woerner Martin	63/4, 68/4, 63/4, 68/4	Yoshida Hiroyuki	81/1, 81/1
Wollborn Tobias	57/5, 57/5	Yoshimori Wataru	119/1, 119/1
Wolters Jorg	39/2, 39/2	Yoshimura Kazuki	15/1
Wrobel Luiz	122/3, 122/3	You Donghyun	41/6, 41/6
Wu Dan	100/1, 100/1	Yu Wenchao	116/5
Wu Fan	139/6, 139/6	Yu Yong	19/2, 19/2
Wu Fan	145/1, 145/1	Yu Zi-Tao	29/3, 112/6, 29/3, 112/6
Wu Haihao	12/3, 12/3	Yuan Qizheng	33/6, 33/6
Wu Hao	67/3, 144/3, 67/3, 144/3	Yuan Ye	81/5, 81/5
Wu Jiandong	17/4, 17/4	Yue Xiaoqing	149/3, 149/3
Wu Juliana	30/1, 47/6, 30/1, 47/6	Yun Byongjo	45/3, 45/3
Wu Ke	11/2, 11/2	Yuto Oi	42/1, 42/1
Wu Wangxia	70/1, 70/1	Zaabout Abdelghafour	64/4, 64/4
Wu Wen	71/1, 71/1	Zabarankin Michael	34/5
Wu Wenjuan	1/2, 1/2	Zabiego Magali	125/6, 125/6
Wu Zan	47/2, 47/2	Zabkova Katerina	63/1, 63/1
Wu Zhipeng	132/6, 132/6	Zadravec Matej	149/1
Wucherpennig Thomas	75/6, 75/6	Zadrrazil Ivan	5/3, 23/5, 27/3, 99/3, 5/3, 23/5, 27/3, 99/3
Xayasenh Arunvady	78/3, 78/3	Zaki Tamer	17/3, 17/3
Xia Guodong	144/1, 144/1	Zaleski Stephane	10/6, 33/1, 10/6, 33/1
Xia Jun	122/3, 122/3	Zamansky Remi	129/4, 129/4
Xia Junjie	135/6, 135/6	Zander Hans-Jorg	83/2, 83/2
Xia Kai	2/1, 2/1	Zangana Mohammed	27/5
Xiang Min	123/4, 123/4	Zanotta Marie-Line	121/5, 138/6, 121/5, 138/6
Xiang Sujuan	140/4, 140/4	Zare Marjan	128/3, 149/4, 128/3, 149/4
Xie Zhihua	22/1, 113/5, 22/1, 113/5	Zarghami Ahad	86/1, 86/1
Xiong Yan	44/4, 44/4	Zeghloul Ammar	49/5
Xu Wei	61/6, 61/6	Zeigarnik Yuriy	20/4, 20/4
Xu Wenkai	40/1, 40/1	Zeng Zhuoxiong	114/6, 114/6
Xue Xinzhi	23/2, 23/2	Zenit Roberto	55/2, 150/5, 55/2, 150/5
Xue Yuqing	25/5, 25/5	Zha Lu	124/6, 124/6
Yadav Ashutosh	32/2, 32/2	Zhakebayev Dauren	99/2, 99/2
Yadav Sumedh	141/5, 141/5	Zhang Baili	30/4, 110/4, 30/4, 110/4
Yamada Saori	80/2, 80/2	Zhang Gangfu	95/3
Yamaguchi Eri	146/3, 146/3	Zhang Huiqiang	103/5, 103/5
Yamaguchi Seiichi	138/1, 138/1	Zhang Jian	19/2, 19/2
Yamamoto Daisuke	80/6, 80/6	Zhang Liang	29/3, 112/6, 29/3, 112/6
Yamamoto Keita	21/2, 21/2		

Zhang Meng	99/1, 144/2, 146/5, 99/1, 144/2, 146/5	Zhendong Yang	25/4, 25/4
Zhang Mingyu	84/1, 84/1	Zheng Chuguang	44/4, 100/1, 124/6, 134/5, 44/4, 100/1, 124/6, 134/5
Zhang Peiyu	142/5, 142/5	Zhou Chengsi	9/5, 9/5
Zhang Qian	113/1, 113/1	Zhou Guanyu	121/5, 121/5
Zhang Qiaoling	25/4, 25/4	Zhou Kun	116/1
Zhang Wenbin	70/1, 70/1	Zhou Lixing	114/6, 114/6
Zhang Wenhao	10/5, 10/5	Zhou Xiangyang	24/2, 24/2
Zhang Xiaoling	149/3, 149/3	Zhou Yixian	82/5, 82/5
Zhang Xiaopeng	29/3, 29/3	Zhou Zhifu	53/2, 53/2
Zhang YiLing	29/3, 29/3	Zhu Ge	33/6, 33/6
Zhang Yuning	20/6, 20/6	Zhu Weibing	10/4, 24/5, 88/2, 10/4, 24/5, 88/2
Zhang Ziwei	83/1, 83/1	Zhurov Andrei	145/2, 145/2
Zhao Hui	126/6, 145/1, 126/6, 145/1	Ziegenhein Thomas	19/4, 72/2, 95/5
Zhao Jianfu	11/2, 11/2	Zinnatullin Rasul	74/2, 74/2
Zhao Liang	88/4, 127/5, 149/6, 88/4, 127/5, 149/6	Ziskind Gennady	79/6, 79/6
Zhao Lihao	8/4, 94/6, 102/4, 8/4, 94/6, 102/4	Zonta Francesco	22/6, 71/1, 84/3, 111/3, 22/6, 71/1, 84/3, 111/3
Zhao Quanbin	77/1, 77/1	Zoueshtiagh Farzam	98/4, 98/4
Zhao Yanlin	26/6, 68/3, 26/6, 68/3	Zummo Giuseppe	43/4, 43/4
Zhao Yuyang	144/4, 144/4	Zun Iztok	37/4, 55/1, 37/4, 55/1
Zhe Jiang	127/5, 149/6, 127/5, 149/6	Zuzio Davide	24/1, 118/4, 24/1, 118/4





## NUMERICAL ANALYSIS OF THE FLOW AND PARTICLE PATTERN IN THE HUMAN NASAL CAVITY

Ali Farnoud, University of Heidelberg, Germany; Xinguang Cui, Lawrence Berkeley National Laboratory, USA; Ingo Baumann, Medical Center of the University of Heidelberg, Germany; Eva Gutheil, University of Heidelberg, Germany

A numerical study of the dispersion and deposition of drugs sprayed into the nasal airway is presented. An unsteady Eulerian-Lagrangian model is developed to simulate the airflow and the particle dispersion and deposition in a realistic human nasal airway using the software package OpenFOAM. A three-dimensional model of the nasal airway of a male patient is constructed from CT scans provided by the Heidelberg Medical School, where the surface and volume grids are generated using the software packages ImageJ, NeuRA2, meshLab, and Ansys ICEM-CFD 11.0. Large eddy simulations with the Smagorinsky sub-grid scale model are performed to study the human inhalation rates including laminar, transitional, and turbulent regimes. The results reveal that the flow velocity has its maximum and minimum values in the nasal valve region and the tips of the meatuses, respectively. The pressure on the surface of the nostril is maximal since the air impinges on the surface in this region. Most of the particles deposit in the anterior and posterior regions of the nasal cavity because of the changes in flow direction in this area. Furthermore, the influence of the initial particle diameter and the inhalation flow rate on the deposition efficiency is evaluated.

## NANOPARTICLE DEPOSITION ANALYSIS IN HUMAN AND RAT NASAL CAVITIES: A NUMERICAL COMPARISON STUDY

Jingliang Dong, RMIT University, Australia; Yidan Shang, RMIT University, Australia; Kiao Inthavong, RMIT University, Australia; Jiyuan Tu, RMIT University, Australia; Rui Chen, National Center for Nanoscience and Technology of China, China; Chunying Chen, National Center for Nanoscience and Technology of China, China

This paper presents a comparative nanoparticle deposition study using anatomically realistic models of a human and rat nasal cavity. Numerical simulation accuracy was assured through comparison with published relevant studies in literature. The flow patterns and detailed particle deposition of 1 nm, 10 nm and 100 nm particles in the human and rat nasal passages were numerically estimated. Using a surface unwrapping technique, the deposition hot spots quantified by deposition flux were fully accessed over the whole nasal cavity. Direct exposure extrapolation was performed based on the regional deposition flux. Particle penetration depth quantified by deposition intensity along nasal passageway is presented. The nanoparticle deposition results demonstrate that the human nasal model exhibits a similar particle filtration trend for all three types of nanoparticles, where the middle passage region receives the most particle exposure dose, followed by the vestibule and the pharynx regions. While for the rat model, majority of the 1 nm particles were blocked by the vestibule region with almost no deposition intensity in the posterior regions. This study bridges the in-vitro exposure experiments and in-vivo nanomaterials toxicity studies by applying a normalized particle exposure dose, and can contribute towards the inter-species exposure extrapolation.

\* Support from ARC (Project ID: DP120103958) and NSFC (21277080, 21477029) are gratefully acknowledged.

## NUMERICAL SIMULATIONS OF NUCLEATE BOILING

Gregory Huber, INP Toulouse, France; Sebastien Tanguy, INP Toulouse, France; Catherine Colin, INP Toulouse, France

This work presents some recent results on the bubble growth and detachment on a heated plate obtained by Direct Numerical Simulation. A numerical tool based on a Level-Set / Ghost Fluids method is used. Previous work focused on the boiling of a static bubble immersed in an overheated liquid has allowed validating the numerical methods. The objective is now to validate the same numerical methods for the case of nucleate boiling on a heated plate. The growth of a water vapor bubble on a heated plate is simulated and compared to experimental data. The convergence study shows that a good agreement is obtained when a sufficiently fine mesh is used. Indeed, in the vicinity of the gas-liquid interface, the computational cells size must be at least one hundred times smaller than the final equivalent radius of the bubble. Parametric studies are also performed in order to observe the influence of the wall superheat and of the contact angle upon the departure bubble radius and the departure frequency. Water is a partially wetting fluid and our numerical results lead us to the conclusion that, for this case, the micro-layer effects could be neglected. Indeed the classical micro-layer models leads to an estimation of the apparent contact angle very close to the Young contact angle for values between 30 to 60°. The contribution of the evaporation heat flux at the contact line is also small on the global mass and energy balances. So, another study focused on nucleate boiling of a perfectly wetting fluid is currently ongoing and should better highlight the contribution of micro-layer effects.

\* The support of the national french space agency (CNES) is acknowledged.

## THROMBUS/BUBBLES FORMATION AND MULTIPHASE FLOW IN BLOOD VESSEL AFTER LASER IRRADIATION

Wenjuan Wu, Xi'an Jiaotong University, China; Dong Li, Xi'an Jiaotong University, China; Bin Chen, Xi'an Jiaotong University, China

Laser therapy has been widely used for treating Port-Wine Stains and other vascular lesions via the principle of selective photothermolysis. There are mainly two treatment end-points to destroy target blood vessel in clinic. One is the vessel rupture under high energy dose, which would cause purpura on the patients' face. The other one advocate using mild laser energy to induce thrombus within the vessel lumen. The subsequent blocking of the blood flow would cause the apoptosis of the target blood vessel, while avoiding the bleeding underneath the skin. However, the occurrence of the thrombus formation and resulting blood blocking involves the complex multiphase flow and corresponding heat/mass transfer phenomenon, which has not been carefully investigated till now. In this paper, a hamster dorsal window chamber model was constructed, through which the target blood vessels are exposed to the microscopy, to investigate the dynamic change of intravascular blood during pulsed laser treatment. In experiments, a specific series of fast events occurred after a 1064nm Nd:YAG laser irradiation and recorded by a high speed camera, including blood coagulation (thrombus), vasoconstriction, formation of bubbles and vessel wall rupture after multiple laser pulses irradiation. The dynamic characteristics of the thrombus formation and corresponding multiphase blood flow in vessel lumen were analyzed. Furthermore, the threshold radiant exposure for bubble formation in blood is recorded. These results may provide useful instructions for the treatment of PWS by near-infrared laser.

\* Support from the National Natural Science Foundation of China (Key Program, No.51336006)

## EVAPORATION HEAT TRANSFER COEFFICIENT OF R410A INSIDE MICRO/MINI-CHANNEL

Jong-Taek Oh, Chonnam National University, Republic of Korea; Nguyen Ba Chien, Chonnam National University, Republic of Korea; Kwang-Il Choi, Chonnam National University, Republic of Korea

This study demonstrates two phase flow boiling heat transfer coefficient of R410A in micro/mini-channels. The experimental data were conducted in horizontal stainless steel tubes with the inner diameter of 0.5 mm, 1.5 mm and 3.0 mm. The testing conditions were performed with the mass fluxes range from 200 to 500 kg/m<sup>2</sup>s, the heat fluxes from 5 to 15 kW/m<sup>2</sup>, the saturation temperature of 0, 5 and 10°C and the vapor quality from 0.1 to dry-out. The effects of mass flux, heat flux, tube diameters and saturation temperature on the heat transfer coefficient were analyzed. The experimental data were also compared with some well-known heat transfer coefficient correlations. A modified heat transfer coefficient correlation was developed with present experimental data.

## DIRECT NUMERICAL SIMULATION OF INTERACTING EVAPORATING DROPLETS

Thibault Dairay, Ecole Centrale de Lyon, France; Aurore Naso, CNRS, France; Peter Spelt, Laboratoire de Mécanique des Fluides et d'Acoustique, France

Dispersed two-phase flows with phase change play an important role in industrial applications as well as in environmental science. In the present study, numerical simulations of the evaporation of finite size droplets are carried out. The main goal of this project is to use DNS results to advance the theoretical framework of large-scale modelling of suspensions in a turbulent flow. The simulations are based on an in-house finite difference code where the Level Set Method is used to capture accurately the interface motion while the Ghost Fluid Method is used to prescribe suitable jump conditions. As validation cases, numerical simulations of the evaporation of an isolated static drop will be presented and compared with an analytical solution for the temperature and mass fraction fields. The evaporation of droplets placed in a laminar flow will then be considered as a first stage of our study; a comparison with experimental data obtained in our facility and with a simple model of evaporation will be presented. We finally report on simulation results for a suspension of sedimenting evaporating droplets that interact hydrodynamically as well as through the temperature and mass fraction fields.

\* This work was granted access to the HPC resources of IDRIS under the allocations 2015-100561 made by GENCI.

## RESEARCH ON MARANGONI CONDENSATION MODES OF ETHANOL-WATER MIXTURES ON A HORIZONTAL SURFACE

Yong Li, Xi'an Jiaotong University, China; Jinshi Wang, Xi'an Jiaotong University, China; Nana Chen, Xi'an Jiaotong University, China; Junjie Yan, Xi'an Jiaotong University, China; Kai Xia, Xi'an Jiaotong University, China

The study of condensation modes is important for the understanding of the condensation mechanism and further theoretical research on Marangoni condensation. Marangoni condensation of ethanol-water mixtures on a horizontal surface was investigated experimentally during the time-series process herein. During this process, lots of condensate droplets appeared on the condensing surface, and then these droplets grew up and partially coalesced; finally formed a condensate film covered the surface. The process time was counted under different experimental conditions. The results indicated that, with the increase in the initial vapor-to-surface temperature difference and the decrease in the ethanol vapor concentration, the process time became shorter. The condensation mode transformed from dropwise to filmwise with the increase in the initial vapor-to-surface temperature differences at the beginning of the time-series process. And the higher ethanol vapor concentration and vapor velocity extended the ranges where the dropwise mode existed signally. The droplet size was obtained by the method of image edge detection. The maximum droplet radius increased with the increase in the initial vapor-to-surface temperature difference and with the decrease in the ethanol vapor concentration.

\* Supports from the National Natural Science Foundation of China (No. 51436006, No.51206133) are gratefully acknowledged.

## CAVITATION MODELLING IN MICRO CHANNELS

Nuray Kayakol, Robert Bosch, Turkey

Cavitating flow is desired to maintain mass flowrate constant in a solenoid valves of diesel common rail injector. Material loss due to cavitation erosion is inevitable because of poor pressure recovery in the micro channels of the injector. Two-phase flow characteristics of a micron sized components of a solenoid valve are modelled with a mixture model approach, which is suited for bubble-liquid flow. The predictive accuracy of a cavitation model based on Rayleigh-Plesset equation and cavitation erosion based on work-hardening of material is dependent on non-geometrical model coefficients. A standard cavitation analysis which provides the distribution of vapour volume fraction and pressure needs to be used together with a cavitation erosion model that considers bubble dynamics, detached turbulent flow and material properties. It is essential to accurately account for turbulence effect on cavitation erosion by using turbulent kinetic energy. Microjet velocity calculated from the bubble interface velocity produces pressure impulse which is smaller than the material yield strength. The study provides basic understanding for flow aggressiveness and material characteristics for future sophisticated modeling of the cavitation induced erosion process.

## NUMERICAL SIMULATION OF VAPOR-GAS BUBBLE DYNAMICS BASED ON MOLECULAR GAS DYNAMICS

Takahiro Nagayama, Hokkaido University, Japan; Misaki Kon, Hokkaido University, Japan; Kazumichi Kobayashi, Hokkaido University, Japan; Masao Watanabe, Hokkaido University, Japan; Hiroyuki Fujii, Hokkaido University, Japan; Hiroyuki Takahira, Osaka Prefecture University, Japan

It is well known that the evaporation and condensation play significant roles in the collapse of cavitation bubble. To deal with the vapor flow due to the evaporation and condensation at a vapor-liquid interface, the molecular gas dynamics analysis based on the Boltzmann equation is essential. Hence, we investigated the dynamics of a spherical bubble filled with the vapor-gas mixture on the basis of the molecular gas dynamics analysis by numerical simulation. We used the Andries-Aoki-Perthame model for the Boltzmann equation of the binary mixture inside the bubble, and the Fujikawa & Akamatsu equation for the equation for radial motion of bubble. In this study, we investigated a pair of governing parameters: the initial number density ratio (the ratio of the number density of gas to that of vapor inside the bubble), and the condensation coefficient included in the kinetic boundary condition. We confirmed that at the final stage of the collapse of the bubble, the condensation mass flux decreases rapidly due to the gas accumulated near the bubble wall, and that the pressure inside the bubble becomes higher than that of the bubble filled with only vapor.

\* Supports from JSPS KAKENHI 26630043 and 25820038 are gratefully acknowledged.

## LES MODELING OF CAVITATION FLOW IN A DIESEL INJECTOR NOZZLE

Boxiong Chen, Chalmers University of Technology, Sweden; Zachary Falgout, Chalmers University of Technology, Sweden; Mark Linne, Chalmers University of Technology, Sweden; Michael Oevermann, Chalmers University of Technology, Sweden

Large Eddy simulations resembling the flow inside a diesel injector nozzle under cavitation conditions are performed in the current study. A pressure based solver implemented within an open source CFD package (OpenFOAM) that takes into account the compressibility of both liquid and vapor phases is used upon which a new cavitation model that accounts for the viscous terms of Rayleigh-Plesset equation is developed to enhance numerical stability. The volume of fluids method (VOF) is adopted under a homogeneous mixture assumption. We look into the comparison between the simulation results of the new model and that from the standard Sauer-Schnerr model. Experiments were also performed, which provide a stronger basis for evaluation of the new model's performance by allowing direct comparison of the model results with real physics. In these experiments, backlit images captured with a high-speed camera are used to capture the dynamics of cavitation in the orifices along a single line of sight. Back-lit imaging provides clear images of the cavitation structures in the flow due to the refractive index difference across multiphase interfaces. Computations also show that the numerical method combined with cavitation models is able to resemble the cavitation phenomena that are observed in the experiments.

## SIMULATION OF A HIGH PRESSURE FUEL PUMP, INCLUDING CAVITATION EFFECTS, WITH TWO MULTIPHASE MODELS

Foivos C. Koukouvinis, City University London, UK; Ioannis Karathanassis, City University London, UK; Jason Li, Caterpillar Fuel Systems, USA; Lifeng Wang, Caterpillar Fuel Systems, USA; Manolis Gavaises, City University London, UK

The present work outlines two methodologies for the simulation of high pressure fuel pumps, commonly used in modern Diesel engines for providing high pressure fuel to the injectors. The described methodologies are a 2-phase homogenous mixture model, with mass transfer terms to include cavitation development, and a single phase barotropic homogenous equilibrium model. Both models aim to describe the significant density ratio of cavitating flows; the fundamental difference is that the 2-phase model assumes a finite mass transfer due to cavitation, whereas in the barotropic model the mass transfer rate is infinite, thus the liquid/vapour is always in thermodynamic equilibrium. Liquid compressibility effects are also included in both methodologies. The pump is a positive displacement pump, involving significant geometrical changes (valve opening/closing, piston motion), which are handled with mesh motion and boundary condition manipulation (switching interior boundaries from interfaces to walls). Comparison of the two methodologies shows a very similar cavitation pattern; closer examination of the results shows that the 2-phase model gives more coherent cavitating structures, whereas the barotropic model predicts more diffused cavitation. On the other hand the barotropic model does not need the solution of a vapour fraction equation, thus it is simpler and faster to run.

## TERNARY DROP COLLISIONS

Carole Planchette, Graz University of Technology, Austria; Hannes Hinterbichler, Graz University of Technology, Austria; Guenter Brenn, Graz University of Technology, Austria

It has been proposed recently to use collisions of drops for producing advanced particles or well-defined capsules, or to perform chemical reactions where the merged drops constitute a micro-reactor. For all these promising applications, it is essential to determine whether the merged droplets remain stable after the collision, forming a single entity, or if they break up. This topic, widely investigated for binary collisions of miscible and immiscible liquid drops, is quite unexplored for ternary collisions. The present study aims to contribute to this field by experimentally investigating collisions between three equal-sized drops of the same liquid arranged centri-symmetrically. Three monodisperse drop generators are simultaneously operated to obtain controlled ternary drop collisions. The collision outcomes are observed on images obtained by visualization of the processes and compared to those of binary collisions, which were also investigated for reference. Similar to binary collisions, a regime map is built, showing coalescence and bouncing as well as reflexive and stretching separation. Significant differences are observed in the boundaries of these regimes. For head-on collisions, modeling the process as two successive steps of drop deformation after the impact shows a weaker dissipative influence in the ternary than in binary collisions.

\* HH gratefully acknowledges support from the Research Center of Pharmaceutical Engineering GmbH at Graz University of Technology.

## ANALYSIS OF THE MECHANISMS FOR GAS BUBBLE AND DROPLET BREAKUP IN TURBULENT FLOWS

Ronnie Andersson, Chalmers University of Technology, Sweden

In this work the basic mechanisms responsible for gas bubble and droplet breakup under turbulent flow conditions are investigated, by means of high resolution CFD simulations and by use of high resolution measurements. The three dimensional and transient analysis shows very good agreement with experiments with respect to: deformation rate, breakup time, fragmentation pattern, and daughter size distributions [1]. More importantly this methodology allows in-depth analysis of the interplay between turbulence and the fluid particles, allowing the phenomena to be understood better. It allows the interaction to be visualized and quantified in detail during the entire interaction period. This knowledge is required to improve the breakup rate models proposed in the literature. Poor understanding of turbulence and particle interaction explains why several assumption are introduced in these models. Unfortunately these assumption degrades even the most physics based model to a tunable model, and a universal model is not yet proposed despite more than 60 years effort. The results presented in this study significantly improves the understanding of the phenomena. It is shown that some assumptions made in the literature need to be relaxed while others are falsified. Consequently, the results will have an impact on the development of breakup rate models in the future.

\* Support from Swedish Research Council, grant 2013-5964, is gratefully acknowledged.

[1] Andersson R., and Helmi A., Applied Mathematical Modelling, 38, pp. 4186-4196, 2014.

## MULTI-SCALE MODELING OF BUBBLE COALESCENCE AND BREAKUP IN HETEROGENEOUS BUBBLY FLOWS

Alessandro Battistella, Eindhoven University of Technology, The Netherlands; Ivo Roghair, Eindhoven University of Technology, The Netherlands; Martin Van Sint Annaland, Eindhoven University of Technology, The Netherlands

Bubble column reactors are widely used in the chemical industry and detailed modeling of heterogeneous bubbly flows is thus of continuous interest. Models for breakage and coalescence of bubbles play a crucial role in determining the bubble size distribution but, despite the widespread use in industry, accurate knowledge of these phenomena is still lacking.

To study the complex behavior of bubbly flows, a multi-scale modeling approach is adopted where the use of small scale, high detail models provides insight in the hydrodynamics and closures needed for larger-scale. In this work, we look for a better correspondence between intermediate size (Euler-Lagrange models) and large-scale (Euler-Euler models), especially concerning binary bubble interactions, such as collisions, coalescence and breakup.

We start by using the discrete bubble model (Euler-Lagrange model), in which bubbles are tracked individually, allowing to obtain statistical insight in bubble interactions in a deterministic fashion. The translation of these statistics (e.g. positioning and size of bubbles after break-up) towards closures for the Euler-Euler model (e.g. Two-Fluid model treating both the liquid and the gas as continuous interpenetrating fluids) will be studied, together with the impact on the large-scale behavior of the bubble column.

## A NEW APPROACH TO DEFINE A NON-ITERATIVE IMMERSSED BOUNDARY METHOD FOR SPHERICAL PARTICLES OF ARBITRARY DENSITY RATIO

Silvio Tschisgale, TU Dresden, Germany; Tobias Kempe, ILK Dresden, Germany; Jochen Frohlich, Technische Universität Dresden, Germany

A vast number of numerical schemes for the simulation of particle-laden or bubble-laden flows has been developed in recent years. Especially for low density ratios or vanishing particle density, the strong coupling of the fluid and the solid part requires special numerical techniques to reach numerical stability. Usually, an iterative procedure is applied to balance fluid forces and particle forces at the interface, which increases the computational effort. In contrast to that practice, this paper presents a simple non-iterative coupling of fluid and spherical particles, based on an immersed boundary method. The scheme is unconditionally stable for arbitrary density ratios and does not require any iteration, which outperforms previous approaches. The contribution presents the underlying methodology and its algorithmic realization including an assessment of convergence. Validation cases illustrate performance and versatility. Furthermore, it is shown how standard codes with explicit coupling can easily be adapted, which substantially extends their field of application. Finally, the enhancement of the method for arbitrary shaped particles is briefly presented.

\* Computational time was provided by ZIH, Dresden. Funding was partially provided by the Helmholtz-Alliance LIMTECH.

## NOVEL PHENOMENOLOGICAL MODELS FOR VISCOUS BINARY DROPLET COLLISIONS

Giulia Finotello, Eindhoven University of Technology, The Netherlands; Niels G. Deen, Eindhoven University of Technology, The Netherlands; Johan Padding, Eindhoven University of Technology, The Netherlands; J. A. M (hans) Kuipers, Eindhoven University of Technology, The Netherlands; Alfred Jongsma, Tetra Pak CPS, The Netherlands; Fredrik Innings, Tetra Pak CPS, The Netherlands

In this work we assess the role of viscosity in droplet-droplet collisions. In spray drying operations liquids are first atomized to droplets and then dried by a stream of hot air. Everywhere in the spray, but in particular close to the liquid nozzle, droplets collide with each other and agglomerate, which determines the quality of the produced powder. It is therefore of prime importance to be able to predict the outcome of droplet-droplet collisions for the modeling and development of spray drier processes. Because of the drying of the droplets the liquid viscosity increases by many orders of magnitude, while the collision outcome (bouncing, coalescence, agglomeration or breakup) strongly depends on the droplet viscosity. The innovative aspect of this work is the explicit modelling of the effect of viscosity on the droplet-droplet collision outcome. The role of viscosity is characterized by the capillary number. Together with other non-dimensional parameters (Weber number, size ratio and impact parameter) it provides a complete description of the collision outcome. These outcomes can be represented in 3D regime maps. Our model is tuned with the aid of DNS employing the Volume of Fluid method and experimental data obtained from two droplet generators. Simulations enable us to analyze the mechanism of viscous dissipation energy during the collision. The results of this investigation can be used for the closure of macro scale models for spray drier operations.

## GENERALIZED IMMERSSED BOUNDARY METHOD FOR UNSTRUCTURED MESH

Mohd Hazmil Syahidy Abdol Aziz, Universiti Teknologi Malaysia, Malaysia; Berend Van Wachem, Imperial College London, UK

The original immersed boundary method (IBM), uses a discrete delta function in a direct forcing scheme to take into account immersed solid boundaries. In CFD, the specifics of the discretization of the IBM have to satisfy a number of restrictions, for instance moment conditions. A simple application of the discretization on unstructured meshes will generally violate these restrictions, consequently the no-slip condition cannot be ensured. The specifics concerning the discretization are discussed in the current contribution and the difficulties will be discussed. This contribution will also present a generalization of the IBM. This novel approximation of discrete delta function according to local mesh configuration results in relatively small stencil sizes while retaining a smooth distributed force. To evaluate the boundary force, an evaluation that is based on the Navier-Stokes equations is proposed. In addition, different types of interpolation methods and weighting functions for force spreading are discussed. Validation and performance of the formulation will also be presented for typical cases, such as flow through settling spheres. This method presents an alternative that is applicable for unstructured meshes which can be optimized according to the features of the flow.

## AN IMMERSSED BOUNDARY METHOD FOR COMPLEX-SHAPED BUBBLES REPRESENTED BY SPHERICAL HARMONIC FUNCTIONS

Benjamin Krull, TU Dresden, Germany; Silvio Tschisgale, TU Dresden, Germany; Jochen Frohlich, Technische Universität Dresden, Germany

An immersed boundary method for complex-shaped bubbles is proposed. Each bubble is represented by a series of spherical harmonic functions, which covers all shapes of simply connected bubble geometries occurring in the various regimes of bubbly flow. Due to the analytical representation of the bubble surface, curvatures and normal vectors can easily be computed with high accuracy, even for strongly deformed bubbles. An appropriate algorithm for the temporal evolution of the bubble surface is derived, based on the local balance of forces. It is simple, intrinsic, does not require any reconstruction step and strictly conserves the volume of the bubble. Furthermore, the algorithm shows a vast stability range which allows simulations using coarse grids and large time steps. It is suitable for simulations of large numbers of complex-shaped bubbles for arbitrary fluid-bubble density ratios. To demonstrate the functionality of the proposed model, simulations of bubbles in different regimes are presented, such as wobbling and spherical-cap shaped bubbles.

\* The present work was partly funded by the Helmholtz Alliance LIMTECH, project A5. Computational time was provided by ZIH, Dresden.

#### IMMERSED BOUNDARY – THERMAL LATTICE BOLTZMANN METHOD FOR PARTICLE SEDIMENTATION IN NON-NEWTONIAN LIQUID

Amin Amiri Delouei, University of Bojnord, Iran; Nazari Mohsen, University of Shahrood, Iran; Kayhani Mohammad Hasan, University of Shahrood, Iran; Goodarz Ahmadi, Clarkson University, USA

In this investigation the direct-forcing immersed boundary – thermal lattice Boltzmann methods is used to investigate the non-isothermal sedimentation of a spherical particle in the pseudo-plastic and dilatant liquids. The proposed direct numerical simulation employs the 4-point interface scheme for considering the presence of moving Lagrangian boundary points on a fixed Eulerian fluid domain. The split-forcing algorithm is used to apply the boundary force density in the lattice Boltzmann equations which recovers the Navier-Stokes equations to second-order accuracy. The solution method is successfully validated by using the previous experimental and numerical results. The effects of non-Newtonian power-law index ( $0.1 \leq n \leq 1.2$ ) and temperature-dependent viscosity ( $0 \leq \beta \leq 0.5$ ) of non-Newtonian power-law fluids are studied for a falling particle at fixed and varied (transient) temperatures for different generalized Archimedes and Prandtl numbers. The results show that the effects of temperature-dependent viscosity are more significant at higher values of non-Newtonian behaviour index and larger values of generalized Archimedes number. For the cases of constant and transient particle temperature, the differences are more noticeable at higher non-Newtonian behaviour index and larger values of generalized Prandtl number.

#### CONTRAIL MODELING AND SIMULATION

Roberto Paoli, University of Illinois at Chicago, USA

Contrails are ice clouds that form by the condensation of water-vapor exhaust from aircraft engines. When contrails spread to form cirrus clouds, they can persist for hours, extending over areas of several square kilometers. These "contrail cirrus," which artificially increase cloudiness and become almost indistinguishable from natural cirrus, are the most uncertain contributors to Earth's radiative forcing attributed to the aviation sector. The presence of contrail cirrus thus represents a source of increasing concern for scientists and policymakers as the volume of air traffic continues to grow.

It has recently become possible to simulate contrail evolution for long periods after their formation using cloud-resolving models such as large-eddy simulations. We review the main physical processes driving the formation and evolution of ice crystals, and present simulation efforts in the four phases of contrail evolution: the jet phase where contrails form; the vortex and dissipation phases where contrails develop and transform in the aircraft wake and interact with the atmospheric environment; and the diffusion phase where atmospheric processes drive contrail cirrus dispersion. We finally provide recommendations for further work.

#### THE EFFECT OF BED GEOMETRY AND COHERENT VORTICES ON INCIPENT GRANULAR BED MOTION IN OSCILLATORY FLOW: LABORATORY MEASUREMENTS AND NUMERICAL SIMULATIONS

Julian Simeonov, US Naval Research Laboratory, USA; Donya Frank, National Research Council Fellow, USA; Joseph Calantoni, US Naval Research Laboratory, USA

Laboratory measurements of oscillatory flow over granular beds have suggested that incipient grain motion will be induced by the pressure gradients (Sleath parameter) in flows with large accelerations, by the mean shear stresses (Shields parameter) in flows with small accelerations, and by the combined effects for intermediate flows. Here, we investigate the effect of the bed geometry and coherent vortex structures on the condition for incipient motion using DNS and Lagrangian observations in an oscillatory tunnel flow. In the DNS, the hydrodynamics is fully resolved everywhere except in the gap between colliding particles when the gap becomes smaller than the grid step. Mechanical contact is accounted for by a soft-sphere approach: Normal and tangential forces are modeled using a linear elastic-plastic law and a history dependent friction law, respectively. In the experiments, we used Lagrangian tracking and PIV to measure the motion of the grains and fluid, respectively. The bed geometries consisted of a pair of exposed beads over a regular hexagonally packed bed, as well as a random loose packed bed of beads. The stability of the pair increases as the pair separation decreases. We consider a modified Shields-Sleath incipient motion criterion to incorporate the effect of bed geometry and coherent vortices.

\* Support provided under base funding to the Naval Research Laboratory from the Office of Naval Research.

#### TANGENTIAL INLET SUPERSONIC SEPARATORS: A NOVEL APPARATUS FOR GAS PURIFICATION

Chuang Wen, Changzhou University, China; Yan Yang, Changzhou University, China; Jens Walther, Technical University of Denmark, Denmark

A novel supersonic separator with a tangential inlet is designed to condense and separate water and heavy hydrocarbons from natural gas. The dynamic parameters of natural gas in the supersonic separation process and the effects of operating parameters on the gas flows were numerically calculated using the Reynolds stress turbulence model with a real gas model. The results show that natural gas expands in the supersonic separator to supersonic velocities with resulting in low pressure (25 bar, from about 101 bar) and temperatures ( $-75^\circ\text{C}$ , from  $30^\circ\text{C}$ ), which causes the condensation and nucleation of some components. The higher back pressure induces the shock position shift forward from the diffuser to nozzle. The shock goes into the diffuser when the pressure recovery efficiency is less than 0.7. If the pressure recovery efficiency is more than 0.8, the shock moves into the nozzle, resulting in the re-evaporation of the condensed components. The increase in the inlet pressure will shift the shockwave location towards the nozzle throat. The shockwave position moves towards the nozzle exit with the rise of the inlet temperature. The Mach number will be less than unity at the nozzle throat and the flow will never be choked if the flow rate is lower than the designed.

#### ESCOSED: OBSERVATIONS OF MORPHODYNAMICS DURING BORA AT THE MOUTH OF THE MISA RIVER

Joseph Calantoni, US Naval Research Laboratory, USA; Alexandru Sheremet, University of Florida, USA; Maurizio Brocchini, Università Politecnica delle Marche, Italy; Matteo Postacchini, Università Politecnica delle Marche, Italy

The shallow mouth of the Misa River, Senigallia, Italy is exposed to wind and waves from the Adriatic Sea and is vulnerable to morphodynamic activity during even moderate storm events (e.g., winter Bora). Sediment loads and transport patterns may be strongly influenced by the confluence of fine cohesive suspended sediment contained in the discharge from the river mixing with coarser sandy material stirred up by waves impinging on the river mouth. Observations of rapid changes in bed elevation along a transect extending offshore of the river mouth were made using a combination of instruments deployed from 23-27 January 2014 at two locations in roughly 5 m water depth and 6 m water depth. At the 5 m water depth location over 0.4 m of deposition was observed roughly during a 6-hour period. Similarly, at the 6 m water depth location nearly 0.2 m of deposition was observed roughly over a 6-hour period with approximately a two-hour time lag. The onset of deposition was concurrent with a change in direction of the mean currents at both locations and a change in direction of wave skewness observed at the 5 m water depth location. We hypothesize that sandbar migration was responsible for the observed changes in bed elevation at both locations. Our analysis will focus on sediment transport modeling to explain rates of deposition and time lag of the observed changes in bed elevation at the 5 m and 6m water depth locations.

\*Support provided by Office of Naval Research in Wash., DC and London, UK (N62909-13-1-N020) and Italian RITMARE (SP3-WP4).

#### STUDY OF A STATIONARY TAYLOR BUBBLE USING THE BRIGHTNESS BASED LASER INDUCED FLUORESCENCE TECHNIQUE (BBLIF)

Joao Vasques, University of Nottingham, UK; Andrey Cherdantsev, Kutateladze Institute of Thermophysics, Russian Federation; David Hann, University of Nottingham, UK; Buddhikha Hewakandamby, University of Nottingham, UK; Barry James Azzopardi, University of Nottingham, UK

Slug flow is one of the most common features in vertical gas-liquid two-phase flows. Its complexity due to the fast movement of the Taylor bubbles makes measurements of its hydrodynamics difficult. Consequently, the measurement of parameters such as entrainment and re-coalescence rates and the falling film profile have not been fully achieved. These factors have a direct impact on the gas exchange process between the Taylor Bubble and its surroundings, thus affecting the successful prediction of the pressure drop across the pipe. To overcome such problems, experiments were carried out on a specially designed apparatus to generate a stationary Taylor bubble.

This paper reports the measurements of the parameters mentioned above at the Taylor bubble-liquid slug interface.

The measurements were achieved using Brightness Based Laser Induced Fluorescence (BBLIF) technique, which offers superior spatial and temporal resolution. This provides detailed information on wave propagation in the film in terms of velocity, amplitude and frequency and on gas entrainment, which consequently allows for a complete system characterisation.

The observations lead to identification of two new mechanisms of gas entrainment, while the re-coalescence behaviour was visually observed for the first time.

## ACOUSTICAL CHARACTERISTICS OF TWO-PHASE HORIZONTAL INTERMITTENT FLOW THROUGH AN ORIFICE

Joohwa Sarah Lee, Audi AG, Germany; Daniele Violato, TNO, The Netherlands; Wolfgang Polifke, Technische Universität München, Germany

Acoustic phenomena in two-phase flow enjoy particular interest from industry, since significant noise may be generated, for example, by large volumetric changes associated with flow through a control valve. In the present study, the acoustic effects of a horizontal intermittent water-air flow through an orifice are investigated experimentally. The two-phase flow is generated by injecting air at a rate of 0.01 to 3.2 [g/s] to water flowing in a pipe with diameter of 25 mm at a given rate in the range from 20 to 800 [g/s], where the pressure varies from 1.5 to 6 bar at ambient temperature. Orifice diameters of 2 mm, 5 mm and 10 mm are investigated. Unsteady pressure fluctuations are recorded to provide data for spectral analysis by pressure transducers located upstream and downstream of the orifice. The flow regime is visually recognized as slug and plug flow. The slug and plug frequency lies between 0.1 and 5 Hz and primarily rises with the higher water mass flow rate. The corresponding Strouhal number declines with the increasing air mass flow rate. The power spectral density of the pressure fluctuation increases with a growth rate,  $f^{5/3}$ , in the low frequency range up to the slug frequency and then decays with a rate of  $f^{-5/2}$  up to 10 kHz. In plug flow regime a hump appears in the spectra between 100-500 Hz. This pattern is apparent downstream of the orifice.

\* We gratefully acknowledge the support from European Union through the FP7 Marie-Curie ITN project, FlowAirS (contract no. 289352).

## SIMULTANEOUS BRIGHTNESS-BASED AND PLANAR LASER-INDUCED FLOUORESCENCE MEASUREMENT OF DOWNWARDS ANNULAR FLOW

Jae Sik An, Imperial College London, UK; Andrey Cherdantsev, Kutateladze Institute of Thermophysics, Russian Federation; Ivan Zadrazil, Imperial College London, UK; Christos N. Markides, Imperial College London, UK

The liquid film surface in annular flow is covered by complex waves of a broad range of length scales, amplitudes, and curvatures. In addition, gas bubbles are entrained into the film and liquid droplets are torn from the film surface. To the best of our knowledge, there is no experimental technique at present that can be used for reliable measurements of local film thickness in this type of flow. In this work, two different approaches of the laser-induced fluorescence (LIF) technique, brightness-based LIF (BBLIF) and planar LIF (PLIF), were applied simultaneously to quantitatively and qualitatively investigate the liquid film topology in downwards co-current gas-liquid annular flow in a vertical pipe with a nominal bore of  $32.4 \pm 0.4$  mm. These two non-intrusive high-resolution techniques were used for flows that covered a liquid Reynolds number range of  $Re_L = 150 - 1500$  and a gas velocities range of  $VG = 0 - 39$  m/s. The comparison has shown good overall agreement between the two LIF approaches, but different challenges are seen with each approach at different circumstances. In addition, the film thickness measurements with BBLIF and PLIF were also compared with film thickness data extracted from simultaneous high-speed visualisation and a twin-wire conductivity probe.

\* The Consortium on Transient and Complex Multiphase Flows and Flow Assurance, and the support from the Engineering and Physical Sciences Research Council through Grant EP/K003976/1 are gratefully acknowledged.

## VISCOUS OIL-WATER HORIZONTAL WAVY STRATIFIED FLOW: CFD AND EXPERIMENTAL DATA COMPARISON

Ricardo Avila, University of São Paulo - USP, Brazil; Andre Arroxellas, State University of Campinas - UNICAMP, Brazil; Marcelo Souza De Castro, State University of Campinas - UNICAMP, Brazil; Oscar Mauricio Hernandez Rodriguez, University of São Paulo, Brazil

Two-phase wavy stratified flow is of common occurrence in directional oil wells and has been used to avoid the formation of emulsions in pipelines. The energy losses and high costs related to the fluids' displacement have been subject of interest for the industry, especially in offshore production. The stability of a flow pattern in specific conditions allows the energy consumption reduction and increases the production, also reducing project and operational costs. In this work, CFD (Computational Fluid Dynamics) is used in an attempt to better understand the interfacial phenomena related to the interfacial shear stress terms, as well as predicting the interfacial wave characteristics, pressure drop and volumetric fraction. Experimental data of interfacial wave characteristics, amplitude, length and wave velocity were compared with CFD results; the same was done for pressure drop and volumetric fraction from experimental data base were compared with predictions from phenomenological models and CFD results. Although the volumetric fraction data were satisfactorily predicted by one of the theoretical models and CFD, the same cannot be said about the pressure drop and interfacial wave characteristics. The comparisons suggest an investigation focused on the closure relations.

## SPATIAL- AND TIME-RESOLVED MEASUREMENTS OF LIQUID HOLD-UP DISTRIBUTION IN A PACKED BED USING ELECTRICAL RESISTANCE TOMOGRAPHY AND VOIDAGE PROBES

Brajesh Kumar Singh, Indian Institute of Technology Delhi, India; Ekta Jain, Indian Institute of Technology Delhi, India; Vivek Buwa, Indian Institute of Technology Delhi, India

Spatial- and time-resolved measurements of liquid hold-up distribution in a cylindrical packed bed were carried out using non-invasive electrical resistance tomography (ERT) and invasive "in-house" developed voidage probes. Experiments were performed in a co-current down-flow gas-liquid packed bed of 0.19 m ID and 1.5 m height. Water ( $4-10$  kg/m<sup>2</sup>s) and air ( $0.2-2$  kg/m<sup>2</sup>s) were used as working fluids and alumina particles ( $ds = 3-5$  mm) was used as the solid phase. These mass superficial gas and liquid velocities correspond to trickle to pulsating flow regimes. The local liquid hold-up fluctuations and radial profiles of time-averaged liquid hold-up measured using ERT and voidage probes were compared. Further, the effect of gas-liquid distributor (uniform vs. local) on local liquid distribution will be investigated. The present study will help to verify the ability of ERT to capture instantaneous local liquid hold-up fluctuations under different flow regimes and will help to understand the dynamics of gas-liquid flow in packed beds.

## MECHANISTIC SIMULATION OF VERTICAL SLUG FLOW WITH THE ONE-DIMENSIONAL TWO-FLUID MODEL

Francesco Galleni, Imperial College, UK; Raad Issa, Imperial College, UK

Numerical prediction of slug flow in vertical pipe utilizing the "slug capturing" numerical methodology is presented. In this method, the development of the flow and its characteristics are obtained solving the one-dimensional transient two-fluid model equations. In addition to its simplicity, this approach has the advantage of being almost completely mechanistic: the flow field develops naturally from the initial conditions as part of the transient calculation, thus minimizing the necessity of empirical correlations to model the slug dynamics. The "slug capturing" technique has been successfully applied in horizontal and slightly horizontal geometries, but the application to vertical flow is new. In the present work, the method was found to be able to capture the formation of slugs automatically in vertical pipes. Several simulations have been carried out for a large number of flow conditions and the results were compared with experimental data, for characteristics such as frequencies and velocities of the slugs. It is found that the agreement is remarkable.

## COMPARISON OF NUMERICAL METHODS FOR SLUG CAPTURING WITH THE TWO-FLUID MODEL

Maurice Hendrix, Delft University of Technology, The Netherlands; Ivar Eskerud Smith, NTNU, Norway; Joost Van Zwieten, Delft University of Technology, The Netherlands; Benjamin Sanderse, Shell Technology Centre Amsterdam, The Netherlands

In the petroleum industry multiphase flow occurs when transporting oil and gas through long multiphase pipeline systems. The behaviour of the flow can take many forms, depending on parameters like fluid velocities, pipe properties and fluid properties. An important flow regime is slug flow, in which liquid pockets, separated by gas bubbles, propagate in an alternating fashion with high speed along the pipeline. Such slugs have a large influence on the sizing of receiving facilities such as slug catchers or separators. The industry uses various flow models for slug flow, but there is a need for increased accuracy. A promising approach is using so-called slugcapturing though the accurate numerical solution of the one-dimensional two-fluid model. That approach is believed to be capable of describing the transition from stratified flow to slug flow see e.g. (Issa & Kempf 2003).

In this paper we investigate the capability of the two-fluid model to describe slug flow by employing three different numerical discretization techniques: classical finite volume, discontinuous Galerkin, and a finite volume Lagrangian approach. We show that the conditional well-posedness of the two-fluid model requires careful discretization techniques in order to obtain converging solutions upon mesh refinement, for both academic and real-life test cases.

## EFFECTS OF SURFACE ROUGHNESS AND INTERPARTICLE COLLISIONS ON ELBOW EROSION

Vinicius Vagundes Dos Santos, Federal University of Uberlândia, Brazil; Francisco Jose De Souza, Federal University of Uberlândia, Brazil; Carlos Antonio Ribeiro Duarte, Federal University of Uberlândia, Brazil

The nature of erosive wear involves so many variables that its modelling became a major challenge for both material and computational experimentation. In this sense, many studies aim at acquiring a better understanding of the basic mechanisms over carefully controlled operating conditions such as particle concentration, flow speed, etc. On the other hand, relatively little attention has been given to the correlation between erosion damage and more fundamental parameters such as surface roughness and coefficients of friction. Using an Euler-Lagrange approach, the erosion of a 90° degree elbow is numerically investigated. After a validation based on experimental data, the numerical sensitivity of the results to the surface roughness and coefficients of friction is evaluated for a range of values. One, two and four-way coupled simulations of the gas-solid flow are also analyzed and a detailed explanation of the different mechanisms of erosion is assessed. In general, it is found that the surface roughness plays a very important role in the overall erosion behavior. The penetration depth gradually diminishes as the surface roughness increases. Also, interparticle collisions were found to have relevant effects even at low particle concentrations.

\* Financial support from Petrobras (0050.0072928.12.9) is gratefully acknowledged.

## LIQUID FILM FLOW WITH SOLUBLE SURFACTANTS: THEORY AND EXPERIMENT

Anna Georgantaki, University of Thessaly, Greece; George Karapetsas, University of Thessaly, Greece; Michalis Vlachogiannis, Technological Educational Institute of Larissa, Greece; Vasileios Bontozoglou, University of Thessaly, Greece

The linear stability of liquid film flow laden with soluble surfactants is considered theoretically and experimentally. Equations of flow are supplemented by mass balances for surfactant concentrations at the interface and in the bulk and by a Langmuir model for interface adsorption-desorption kinetics. The resulting eigenvalue problem is solved analytically in the limit of long-wave disturbances and numerically for disturbances of arbitrary wavelength. The long-wave instability is found to depend only on surfactant solubility and interface concentration. Shorter waves are more stable but their fate also depends on sorption kinetics. With increasing surfactant solubility, maximum stabilization is predicted to shift gradually from close packing to lower interface concentrations of surfactant. Experiments to test the above predictions are performed with water laden with SDS in a 3 m long inclined channel, using conductance probes to record spatio-temporal evolution of inlet disturbances. Strong stabilization is documented independent of channel inclination, confirming the dominance of Marangoni over gravity forces. Traveling waves are near-sinusoidal, in agreement with the predicted strong damping of superharmonics. The maximum stable Reynolds number ( $Re \sim 130$ ) occurs with SDS concentration around 10% of CMC (critical for onset of micelle formation). Quantitative comparison between theory and experiment reveals fundamental weaknesses of the Langmuir model, but is met with reasonable success if the model parameters are fitted empirically to low-concentration capillary data.

## FINITE VOLUME / FINITE AREA INTERFACE-TRACKING METHOD FOR SOLUBLE SURFACTANTS

Holger Marschall, Technische Universität Darmstadt, Germany; Chiara Pesci, Technische Universität Darmstadt, Germany; Paul S. Weber, Technische Universität Darmstadt, Germany; Dieter Bothe, Technische Universität Darmstadt, Germany

We have developed a comprehensive framework for numerical modeling of interfacial transport of soluble and insoluble surfactants (doi:10.1016/j.compfluid.2014.12.017), which extends the Arbitrary Lagrangian-Eulerian (ALE) Interface-Tracking method by Tuković and Jasak (doi:10.1016/j.compfluid.2011.11.003) to cope with interfacial multi-physics at fluid interfaces. This contribution is concerned with transport of surfactant mixtures on the interface and in the bulk, which is coupled via sorption processes resulting in a multi-equation/multi-region coupling problem. In particular, this talk will focus on numerical modeling of the two limiting sorption regimes, namely diffusion-controlled (fast) and kinetically-controlled (slow) adsorption, which necessitate substantially different numerical treatments. To account for both cross-diffusion and non-idealities of surfactant mixtures, diffusive fluxes are modeled by means of the interfacial Maxwell-Stefan equations. From the inversion of the Maxwell-Stefan system, spatially heterogeneous effective diffusivities arise and can lead to numerical instabilities. We propose an enhanced discretisation procedure which is straight forward to implement for the finite area diffusion operator. The resulting system of interfacial transport equations is solved simultaneously to account for its coupled nature. Verification and validation results are provided for a selection of test cases. Application relevance is exemplified for freely rising bubbles in quiescent liquid and Taylor bubbles rising in a squared channel under the influence of surfactants.

## A HYBRID NUMERICAL METHOD FOR TWO-PHASE FLOW WITH SOLUBLE SURFACTANT

Michael Booty, New Jersey Institute of Technology, USA; Michael Siegel, New Jersey Institute of Technology, USA

A hybrid method is described that combines an asymptotic reduction for the evolution and transport of dissolved bulk surfactant in the limit of large bulk Peclet number with a surface-based numerical method for the dynamics of a bubble or drop with adsorbed surfactant when it is deformed by an imposed flow. Here, the boundary integral method is adopted as the surface-based method in the zero Reynolds number limit. Computations that use the hybrid method to simulate various examples and features of bubble and drop dynamics with soluble surfactant are described. These include: a validation of the method in 2D for an inviscid bubble in an imposed strain or shear flow; the influence of parameters such as a Biot number and an equilibrium partition coefficient that describe the bulk-interface surfactant exchange dynamics in an axisymmetric, uniaxial extension; conditions for drop tip streaming, and the influence of flow focusing on tip streaming. The accuracy and efficiency of the method are also considered.

\* Support from grant DMS-1412789 from the National Science Foundation is gratefully acknowledged.

## ON THE INFLUENCE OF SURFACTANTS ON THE WAKE AND SHAPE OF GASEOUS BUBBLES ASCENDING IN LIQUIDS

Felix Schraner, Technische Universität München, Germany; Johannes Heinz, Technische Universität München, Germany; Nikolaus Adams, Technische Universität München, Germany

Industrial and natural two-phase flows of immiscible fluids with large density and viscosity ratios, such as ascending bubble flows, are impure. The shape and wake flow of ascending bubbles are a consequence of the interaction of gas and liquid at and with their common interface. At the interface, surfactants accumulate and affect its dynamic behavior. The dependence on the concentration and properties of the surfactant is, however, hard to study experimentally as well as numerically due to the lack of appropriate methods. We have developed a level-set based Eulerian sharp interface method for the numerical simulation of viscous, incompressible, immiscible two-phase flows. It accounts for viscous, capillary and Marangoni stresses directly at the interface and is fully conservative. The two latter depend on the local surfactant concentration, which is also modeled conservatively in Eulerian space. In this work, we employ this state of the art method to study the influence of surfactant parameters on the wake and shape of gaseous bubbles rising in liquid media. Besides investigating the bubble dynamics itself, we illuminate the dynamic redistribution of surfactant. As a consequence of our investigations, we give first guidelines for improving the controllability of bubble shape and wake by surfactant parameters.

\* We acknowledge the Deutsche Forschungsgemeinschaft (DFG) for funding. F.S.S. is member of the TUM Graduate School.

## A NEW SUBMODEL FOR MOVING PARTICLES UNDERGOING PHASE CHANGE SOLID-LIQUID-GAS

Hemant Bansal, University of Alberta, Canada; Petr Nikrityuk, University of Alberta, Canada

This work is devoted to the development and validation of a subgrid model describing heat and mass transfer between a hot ambient gas and moving particles undergoing phase change from solid to gas under the influence of convection. Such kind of submodels plays the role of 'scale bridges' between microscale (e.g. interfacial heat transfer) and macroscale (e.g. bulk flow) phenomena in different applications in chemical and materials science. Applied to multiscale modeling of particulate flows with phase change phenomena, our model serves as a coupling between equations describing particle movement in Lagrangian space and mass and heat conservation equations defining gas and liquid phases in Eulerian space. The input parameters in our model are the particulate Reynolds number ( $Re$ ), the Grashof number ( $Gr$ ), the Stefan numbers ( $Ste$ ) for liquid and gas phases and the Prandtl numbers ( $Pr$ ). The model takes into account the solid phase melting/solidification and liquid phase evaporation/condensation. The model has been validated against experimental data published in the literature applied to the melting of ice cylinder in the atmospheric air under different flow conditions. Good agreement between our model predictions and published experimental data is observed.

\* Support from NSERC-Canada is gratefully acknowledged.

## DIRECT NUMERICAL SIMULATION (DNS) OF FLUID-SOLID MASS TRANSFER USING A GHOST-CELL BASED IMMERSED BOUNDARY METHOD

Jiangtao Lu, *Multi-scale Modelling of Multi-phase Flows, The Netherlands*;  
E.A.J.F. Peters, *Multi-scale Modelling of Multi-phase Flows, The Netherlands*; J.  
A. M (hans) Kuipers, *Eindhoven University of Technology, The Netherlands*

Among various mass transfer simulation strategies, DNS models are a powerful tool to obtain improved correlations for interface transfer which could be applied in coarser scale models. In this paper, an efficient ghost-cell based immersed boundary method is introduced to simulate mass transfer problems in particulate flows. The boundary condition is enforced at the exact particle surface and incorporated into the discrete momentum and species equations directly. Both directional quadratic and quadratic interpolation schemes are applied in the reconstruction procedures, and thus both Dirichlet and Neumann boundary conditions could be realized.

The present simulations are performed for different fluid-solid systems. Starting with the case of forced convective mass transfer around a stationary sphere, the simulation results are verified to agree with well-known empirical expressions for the Sherwood number. Following that, we apply our method to study a three-bead 'reactor' and dense stationary sphere arrays. In the former case, the influence of Reynolds number and sphere-sphere distance are investigated; while for the latter case, we gradually switch from a statistically homogeneous periodic array to a heterogeneous particle distribution to assess the impact of clustering of particles on mass fluxes.

\* Support from Netherlands Center for Multiscale Catalytic Energy Conversion (MCEC) is gratefully acknowledged.

## DETAILED NUMERICAL SIMULATIONS OF RESOLVED MULTI-COMPONENT FUEL DROPLETS EVAPORATING IN HOMOGENEOUS ISOTROPIC TURBULENCE

John Palmore, *Cornell University, USA*; Olivier Desjardins, *Cornell University, USA*

We demonstrate detailed numerical simulations of a multi-component jet fuel surrogate evaporating into a turbulent gas flow. This work studies how evaporation interacts with turbulence and, ultimately, how the individual fuel components, each having different properties, are redistributed by the turbulent flow.

Simulations are performed using the NGA code which solves the Navier-Stokes equations for multi-component, liquid-gas flows under a low-Mach number assumption. We explicitly track the liquid-gas interface location using a conservative level set method, which allows us to enforce the appropriate boundary and jump conditions on the interface. A novel approach is used to represent the discontinuous density field, so traditional conservative transport schemes for reacting flows may be used in conjunction with classical two-phase flow techniques.

We present results for evaporating droplets in homogeneous isotropic turbulence. The effects of evaporation on turbulence are noted by comparing simulations of evaporating and non-evaporating droplets. Finally, we compare simulations of multi-component and mono-component fuel surrogates for their fuel vapor distributions. Results are discussed with their implications to combustion in mind.

## DROP IMPACT ON SOLIDS

Christophe Josserand, *Institut D'Alembert, France*

Drop impact on solids is important in many situations ranging from ink-jet printing to combustion and erosion by rain. The impact dynamics depends on many parameters, primarily on the liquid viscosity and gas-liquid surface tension, quantified by the Reynolds and Weber numbers respectively. Ten years ago, it has been shown that the surrounding gas pressure could control the formation of a splash or the spreading during the impact. Interestingly, this effect cannot be understood within a simple model involving simply the cushioning of the gas beneath the drop prior to impact. I will discuss in this talk the different mechanism that could be invoked to explain the gas influence in the impact dynamics. In particular, I will discuss using simple scaling laws the non-continuum gas effect, the inertial scenario and the role of the compressibility of the gas.

## DIRECT NUMERICAL SIMULATION OF HEAT TRANSFER IN DENSE PARTICLE-LIQUID TWO-PHASE MEDIA

Jingchen Gu, *Osaka University, Japan*; Katsuya Kondo, *Osaka University, Japan*; Shintaro Takeuchi, *Osaka University, Japan*; Takeo Kajishima, *Osaka University, Japan*

To study the effects of heat conductivity within a solid particle and particle motion on the heat transfer, a direct numerical simulation of liquid-solid two-phase flow is conducted. To deal with interaction between the fluid and particles, our original immersed solid approach on a rectangular grid system is improved. The local heat flux at fluid-solid interface is discretized considering the direction of the solid surface, and it is incorporated into an implicit scheme of temperature. Then, we develop an efficient simulation method to track the neutrally buoyant particles having wide range of heat conductivity ratios to the fluid. Our numerical results show the importance of directly resolving the temperature distributions not only in the liquid but also in the particles. When the heat conductivity ratio is greater than unity, the inhibition of heat transfer in the whole domain is observed because convection heat transfer is restrained with masses of particles. When the heat conductivity ratio is equal to unity, a region of negative temperature gradient is found near wall because of the presence of the particles.

## OBLIQUE IMPACT OF DROPLETS ON FLOWING LIQUID FILMS

Zhizhao Che, *Imperial College London, UK*; Renad Ismail, *Imperial College London, UK*; Idris Adebayo, *Imperial College London, UK*; Omar K. Matar, *Imperial College London, UK*

The oblique impact of droplets on liquid films flowing on an inclined substrate is studied experimentally. Different outcomes are identified and analysed, such as bouncing, partial coalescence, total coalescence, and splashing. The flow rate of the liquid film, the inclination angle of the substrate, the speed and the size of the droplet are controlled and their effects on the impact process are studied. The experiments showed that the probability of bouncing increases with the Reynolds number of the liquid film. A lubrication model is proposed to study the drainage of the gas layer between the droplet and the liquid film, and the analyses show that the tangential movement of the liquid film can delay the drainage of the gas layer and promote the bouncing of droplets, which confirms the experimental observation. In addition, due to the flow in the liquid film and the oblique impact direction, the splashing process is asymmetric. The propagation of the asymmetric crown and the generation of secondary droplets on the rim of the crown are analysed through image processing, and the results show that the flow in the liquid films significantly affects the splashing process.

## PHENOMENA OF DROP IMPACT ON HOT SURFACES

Jan Breitenbach, *TU Darmstadt, Germany*; Ilia V. Roisman, *Technische University Darmstadt, Germany*; Cameron Tropea, *Technische University Darmstadt, Germany*

Outcomes from an isothermal drop impact without phase change (rebound, deposition or splash) are determined by the impact parameters, liquid material properties, substrate morphology and wettability. Non-isothermal drop impacts with phase change can be accompanied by the Marangoni effects, nucleate or film boiling, evaporation and other thermodynamic effects. By varying the surface temperature, the phenomenon of drop impact can change significantly and additional thermodynamic influences can be observed, e.g. the Leidenfrost effect.

The main focus of this experimental study is the characterization of the hydrodynamic outcome of a drop impact on a hot surface. We have performed experiments with pure water drops, at different impact parameters, surface temperatures and surface roughness. The observed hydrodynamic phenomena led to a classification into various impact regimes: deposition, partial rebound, rebound and atomization. The regime maps are presented quantitatively and corresponding models have been proposed; conditions for atomization were identified. The influence of surface roughness and morphology has also been investigated and quantified.

\* The authors gratefully acknowledge financial support from the Deutsche Forschungsgemeinschaft (DFG) in the framework of SFB-TRR75 (C4).



## SPLASH OF A LIQUID DROP IMPACTING ONTO A WALL FILM OF DIFFERENT LIQUID

Hannah Kittel, Technische Universität Darmstadt, Germany; Ilia V. Roisman, Technische Universität Darmstadt, Germany; Cameron Tropea, Technische Universität Darmstadt, Germany

Splashing as a result of drop impact onto a wetted substrate has been studied rather extensively due to its importance in many industrial applications. It is well-known that the splashing threshold of a drop impacting onto a wetted wall is determined by the inertia, viscous and capillary forces. However, the case when the liquids of drop and wall film have different properties has seldom been studied and is definitely not fully understood. The main objective of this experimental work is to study the impact of a single drop of Newtonian liquid onto a liquid wall film of different viscosity. The experimental setup consists of a drop-on-demand generator, wetted, horizontal, glass impactor, and an observation system, consisting of the illumination source and a high-speed video camera. The impact parameters (drop diameter, impact velocity), the relative film thickness and the viscosities of the drop and wall film are varied in the experiments. The drop and wall liquids in the experiment are immiscible. The splashing threshold is determined for various combinations of the liquid viscosities. Finally, the critical value of the dimensionless number  $K = \sqrt{Re} We$  is determined as a function of the relative film thickness, scaled by the initial drop diameter and by the viscosity ratio of the two liquids.

\* The authors gratefully acknowledge financial support from the Deutsche Forschungsgemeinschaft (DFG) in the framework of SFB-TRR150(A02).

## TURBULENCE MODULATION IN CONCENTRATED, HIGH REYNOLDS NUMBER SOLID-LIQUID FLOWS

David Breakey, University of Alberta, Canada; Yogesh Agrawal, University of Alberta, Canada; Parsa Amini, University of Alberta, Canada; Sina Ghaemi, University of Alberta, Canada; David Nobes, University of Alberta, Canada; Sean Sanders, University of Alberta, Canada

Most experimental investigations of turbulence modulation in particle-laden flows are limited to low solids concentrations ( $< 1\%$  by volume) and relatively low Reynolds numbers ( $Re \leq 30\,000$ ). Numerical modeling, which provides a much wider range of turbulent statistics, is currently limited to even lower  $Re$ . The issue is that most industrial solid-liquid flows operate at high concentrations and high  $Re$ . The present study investigates the effect of large particles on the liquid phase turbulence at high concentrations ( $> 5\%$  by volume) and  $Re \geq 105$  using refractive index-matched mixtures of an aqueous potassium thiocyanate (KSCN) solution and monosized ( $d = 3\text{ mm}$ ) borosilicate glass beads. Tests were conducted using a 50.6 mm (diameter) by 7 m (length) vertical flow loop. A combination of PIV and PTV techniques was used to obtain the streamwise and radial fluctuating velocities for the pure (unladen) liquid, the liquid containing particles and the particles themselves. The presence of the particles had surprisingly little effect on the carrier fluid turbulence at lower solids concentrations but a much more significant effect as the concentration increased. In all cases the particle turbulent fluctuations were greater than those of the liquid phase. The effects of particle-particle and particle-wall collisions are shown to be responsible. In addition, a relationship between the frequency of particle collisions and the change in fluid turbulence characteristics will be discussed.

## ROLE OF PREFERENTIAL CONCENTRATION IN THE DESTABILIZATION OF SHEARED PARTICLE-LADEN FLOWS

Mohamed Housseem Kasbaoui, Cornell University, USA; Donald Koch, Cornell University, USA; Olivier Desjardins, Cornell University, USA

In particle-laden flows, preferential concentration is the tendency of inertial particles to leave vortical regions and gather along straining regions. A recent theoretical study found preferential concentration to cause the initial destabilization of homogeneously sheared suspensions in the presence of gravity on a time scale that is the inverse of the shear rate. On a longer time scale, a secondary Rayleigh-Taylor (RT) instability can further amplify the perturbations, both in the fluid and particle phases, resulting in enhanced clustering. These phenomena are investigated by means of numerical simulations. Randomly distributed particles are placed in homogeneous shear and a wave-like perturbation is introduced. We observe the preferential concentration instability unfolding, which later on bootstraps the RT instability, confirming the theoretical findings. The growth rates are characterized and compared to theory. When the suspension's mass loading is of order unity, often the case for gas-solid flows, the non-zero slip velocity between phases might lead to sustained clustering that can strongly modify the flow.

## PARTICLE CONCENTRATIONS IN RIB-ROUGHENED WALL TURBULENCE

Lihao Zhao, Norwegian University of Science and Technology, Norway; Helge Andersson, Norwegian University of Science and Technology, Norway

The current work aims to study the flow of a suspension of solid spherical particles in a turbulent channel with rough walls. The Eulerian flow field in a roughened channel at a frictional Reynolds number 395 was fully resolved by Direct Numerical Simulations. 'k-type' roughness was considered by introducing transverse square ribs on both walls. The roughness height was 13.6 in wall units and the pitch-to-height ratio was 8. A Lagrangian approach was adopted for particle tracking in the flow field. We focused on particle inertia effects on the particle concentration and performed simulations with four sets of particles with Stokes numbers ranging from 1 to 100, each case with 105 particles. Statistics of the fluid flow field were validated by comparing with published data and showed significant differences compared with a smooth-wall channel flow inside the roughness layer (within 3-5 roughness heights). The particle statistics revealed that the particle spatial distribution was strongly modulated due to the presence of the wall-mounted ribs compared with the case with smooth walls. Particle inertia was found to be of importance on the preferential clustering of the particles in the roughness layer and, in particular, in the cavities between two subsequent roughness elements.

## SIMULATION OF TURBULENT FLOW AND PARTICLE TRANSPORT AND DEPOSITION USING LATTICE BOLTZMANN METHOD AND RANS MODEL

Hasan Sajjadi, Clarkson University, USA; Maziar Salmanzadeh, Shahid Bahaonar University, Iran; Goodarz Ahmadi, Clarkson University, USA; Saeed Jafari, Shahid Bahaonar University, Iran

In this study the Lattice-Boltzmann computational approach in conjunction with RANS model was used and turbulent channel flow and particle transport and deposition in the channel were studied. The new Lattice Boltzmann Method (LBM) solved the RANS equations coupled with the  $k-\epsilon$  turbulence models. In particular, the LBM formulation was augmented by the addition of two transport equations for the probability distribution function of populations of  $k$  and  $\epsilon$ . The discrete random walk (DRW) model was used to generate the instantaneous turbulent fluctuations. The analytical fits to the root mean-square velocity fluctuations obtained by the direct numerical simulation (DNS) of turbulent flow were used in the analysis. Particular attention was given to the proper evaluation of the wall normal turbulent velocity fluctuations. The simulation results were compared with the available numerical simulation and experimental data. It was shown that the new LBM-RANS provided reasonably accurate description of turbulent flows and particle transport and deposition at modest computational cost.

## A SLUG CAPTURING METHOD APPLIED TO UNCONVENTIONAL SCENARIOS: HIGH VISCOUS OILS AND ACTUAL PIPELINE GEOMETRIES

Marco Ferrari, Università degli Studi di Brescia, Italy; Arianna Bonzanini, Università degli Studi di Brescia, Italy; Pietro Poesio, Università degli Studi di Brescia, Italy

One-dimensional, hyperbolic, transient five equations two-fluid model can predict the formation, growth, and subsequent development of slugs in horizontal and near-horizontal flow automatically and they can even be used to predict the flow pattern, which is now a result and not an input parameter. Statistical characteristics (slug velocity, length, and frequency) are numerically predicted and they show good agreement with experimental data. However, the capabilities of this approach have been tested only for water-air flows in a straight horizontal pipe.

In this paper, we move the focus on the application of the code to unconventional problems that involve high viscous oils and more complex pipe geometries. The paper is divided into two parts: in the first part of the paper, we test the possibility of slug capturing approach to describe and predict the relevant features of air/high viscosity oils. In that frame, we will check the effect of different closure relations on the results. Comparisons between slug velocity, frequency, and length distribution are discussed. In the second part of the paper, we move from simple geometries toward more complex conditions that may be representative of actual application cases. Despite only few experimental conditions being available, the comparison shows results in reasonable agreement.

# A MECHANISTIC APPROACH FOR HORIZONTAL SLUG FLOW FROM TWO-PHASE (GAS-LIQUID) TO THREE-PHASE WITH THE FORMATION OF GAS HYDRATES

Carlos Bassani, Federal University of Technology - Paraná, UTFPR, Brazil; Fausto Barbuto, Federal University of Technology - Paraná, UTFPR, Brazil; Amadeu Sum, Colorado School of Mines - CSM, USA; Rigoberto Morales, Federal University of Technology - UTFPR, Brazil

Pipe blockage due to gas hydrate formation is a main concern in the oil and gas industry due to the costs of production impairment or interruption. Assuming a hydrate formation rate based on the subcooling of the system and on the gas-water interfacial area, the present work models the effects of the hydrate on the hydrodynamics and heat transfer of slug flow. The hydrate phase is assumed as being homogeneously dispersed in the water. The gas and water consumption become source terms in the mass balances equations, whereas the volume contraction of the mixture due to the gas consumption is modelled as a pressure drop term. Finally, the exothermic characteristic of the hydrate formation is taken into account in the energy conservation equation. The model follows a quasi-stationary approach, assuming a known unit cell translational velocity and transforming the time-rates into spatial variations. The simulator is capable of predicting the hydrodynamics and heat transfer of the three-phase (gas-water-hydrate) slug flow. Some results are presented so that the differences between the slug flow temperature and pressure distributions with and without hydrates can be understood.

# LINEAR STABILITY ANALYSES OF DISCRETE REPRESENTATIONS OF THE TWO-FLUID MODEL FOR STRATIFIED PIPE FLOW

Andreas Akselsen, Norwegian University of Science and Technology, Norway; Ole Jørgen Nydal, Norwegian University of Science and Technology, Norway

Many dynamic pipe flow simulator tools are capable of accurately predicting the onset of hydrodynamic flow instability and regime transition. The quality and reliability of such predictions are however strongly dependent upon the numerics of these simulator tools, the scheme type and resolution in particular. Many modern simulation tools to date still rely on low-precision schemes like the upwind and Lax-Friedrich schemes for capturing flow instabilities on discrete meshes.

Analytical linear stability analyses are performed on the discretised model representations and presented in the present work. These analyses provide algebraic expressions which give direct information on i) when a studied scheme will predict linear wave growth, ii) the rate of growth and the expected growing wavelength, and iii) the influence of the time integration. The analyses also provide insight into the mechanism of hydrodynamic wave instability and into the differences between viscous and inviscid flows in this respect. For example, it is found that numerical viscosity can promote premature instability in otherwise low-viscosity flows. Results are compared with actual simulations. These analyses are of interest in understanding the predictions made by simulator tools and in choosing the appropriate grid resolution with respect to prediction reliability and simulation efficiency.

# NUMERICAL STUDY OF TAYLOR BUBBLES RISING THROUGH SUDDEN EXPANSIONS AND CONTRACTIONS

Chengsi Zhou, INSA de Lyon, France; Ronnie Knikker, INSA de Lyon, France; Remi Revellin, INSA de Lyon, France

The numerical investigation of the rising individual Taylor bubble through a vertical sudden expansion and contraction in stagnant Newtonian liquid is presented. The CFD procedure is based on the open source package Gerris which adopts the volume-of-fluid (VOF) method to represent the gas/liquid phase. The numerical method is verified using the validated results of single Taylor bubbles rising in straight columns. Our investigation focuses on the transient process of the bubbles passing the singularity. The variations of the bubble velocity, liquid film and pressure drop are investigated. The results show that the expansion with greater expansion ratio yields more perturbations on the bubbles. The expansion has strong effects on the tail of the bubble. The unstable bubble tails break into smaller bubbles in some of the test cases. We also found that the bubble shape variations depend on the length of the bubbles. For a bubble passing through a contraction, the clog phenomenon has been observed. This study is performed over a broad range of Eötvös number and expansion/contraction ratio, which provides new insights to fully understand this phenomenon.

# AN EXPERIMENTAL INVESTIGATION ON THE EVOLUTION OF HIGHLY VISCOUS OIL SLUG LENGTHS IN HORIZONTAL PIPES

Gianluca Losi, Università di Brescia, Italy; Pietro Poesio, Università degli Studi di Brescia, Italy

Slug flow is an unfair intermittent regime that often occurs in gas-liquid pipe flow and it is exalted by the viscosity of the liquid phase. A relevant number of investigations have been carried out to quantify the average stable length of a liquid slug. However, in some practical application, be apart of more informations about slug length distributions and their evolution along the pipe is fundamental and, for this reason, slug length evolution for highly viscous oil/air slug flow is experimentally investigated.

Six capacitance probes are placed along a 9 m – 22 mm I.D. pipe and they are used to monitor the slug flow structures evolution. An algorithm is developed to track and to measure slug velocities and lengths. A statistically relevant number of slugs is observed during the experiments and the probability density functions of their lengths are built.

A literature slug tracking model has been implemented, which calculates the increase or decrease in length for each individual slug from a random inlet distribution, managing also events like shorter slugs disappearance.

The PDFs, built from experiments, are compared with those calculated by the model and the results, both in terms of mean slug length and shapes of the distributions are quite good over a range of flow rates.

# EXPERIMENTAL INVESTIGATION OF THE FLOW STRUCTURE AROUND TAYLOR BUBBLES IN VERTICAL DUCTS IN THE PRESENCE OF DISPERSED BUBBLES

Emilio Paladino, SINMEC / Federal University of Santa Catarina, Brazil; Rafael Cerqueira, SINMEC / Federal University of Santa Catarina, Brazil; Ronei Brito, SINMEC / Federal University of Santa Catarina, Brazil; Clovis Maliska, SINMEC / Federal University of Santa Catarina, Brazil; Cristiano Meneghini, SINMEC / Federal University of Santa Catarina, Brazil

The flow structure around Taylor bubbles rising in vertical ducts, through stagnant and co-current air-water flow, with and without the presence of dispersed bubbles in the liquid stream, was studied in this work. An experimental apparatus was specially designed to allow control of liquid and dispersed bubbles superficial velocities and the length of the Taylor bubbles injected into the stream. It is observed that, for large values of the inverse viscosity number, a large number of instantaneous fields is needed to obtain a consistent ensemble averaged velocity fields around Taylor bubbles. Thus, PIV technique was combined with a laser diode triggering system to obtain PIV images with the Taylor bubbles in the same positions and calculate the averaged flow fields around them. Images with high speed camera were also obtained to visualize the interactions of small bubbles with Taylor bubbles. With this instrumentation, the flow structure around Taylor bubbles in the presence of small dispersed bubbles in the liquid stream was studied, which is a more realistic representation of slug flow. Results show that the presence of dispersed bubbles in the liquid stream, even in a low volumetric fraction, substantially modifies the flow field around Taylor bubbles, increasing their terminal velocity and changing flow structure, mainly in the wake region, which would affect thermal and mass transfer processes within the liquid slug region.

# TAYLOR BUBBLE DESTRUCTION AT SUDDEN EXPANSIONS

William Carter, University of Nottingham, UK; Ezekiel Agunleji, University of Nottingham, UK; Barry James Azzopardi, University of Nottingham, UK

Azzopardi et al. (2014) have shown that the frequency of large bubbles, usually called Taylor bubbles, encountered in slug flow persist when passing through pipe diameter (contractions) or orientation (pipeline, bend, riser combinations). In contrast, the more limited data for sudden expansions shows that the Taylor bubbles can be destroyed under these conditions. This process has implications for what occur in volcanoes. Large bubbles are known to rise through filled columns of the very viscous magma. They will be affected by any enlargement in the conduit diameter. In particular, where there is a lava lake above the vertical conduit as occurs in Mount Erebus, Antarctica. Here we present the results of Taylor bubbles injected into a narrow pipe protruding into a larger vessel. The effects of the inlet pipe and the receiving vessel have been studied using air and water. The size of daughter bubbles produced as well as the rise velocity of the original/produced bubbles and reductions in the diameter of the neck between new bubble and the residual Taylor bubble have been quantified. The results are interpreted considering the balance between buoyancy and surface tension forces.

Azzopardi et al. (2014) Int. J. Multiphase Flow 67, 22-31

# INTERACTION OF A BUBBLE RISING IN A THIN-GAP CELL WITH THE VORTICES RELEASED BY A COMPANION BUBBLE

Patricia Ern, INP Toulouse, France; Audrey Filella, INP Toulouse, France; Veronique Roig, INP Toulouse, France

We investigate experimentally the hydrodynamical interaction at high Reynolds numbers between two bubbles rising in a liquid otherwise at rest and confined in a thin gap cell. When rising alone, the bubbles considered display in the plane of the cell an oscillatory motion associated to an unsteady wake and a periodic release of vortices. Experiments consisted in the measurement of the bubbles motion and of the velocity field induced in the liquid phase by high-frequency Particle Image Velocimetry. Depending on the relative size of the bubbles and their relative position, different mechanisms of interaction between a trailing bubble and the vortices released by a leading bubble are identified and characterized. These mechanisms are complementary to the strong vertical entrainment observed for bubbles rising in line, discussed in a companion presentation. We show here that the vortices of the wake of the leading bubble may induce significant transverse motions experienced by the trailing bubble, corresponding to an attraction behaviour towards the center of the wake or to an ejection behaviour out of the main ascending flow associated to the wake. In turn, the motion of the second bubble may also strongly impact the vortices present in the flow, resulting in destruction or strengthening of the vortices. We will also discuss a situation where no significant effect of the interaction is observed on the path of the two bubbles, while a significant interaction occurs between the liquid flows induced by the bubbles.

# NUMERICAL SIMULATION OF A SINGLE RISING BUBBLE BY A VOF METHOD WITH ENHANCED CURVATURE ESTIMATION BASED ON POINT-CLOUD SAMPLING

Bruno Kassar, PUC-Rio, Brazil; Joao Carneiro, Instituto Sintef do Brasil, Brazil; Angela Nieckele, PUC-Rio, Brazil

Volume of Fluid is a widely employed method for multiphase flows simulations. Among its advantages are good mass preservation characteristics. For surface tension dominated flows, however, previous works in the literature have shown that accuracy in curvature estimations may still be an issue. Different approaches have been proposed before to mitigate this problem including height-functions, volume fraction smoothing by convolving kernels, parabolic fittings, among others. This work proposes a novel approach for curvature estimation, based on sampling the interface with a cloud of points and normals at the 0.5 level isosurface of the volume fraction field and computes the curvature for each point of the cloud by normal fitting. The new method is implemented into the standard OpenFOAM® VOF solver and significant improvements on the pressure jump estimations and in spurious currents reduction were observed. Bubble rising simulations were also performed and results compared to benchmark data.

\* Support from CAPES and CNPq are gratefully acknowledged.

# DIRECT NUMERICAL SIMULATION OF A TURBULENT PIPE FLOW WITH DEFORMABLE BUBBLES TO INVESTIGATE THE DYNAMICS OF BUBBLES

Kiyoung Kim, Seoul National University, Republic of Korea; Jungwoo Kim, Seoul National University of Science & Technology, Republic of Korea; Haechon Choi, Seoul National University, Republic of Korea

Bubbly flows are readily observable in a wide range of the industrial area and our daily life. However, their characteristics such as the turbulence statistics or mixing performance are still poorly understood, especially for deformable bubbles. The main obstacle lies in the difficulty of experimental measurements which contain fundamental uncertainty due to two different phases. In our study, a direct numerical simulation of a turbulent bubbly flow is conducted to examine the characteristics of the bubbly flow in a pipe. The pipe geometry is constructed using an immersed boundary method in Cartesian coordinates and the phase interface is tracked by the level-set method, with which bubbles located at any places in a pipe can be adequately resolved. The 3.3mm air bubbles go upward in a 40mm diameter pipe filled with water. The void fraction is 0.8% and the Reynolds number is 5300 based on the bulk velocity and the pipe diameter. We calculate the turbulence statistics and compare them with those of experiment at a similar condition, showing a reasonable agreement. The core region is a well-mixed region and shows almost constant velocity. However, bubbles near the wall drag water in a vertical direction by the buoyancy, which makes large velocity gradient there. Most of bubbles move near the wall, but they often travel to the center region. We investigate the dynamics of air bubbles by tracking their position, velocity and aspect ratio, and the results will be given at the presentation.

# NUMERICAL SIMULATION OF UNDERWATER EXPLOSION WITH GHOST FLUID METHOD

Hong Chen, Harbin Engineering University, China; Weibing Zhu, Harbin Engineering University, China; Boo Cheong Khoo, National University of Singapore, Singapore

The present study is focused on the numerical simulation of complex processes underwater explosion near the free surface, its interaction with interface and the resulting unsteady cavitation evolution. The compressibility effects of liquid water are taken into account and the cavitation is governed by cavitation model which is based on the compressible Euler equations. The breakup of the interface and collapse, jet, ring rebound of bubble and spike of free surface were successfully calculated, and also we got the unsteady features of cavitating flow due to the rarefaction waves propagating through the liquid. The numerical method which was solved for the compressible multi-medium flows is based on Ghost Fluid Method, with Level set method tracking the interface, and the Harten-Lax-van Leer contact (HLLC) scheme was used for the approximate Riemann solvers. High-order accuracy is achieved using a weighted average flux (WAF) scheme. Computations of underwater explosion near the free surface have demonstrated robustness of the proposed algorithm.

# EFFECT OF BUBBLE CLUSTERS ON TURBULENT FLOW STRUCTURES IN UPWARD BUBBLY FLOW

Wenhao Zhang, The University of Tokyo, Japan; Yoshito Sekiguchi, The University of Tokyo, Japan; Hiroaki Nakanishi, The University of Tokyo, Japan; Jun Sakakibara, Meiji University, Japan; Shu Takagi, University of Tokyo, Japan

We have conducted the measurement of upward, turbulent bubbly channel flows. In water gas vertical bubbly flow, mono-dispersed 1 mm spherical bubbles do not coalesce due to surfactants in liquid phase. Furthermore, in some specific condition, bubbles migrate toward the walls and form bubble clusters at the vicinity of walls, which causes the significant reduction of Reynolds stress near the walls [Takagi, S. and Matsumoto, Y., Annu. Rev. (2011)]. This result suggests that the turbulent structure change dramatically due to bubble clusters. For further investigation of the turbulent structure, we constructed a measuring system with high spacial and time resolution called Scanning Stereoscopic Particle Image Velocimetry (SSPIV), and the three-dimensional of three-component velocity field (30 mm × 40 mm × 20 mm) is measured as the test section. Based on the data, by calculating the velocity gradient tensor, the vortices can be visualized using the definition based on local swirling strength and the pressure minimum characteristic [Nakayama, K. et.al, Fluid Dyn. Res. (2014)]. And the large-scale vortices that dominate the turbulent structure are extracted. Moreover, we conducted a new measuring system, which can photograph bubbles from the front side when conducting SSPIV measuring. From the motion of bubbles and the distribution of large-scale vortices, the effect of bubble clusters on turbulent flow structures are analyzed. The results will be discussed in the presentation.

# NUMERICAL SIMULATION OF MICROLAYER FORMATION DURING THE INERTIA-CONTROLLED GROWTH PHASE OF BUBBLES IN NUCLEATE BOILING

Alexandre Guion, Massachusetts Institute of Technology (MIT), USA; Jacopo Buongiorno, Massachusetts Institute of Technology, USA; Stephane Zaleski, Université Pierre et Marie Curie (Paris 6), France; Shahriar Afkhami, New Jersey Institute of Technology (NJIT), USA

Predicting nucleate boiling heat transfer is complicated by the need to consider phenomena occurring over multiple scales, from the adsorbed liquid layer ( $n.m$ ) and the microlayer ( $\mu m$ ) at the wall, up to the bubble diameter (mm). In practice, numerical simulations of boiling resolve the macroscopic liquid-vapor interface of the bubble, but resort to subgrid models to account for micro scale effects, such as the evaporation of the microlayer. Current microlayer subgrid models are based on a steady state analysis of the liquid film, however direct experimental observations of the microlayer suggest that its shape and thickness evolve significantly during the growth of the bubble. Realistic time-dependent microlayer evaporation models necessitate initialization of the microlayer profile underneath the bubble. In this work, the VOF method implemented in the open-source code Gerris, 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 the fluid properties, contact angle and wall superheat. This database will be used to develop a subgrid model or correlation to provide the initial condition for microlayer evaporation during the subsequent thermally-controlled bubble growth phase

## PREDICTION OF THE BUBBLE DEPARTURE IN POOL AND FORCED CONVECTION BOILING CONSIDERING THE SUBLAYER: A SUB MODEL OF CFD APPROACHES

Wei Ding, TUDresden / HZDR, Germany; Eckhard Krepper, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Uwe Hampel, TUDresden, Germany

Time averaged Eulerian multi-phase approaches and heat flux partitioning method is popular to be applied in the computational fluid dynamic simulations of wall boiling. In these CFD simulations, the many sub models like the complex boiling phenomena are particularly important. In this paper, a new mechanistic model for the nucleation boiling and high potentially extended for film boiling will be introduced. The model is based on the force balanced on bubble and on contact angle at a single nucleation site. It considered the evaporation of microlayer under the bubble, thermal diffusion and condensation around the cap of bubble. The model also considered the bubble geometry change at different time period: hemi sphere at inertial, from hemi sphere to sphere in thermal diffusion controlled period and sphere plus bottleneck in the departure period. Validation of the model is done through comparison against experiments in both pool boiling and flow boiling under different conditions. It doesn't require case dependent recalibration in the validation. The simulation results are able to show the dependency of bubble departure diameter (lift off diameter) and frequency on the different physical quantities such as heat flux, liquid material, sub cooling temperature, design of channel (diameter, length), mass flow rate and so on. Based on mutual effect from neighboring nucleate sites, the model is able to covering nucleate boiling, transition boiling and film boiling.

\* Support from BMWi grant 1501473C is gratefully acknowledged.

## NUMERICAL ANALYSIS OF BUBBLE DYNAMICS WITH VAPOR-LIQUID BOUNDARY CONDITIONS DERIVED FROM MOLECULAR GAS DYNAMICS

Shotaro Morikawa, Hokkaido University, Japan; Yoshinori Jinbo, National Taiwan University, Taiwan; Yuki Tatekura, Hokkaido University, Japan; Kazumichi Kobayashi, Hokkaido University, Japan; Masao Watanabe, Hokkaido University, Japan; Hiroyuki Fujii, Hokkaido University, Japan; Koichi Sasaki, Hokkaido University, Japan

The collapse and rebound of a bubble with phase changes (evaporation and condensation) at the vapor-liquid interface are fundamental physical phenomena in the fields of industry and medicine. In this study, we propose a mathematical model of describing the dynamics of a single spherical vapor bubble with phase changes at the vapor-liquid interface under the assumption of uniform pressure inside the bubble. The present mathematical model is constructed of a physically correct set of twenty equations including ordinary differential equation, integral equation, and integro-differential equation from both macroscopic and microscopic viewpoints. To deal with evaporation and condensation phenomena at the interface precisely, we utilize the Sone & Onishi equation for the boundary conditions of the interface derived from the analysis of the Boltzmann equation. In the boundary condition at the interface, there are two parameters, so called the evaporation and condensation coefficients. The calculation results of the mathematical model show that motions of the bubble wall and pressure inside the bubble due to the bubble collapse are greatly affected by the value of evaporation and condensation coefficients.

\* Supports from JSPS KAKENHI 26630043 and 25820038 are gratefully acknowledged.

## MODELING CAVITATION BUBBLES IN A FOCUSED ULTRASOUND FIELD

Kazuki Maeda, California Institute of Technology, USA; Tim Colonius, California Institute of Technology, USA; Wayne Kreider, University of Washington, USA; Adam Maxwell, University of Washington, USA

We model cavitation bubble clouds that are initiated by a pulse of focused ultrasound, with application to a proposed medical procedure called burst wave lithotripsy (BWL). Bubbles are modeled as spherical, radially oscillating cavities dispersed in continuous liquid phase. We solve the evolution of bubble radius and continuous flow field using a WENO-based flow solver on an Eulerian grid. Lagrangian particles represent the bubbles and are coupled with the continuous phase at the sub-grid-scale through volume averaging. We use the solver to simulate the dynamics of cavitation bubble clouds as a function of their size, void fraction, and nuclei distribution. Cavitation and bubble dynamics are initiated by focused ultrasound wave packets of peak amplitude  $O(10)$  MPa and frequency  $O(100)$  kHz that are generated from a model transducer. In the simulation, the reflection and scattering of the incident ultrasound waves due to interactions with the bubble clouds, as well as the evolution of the void fraction of the clouds are successfully captured. Finally, we compare the results with experiments employing high-speed videography and a passive cavitation detector.

\* Support from National Institutes of Health under grant 2P01-DK043881 is gratefully acknowledged

## PRELIMINARY RESULTS OF SOBER-SJ10 ON SINGLE BUBBLE POOL BOILING HEAT TRANSFER IN MICROGRAVITY

Ke Wu, Institute of Mechanics, Chinese Academy of Sciences, China; Jianfu Zhao, Institute of Mechanics, Chinese Academy of Sciences, China; Hui-Xiong Li, Xi'an Jiaotong University, China; Kai Li, Institute of Mechanics, Chinese Academy of Sciences, China

SOBER-SJ10 (Single bubble pOol Boiling ExpeRiment aboard SJ-10) is one of 27 experiments of the program SJ-10, proposed to study local convection and heat transfer around an isolated growing vapor bubble during nucleate pool boiling on a flat plate heater in microgravity. An integrated heater has developed to trigger an embryo bubble with accurate spatial and temporal positioning, to provide an approximate constant input power, and to measure the local temperatures on the surface of heating wall underneath the growing bubble. A series of ground test of the flight module of SOBER-SJ10 have been conducted to verify the function and performance of the developed hardware. The data are in good agreement with others, validating the facility SOBER-SJ10. The space flight experiment will be performed in April, 2016. Preliminary results, particularly on the spatiotemporal distribution of local temperature on the heating surface, as well as boiling curve and bubble behaviors, will be presented. The data in normal and microgravity will also be compared with each other, as well as those reported in the literature.

\* Support from the grants of 11372327 and 11402273 (NSFC) and of XDA04020404 (Strategic Priority Research Program on Space Science/CAS) is gratefully acknowledged.

## TESTING A BUBBLE DYNAMIC MODEL FOR BULK CAVITATION IN UNDERWATER EXPLOSION SIMULATIONS

Rasmus Wedberg, Swedish Defence Research Agency, Sweden; Niklas Alin, Swedish Defence Research Agency, Sweden

Simulations of the effects of underwater explosions commonly employ a simple pressure cut-off model to account for the effects of bulk cavitation. The effects from the cavitation closure are clearly identifiable in experiments and the subsequent reloading induce additional displacements in a structure. These effects thus form a relevant feature to predict. In this work, a physically based model, treating the bulk cavitating fluid as a dispersion of spherical bubbles in compressible water, is tested as a potential improvement of the pressure cut-off approach. Similar to the model of Ando [Ando, Colonius, Brennen, Int. J. Multiphas. Flow 37, pp. 596–608, (2009)], the bubbles evolve according to the Gilmore equation, but here, a multi-material arbitrary Lagrangian-Eulerian approach is used for the flow simulation. A set of previously performed underwater experiments, in which a steel plate is impacted by the shock wave from a detonating PETN charge, are simulated using a two-dimensional axisymmetric setup. The size and closure time of the bulk cavitation zone formed after shock reflection are found to be sensitive to the initial bubble radius and initial volume fraction of vapor and non-condensable gas. These results also differ from the corresponding values obtained using the pressure cut-off. The end results for the net effect onto the steel plate obtained using the bubble model differ slightly from those obtained using the pressure cut-off.

\* Support from The Swedish Armed Forces is gratefully acknowledged.

## CAVITATION CONTROL BY ACTIVE MASS INJECTION THROUGH A SLOT CHANNEL IN THE SURFACE OF A 2D HYDROFOIL

Mikhail Timoshevskiy, Novosibirsk State University, Russian Federation; Konstantin Pervunin, Kutateladze Institute of Thermophysics, SB RAS, Russian Federation; Dmitriy Markovich, Kutateladze Institute of Thermophysics, SB RAS, Russian Federation

We studied experimentally cavitating flow over the suction side of a symmetric 2D foil – a scaled-down model of Francis turbine guide vanes (GV) – in different cavitation regimes at several attack angles. In order to provide the flow control, the GV profile was manufactured with a spanwise slot channel in its surface through which the liquid was supplied along the foil surface at different flow rates. High-speed imaging was used to analyze spatial patterns and time dynamics of the gas-vapor cavities, as well as for evaluating the characteristic integral parameters. A PIV technique was applied to measure the velocity fields and its fluctuations, which were compared with the data measured in the non-cavitating flows. A hydroacoustic pressure transducer was employed to perform local pressure measurements and to analyze frequency characteristics of gas-vapor cavities. Cavitation inception was found to occur at smaller cavitation numbers in case of flow manipulation. For a small attack angle, active mass injection resulted in a significant modification of cavitation regimes, such as vanishing or substantial suppression of periodic oscillations of an attached cavity length at initially unsteady regimes. At a higher incidence, a stable cavity length was reduced depending on the injection flow rate. However, at unsteady regimes no influence of the flow control was registered for the whole range of the injection flow rates.

## A NEW EXPERIMENTAL DATASET TO VALIDATE CFD MODELS OF AIRBORNE NANOPARTICLES AGGLOMERATION

Emmanuel Belut, INRS, France; Theo Christophe, INRS, France

The dynamics of airborne nanoparticles undergoing transport, agglomeration and deposition in a chamber is experimentally studied. We consider the case of a small chamber swept by a constant airflow ( $Re_H=14000$ ) laden with nanoparticles at a controlled concentration and particle size distribution (PSD). A high concentration ( $> 10^{12} \#m^{-3}$ ) is used so as to make agglomeration significant, both Brownian and turbulent agglomeration occurring in the chamber. We then consider the steady state reached by the aerosol in the chamber, and measure the concentration and PSD fields by means of a scanning mobility particle sizer (SMPS), for two types of aerosol particles (NaCl particles -  $GMD=45nm$  - and copper oxide particles -  $GMD=8nm$ ). Air mean velocity and turbulence parameters are also characterized by means of a two-component LDA system. The obtained experimental data aim at validating CFD models of turbulent two-phase flows where low-inertia particles undergo transport, agglomeration and wall-deposition. The obtained dataset is indeed the first to provide such space-resolved PSD measurements of airborne nanoparticles, together with detailed and well-controlled flow data. The influence of particles morphology on coagulation kinetics is highlighted and documented through additional transmission electron microscopy (TEM) observations.

## HYDRATES FORMATION, AGGREGATION AND SLURRY FLOW PROPERTIES IN DIFFERENT MULTIPHASE FLOW PATTERNS

Lin Ding, China University of Petroleum Beijing, China; Bohui Shi, China University of Petroleum Beijing, China; Xiaofang Lv, China University of Petroleum Beijing&Changzhou University, China; Yang Liu, Virginia Tech, USA; Haihao Wu, China University of Petroleum, China; Wei Wang, China University of Petroleum Beijing, China; Jing Gong, China University of Petroleum Beijing, China

Experimental study was performed on a high pressure flow loop to investigate the inter-coupling of hydrates and multiphase flow patterns. Four typical flow patterns were included: stratified, bubble, slug and annular flow. Hydrates formation kinetics and slurry flow properties in each flow pattern were analyzed. Hydrates particles behaviors were recorded by a focused beam reflectance measurement probe and a particle video microscope probe for further analysis of hydrates agglomeration and plug formation mechanism. In stratified flow, after hydrates formation, hydrates aggregation was gentle and the aggregates grew continuously, which consequently led to a final blockage. In bubble flow and slug flow, the hydrates aggregation was quite violent. However, no blockage occurred at these two flow patterns, probably due to the shearing and broken of the aggregates. In annular flow, the induction time for hydrates formation was much longer and the flow would block immediately after the hydrates formation onset, though no hydrates aggregation was observed, suggesting that the blockage in annular flow was probably due to the fast hydrates film growth and deposition on the pipe wall. Therefore, the risk of hydrates blockage in stratified flow and annular flow is much greater than other flow patterns.

Support from grant 51306208 and grant 51274218 are gratefully acknowledged

## PERFORMANCE ASSESSMENT OF EULERIAN VERSUS EULERIAN-LAGRANGIAN LEVEL SET IMPLEMENTATIONS FOR THE SIMULATION OF TWO PHASE FLOWS

Gael Lorieul, Université catholique de Louvain, Belgium; Philippe Chatelain, Université catholique de Louvain, Belgium; Yann Bartosiewicz, Université catholique de Louvain, Belgium

The purpose of this work is to accurately and efficiently simulate sloshing phenomena occurring in the pool of a Liquid Metal nuclear Reactor. The geometry of the interface is a disturbed flat surface, with the possible occurrence of gas entrainment. Two different approaches for two-phase flow simulation are assessed : (i) a standard Eulerian velocity-pressure formulation, and (ii) an Eulerian-Lagrangian vorticity formulation consisting in a Vortex Particle-Mesh (VPM) method. For both methods, interfaces are captured by means of a level-set method, which relies on non-oscillatory high order stencils like WENO5 for its advection and reinitialization.

The VPM method is particularly relevant as vorticity and its sources are confined to the neighborhood of the interface and of the solid walls. Additionally, a VPM method is quite efficient for convection-dominated flows since its Lagrangian treatment waives classical stability constraints for convection (CFL). We thus expect significantly lower computational costs together with higher accuracy for the VPM method.

Our study assesses those claims by systematically comparing the performance of these two formulations in relevant benchmarks such as dam break, rising bubble, etc.

\* Financial support from BelV is gratefully acknowledged.

## BUBBLE DYNAMICS IN PLUG FLOW REGIME FOR HORIZONTAL GAS-LIQUID FLOW

Jyotirmay Banerjee, S V National Institute of Technology Surat, India, India; Jignesh Thaker, S V National Institute of Technology Surat, India, India

Visualization study is reported for the interfacial structure of plug flow in a transparent 25 mm (inner) diameter horizontal pipe. Plug flow patterns are captured using high speed Photron FASTCAM camera at a rate of 1600 frames per second. Influence of superficial Reynolds number of liquid (ReSL) and gas (ReSG) on the shape and size of the plug bubble is examined. Expansion, contraction and breakup of plug bubble tail along the test section is analyzed for different inlet flow conditions. It is established that the expansion of plug bubble tail is due to surface tension and shearing effects between plug bubble and pipe wall. The plug bubble tail stretches until it reaches the sharp edge called the plug bubble rim. It is observed that sharp edge of plug bubble rim either contracts gradually or breaks into small bubbles as the flow progresses depending on the waviness present on the bubble rim interface. This is because of the gradual reduction in surface tension over the plug bubble rim and higher momentum of liquid plug. Small bubbles which gets detached from the bubble rim undergoes deceleration due to surface tension induced internal pressure and coalesces with the nose of following plug bubble.

\* Support from Science and Engineering Research Board (SERB), Department of Science and Technology (DST), India (sanction letter no. SB/S3/MIMER/0111/2013 dated 23-05-2014) for development of the two-phase flow facility is gratefully acknowledged.

## PREFERENTIAL CONCENTRATION IN COHERENT FLOW STRUCTURES: FOCUS ON PARTICLE-PARTICLE INTERACTIONS

Pawel Kosinski, University of Bergen, Norway; Boris V. Balakin, University of Bergen, Norway; Alexander Kartushinsky, Tallinn University of Technology, Estonia

It has been observed that particle concentration in multiphase turbulent flows may significantly vary within the flow and the particles form regions of higher and lower concentration. This phenomenon is usually called as preferential concentration and has been already intensively researched in literature. The main focus has been on investigation on particulate flows in mixing layers, wakes, jets and isotropic turbulence using both experimental and numerical tools. As the attention has been paid to particle-fluid interactions, an issue that has been perhaps less studied in this context is particle-particle interactions. Therefore the objective of the present research was to look into this problem by using numerical simulations. The Eulerian-Lagrangian approach was adopted, where the particles were immersed in coherent flow structures, here described by the Stuart vortices. The particle-particle collisions were studied using various techniques: (i) soft-sphere models, (ii) the "standard" hard-sphere model available in literature, (iii) a new hard-sphere model (developed in this research) and (iv) a hard-sphere model extended to cohesive interactions (developed in our previous research).

\* Support from grant EMP230 is gratefully acknowledged.

## DEVELOPMENT OF A FULLY COUPLED SOLVER FOR THE EULERIAN-EULERIAN APPROACH

Paul Bartholomew, Imperial College London, UK; Andrew Marquis, Imperial College London, UK; Fabian Denner, Imperial College London, UK; Berend Van Wachem, Imperial College London, UK

When modelling dense, dispersed two-phase flows involving millions or more particles, the Eulerian-Eulerian, or two-fluid model provides the only practical framework of simulation. However, classical approaches to solve this framework suffer from convergence issues and are computationally expensive. Recent developments in available computing resources have seen a move towards coupled solver methods where the implicit velocity-pressure coupling can improve convergence and robustness – especially when source term gradients are large as is usually the case in multiphase flows.

This work discusses the development of semi- and fully-coupled solver frameworks for the Eulerian-Eulerian model. Both approaches have a fully implicit pressure-velocity coupling, but in the semi-coupled approach the phase continuity equations are deferred. We present a range of test cases including dense (a fluidised bed) and more dilute examples which highlight the relative strengths and weaknesses of treating the volume fraction implicitly versus the segregated or semi-coupled treatment. This is followed by a discussion of expected future developments for these methods.

# ON THE INFLUENCE OF THE NUMERICAL STRATEGY ON THE PREDICTIVE CHARACTER OF AN EULER-EULER MODEL FOR TWO-PHASE FLOW SIMULATIONS IN SOLID ROCKET MOTOR INSTABILITIES

Valentin Dupif, ONERA, France; Marc Massot, CentraleSupléc, EM2C, France; Joel Dupays, ONERA, France; Frederique Laurent, EM2C, CentraleSupléc, France; Clement Le Touze, ONERA, France

Solid Rocket Motors involve strongly coupled two-phase flow. The presence of a polydisperse spray of particles resulting from the combustion of aluminized propellant has been shown to have a strong impact on stability and can eventually yield thrust oscillations. The ability to conduct predictive simulations of such a harsh environment is highly desirable. Euler-Euler models relying on moment methods for the disperse phase constitutes an interesting approach due to its efficiency at coupling both phases and its ability for HPC. A multi-fluid model coupled to a new numerical strategy based on operator splitting is introduced in order to cope with the natural high stiffness of the resulting systems of conservation laws. The predictive character of the method is strongly related to the possibility of using accurate methods while preserving stability and robustness in the presence of intrinsic singularities occurring in the disperse phase equations. The purpose of this contribution is to stress the impact of several numerical strategies on the solution. Relevant test cases for solid propulsion involving hydrodynamic instabilities and acoustic coupling are presented. A strategy is proposed in order to produce reliable predictions.

\* Support from ONERA is gratefully acknowledged.

# NUMERICAL PREDICTION OF SEDIMENT DEPOSITION AT A RIVER CONFLUENCE USING AN EULER-LAGRANGE METHOD

Souria Hamidouche, Université de Lorraine - LEMTA, France; Boris Arcen, Université de Lorraine - LRGP, France; Anne Taniere, Université de Lorraine - LEMTA, France

Sediment deposition near to a river confluence is a crucial challenge for navigation since it can strongly affect the bed morphology and thus the barges traffic. The prediction of particle deposition is thus important in order to optimize the dredging operations which are necessary to maintain a sufficient draft. In the present study, we numerically examine the transport and deposition process of sediments at Scarpe and Escaut confluence (north of France) using an Euler-Lagrange approach. This method is known to be able to accurately predict the behaviour and spatial distribution of inertial particles in turbulent fluid flow. In addition, polydispersity can be easily introduced. Nonetheless, contrary to the standard method (based on convection-diffusion equation for sediment concentration) used to predict the behaviour of sediments in such a complex configuration, it has a higher computational cost. In our study, the mean fluid flow is predicted by solving the 3D Reynolds-Averaged Navier-Stokes (RANS) equations. The trajectory of each sediment is then computed from the particle equation of motion and a standard stochastic dispersion model to predict the sediment/turbulence interaction. In the full length paper, we will examine the influence of the RANS model and stochastic dispersion model on the deposition of monodispersed sediments. Then, results obtained taking into account the sediments polydispersity will be presented and analysed.

# COMPUTATIONAL MODELLING OF PROPPANT TRANSPORT IN FRACTURES

Goodarz Ahmadi, Clarkson University, USA; Amir Abdollahi Mofakham, Clarkson University, USA; Dustin Crandall, National Energy Technology Laboratory, USA; Grant Bromhal, National Energy Technology Laboratory, USA

Fluid flow and proppant transport in rock fractures were studied using a computational modeling approach. Rock fracture geometries were obtained from high resolution CT images of fractured rocks using a computed tomography scanner. A series of computer simulations for a range of flow rates and proppant sizes were performed. For dilute suspensions, a one-way coupling approach was used and the transport and deposition of particles in the fracture were evaluated. For higher particle concentrations, two-way coupled simulations were performed and the effect of particle collisions was included in the analysis. It was shown that proppants of various sizes penetrate different distances into the fracture. An empirical relationship between the flow rate, proppant size and penetration distance was also presented. The simulation results may be useful in developing optimal operation conditions for hydraulic fracturing for effective performance and for minimizing environmental consequences.

# A THERMODYNAMICALLY CONSISTENT PHASE-FIELD MODEL FOR TWO-PHASE FLOWS AND ITS ENERGY-LAW PRESERVING COMPUTATIONAL METHOD

Ping Lin, University of Dundee, UK

We develop a phase-field model [1] for the binary incompressible (quasi-incompressible) fluid with thermocapillary effects, which allows for the different properties (densities, viscosities and heat conductivities) of each fluid component while maintaining thermodynamic consistency. The governing equations of the model including the Navier-Stokes equations with additional stress terms, Cahn-Hilliard equations and energy balance equation are derived within a thermodynamic framework based on entropy generation, which guarantees thermodynamic consistency. A sharp-interface limit analysis is carried out to show that the interfacial conditions of the classical sharp-interface models can be recovered from our phase-field model. Moreover, a few examples including thermocapillary convections in a two-layer fluid system and thermocapillary migration of a drop are computed using a continuous finite element method. The results are compared to the corresponding analytical solutions and the existing numerical results as validations for our model. For the isothermal variable-density case [2] we also show how an energy law preserving continuous finite element scheme can be derived.

[1] ZL Guo & P Lin, A thermodynamically consistent phase-field model for two-phase flows with thermocapillary effects, J. Fluid Mech. 766 (2015), 226-271.

[2] ZL Guo, P Lin & J Lowengrub, A numerical method for the quasi-incompressible Cahn-Hilliard-Navier-Stokes equations for variable density flows with a discrete energy law, J. Comput. Phys. 276 (2014), 486-507.

# A NUMERICAL INVESTIGATION OF PARTICLE-BED-TURBULENCE INTERACTIONS IN OSCILLATORY FLOWS

Chaitanya Ghodke, Oregon State University, USA; Sourabh Apte, Oregon State University, USA

Particle-resolved direct numerical simulations are performed to investigate the effects of an oscillatory flow field over a rough-bed, corresponding to the experimental setup of Keiller & Sleath (1976) for transitional and turbulent flows over a range of Reynolds numbers (95-400) based on the Stokes-layer thickness. The unsteady nature of hydrodynamic forces on particles and their cross-correlations with measurable flow variables are studied. Temporal correlations showed drag and lift to be positively correlated with a phase difference, which is approximately equal to the Taylor micro-scale related to drag/lift correlations. Spatio-temporal correlations between the flow field and particle force showed that the lift force is well correlated with the streamwise velocity fluctuations up to distances of the same order as the particle diameter, whereas pressure fluctuations are correlated and anti-correlated with the lift force in the front and aft regions of the particle, respectively, as a result of wake effects. Further statistical analyses showed that the near-bed velocity and pressure fluctuations fit poorly with Gaussian distribution, a result of critical importance for modeling of incipient motion of sediment grains.

# MODELING OF CORIUM SPREADING IN A CORE CATCHER

Hugo Perrier, Imperial College London, UK

In the event of a nuclear accident, reactor core meltdown can lead to failure of the reactor containment building and radioactivity releases to the public and the environment. This can be prevented by using so-called core catchers to spread the molten material (Corium) over a large area. The Corium can then be cooled efficiently and the accident progression can be stopped. Numerical simulation of Corium spreading flows is very challenging as it involves solidification of a non-eutectic mixture, free surface flow, temperature dependent fluid/solid properties, radiation heat transfer and internal volumetric heat generation.

In the present contribution a Volume of Fluid methodology is derived and discussed to capture the free surface evolution of the Corium interface, combined with a phase change model to account for latent heat effects and solidifying fluid behaviour. Special attention is required at the free surface where the solid phase, liquid phase and gas phase can be found in a single computational cell.

Validation is presented for each individual physical phenomenon and the final methodology is applied to a Corium spreading situation, and a comparison is made with experimental data. During the talk, the physical mechanisms will be discussed.

## PHASE RETRIEVAL METHOD FOR HOLOGRAPHIC PARTICLE MEASUREMENT WITH TWO CAMERAS

Yohsuke Tanaka, Kyoto Institute of Technology, Japan; Shigeru Murata, Kyoto Institute of Technology, Japan; Shunsuke Tani, Kyoto Institute of Technology, Japan

Digital in-line holography is a powerful tool for measuring size, three-dimensional position and velocity of small particles. However, it has been troubled by the so-called twin-image problem from two images appearing in the reconstruction of a holographic pattern. Phase retrieval method resolves the problem by using two or more holographic patterns. This paper presents a phase retrieval method for holographic particle measurement with two cameras. The influence of the number of particles on signal-to-noise ratio is numerically verified by comparing reconstructed particles with and without phase retrieval method. Finally, we experimentally demonstrate a measurement of free-falling particles in a static air: average particle diameter is 70  $\mu\text{m}$  and particle Reynolds number is 0.8.

## APPLICATION OF AN ARRAY OF FOUR PHASE-DETECTION PROBE SENSORS TO HYDRAULIC JUMPS AT LARGE REYNOLDS NUMBERS

Hang Wang, The University of Queensland, Australia; Hubert Chanson, The University of Queensland, Australia

The measurement of turbulent velocity in hydraulic jumps with relatively high Reynolds numbers is difficult because of the presence of entrained air bubbles. This study presents a method aimed to characterise the three-dimensional velocity field in hydraulic jumps using a four-sensor phase-detection probe array. Besides the longitudinal velocity and turbulence intensity that were measured for both positive and negative velocity flow regions, a characteristic instantaneous transverse velocity component was derived together with a measure of its fluctuations. The transverse velocity component characterised the three-dimensional nature of turbulent structures, although the time-averaged flow pattern was two-dimensional and the average transverse velocity was zero. The corresponding velocity fluctuation was found to be one order of magnitude smaller than the longitudinal velocity fluctuation. While the longitudinal fluctuation was significantly affected by the large-scale flow instabilities, the instantaneous transverse velocity fluctuation was thought to be free of the effects of these unsteady, pseudo-periodic motions.

\* Support from grant ARC DP120100481 is gratefully acknowledged.

## A NUMERICAL STRATEGY TO ACCOUNT FOR THE EFFECT OF SELF-INDUCED GEOMETRY CHANGES IN WEAR ESTIMATION

Gianandrea Vittorio Messa, Politecnico di Milano, Italy; Stefano Malavasi, Politecnico di Milano, Italy

The erosion of a surface impinged by solid particles dragged by a liquid is a serious concern in the oil&gas industry. Recent experiments indicate that the changes in the geometry produced by the erosion may significantly alter the evolution of the erosion process itself. This typically happens when a surface is exposed to aggressive flow condition for a prolonged period, and it becomes a significant issue for complex devices such as control valves. However, none of the predictive techniques available at present, which mainly involve the use of algebraic erosion correlations in conjunction with an Eulerian-Lagrangian model, appears capable in handling this phenomenon. In this study, we propose a numerical strategy that allows accounting for the effect of the self-induced geometry changes on the wear estimates. The strength of the method resides in its computational efficiency, arising from the fact that the evolution of the erosion process is estimated by an effective post-processing technique applied to the solution of a steady-state Eulerian-Lagrangian simulation. The good agreement between our predictions and experimental data available in the literature regarding an abrasive jet impingement test makes the application of this strategy to more complex flows very promising.

## SIMULTANEOUS ANALYSIS OF ELECTROCHEMICAL IMPEDANCE SPECTROSCOPY AND VISUALIZATION TECHNIQUE TO EXPLORE TWO-PHASE FLOW BEHAVIOUR IN THE PROTON EXCHANGE MEMBRANE FUEL CELLS (PEMFCs)

Homayoon Kanani, Islamic Azad University of Hashtgerd, Iran; Shams Mehrzad, Khaje Nasir Toosi university of technology, Iran

Water management is one of the most important issues in the Proton Exchange Membrane Fuel Cells (PEMFCs). Excessive liquid water and condensation of water vapour could block the gas channels and decrease the performance of the cell. Electrochemical Impedance Spectroscopy (EIS) is a powerful tool to the prediction of water blockage and slug flow formation in the channels.

In this experimental study, simultaneous EIS of the cell and direct visualization of the cathode side of the serpentine and parallel transparent PEMFCs are used to explain flooding phenomena and slug/plug flow formation. The results of visualization show that the permanent blockage of the channels could occur in the parallel flow field design, while it is periodically in the serpentine design. Water coverage ratio in the channels was obtained by image processing technique. Analysis of the results shows that the low frequency data of the EIS can track the variation of water content in the channels. For serpentine flow field, EIS method could predict all of slug/plug formation in the channels about 100 to 200 seconds before happening.

## MULTIPHASE FLOW QUANTIFICATION USING MAGNETIC RESONANCE IMAGING AND COMPUTATIONAL FLUID DYNAMICS

Andrew Sederman, MRRC, UK; Wes Maru, Oil & Gas Measurement Limited, UK; Susithra Lakshmanan, MRRC, UK; Daniel Holland, University of Canterbury, New Zealand

Liquid-liquid flows are widely encountered in many industrial applications; despite their importance, liquid-liquid flows are not as well studied as gas-liquid flows. We have experimentally investigated oil-water flow in order to quantify the flow regime, water-cut and the mixing efficiency of a mixing device with a Jet In Cross Flow (JICF) configuration. Such configurations are utilised in applications such as liquid sampling.

We have commissioned a multiphase flow loop and developed Magnetic Resonance Imaging (MRI) acquisition methods to quantify the water-cut as a function of position in the pipe flow. For this, a chemically shift selective (CHESS) MR sequence was developed to remove unwanted signal from the oil component and only measure the water phase. The sequence was quantified with various water-oil mixtures up to 25%. The CHESS sequence was then combined with imaging sequences to visualize the water distribution. This technique has been used to study the mixing at a range of different flow rates, water cuts and mixing configurations and to characterise a novel mixing device that is used to homogenise oil-water flows. We also developed an advanced computational fluid dynamics (CFD) model based on the OpenFOAM® computational framework to simulate the water-oil mixing under JICF configuration which is then validated with the experimental measurements. Support from Innovate UK and OGSL, UK is gratefully acknowledged.

## COMPARISON OF TWO METHODS CALCULATING PARTICLE IMPACT ON A FLAT PLATE IN ORDER TO DEDUCT HYDRO-ABRASIVE EROSION

Wiebke Boden, Ecole Centrale de Lyon, France; Stephane Aubert, Ecole Centrale de Lyon, France; Jean-Christophe Marongiu, Andritz Hydro SAS, France; Richard Perkins, Ecole Centrale de Lyon, France

Particles suspended in water jets impacting a surface cause surface damage by the mechanism of hydro-abrasive erosion. Existing erosion models are able to reproduce an approximate wear map of the surface. Nevertheless the latter suffer from significant qualitative and quantitative nonconformance when compared to wear maps in reality. Consequently, to numerically produce meaningful wear maps it is necessary to directly compute the particles' impact on the surface to deduct therefrom the subsequent abrasive damage. In the following the statistical impact maps obtained by a numerical approach will be compared to those of an analytical solution. We will address the question of the additional benefit by a numerical approach regarding the beforehand mentioned purpose. For the numerical computation an in-house available Smoothed-Particle-Hydrodynamics (SPH) code developed by ANDRITZ Hydro AG expressed in an Arbitrary-Lagrangian-Eulerian framework (SPH-ALE) approach and implying one-way-coupling is used. Turbulent dispersion of the inertial particles and wall effects are taken into account. As within the SPH method each fluid and suspended particle is individually tracked this becomes expensive regarding computational costs for simulation of entire wear maps. Hence the results of a simple but comparable analytical approach of particles in potential flow, involving low computational costs, are shown.

\* Support from grant Initial Training Networks FP7-PEOPLE-2013-ITN is gratefully acknowledged.

#### FUEL SPRAY SIMULATION DURING OPENING VALVE OF FUEL INJECTOR WITH PARTICLE METHOD

Eiji Ishii, Hitachi, Ltd., Japan; Kazuki Yoshimura, Hitachi, Ltd., Japan; Yoshihito Yasukawa, Hitachi, Ltd., Japan; Hideharu Ehara, Hitachi Automotive Systems, Ltd., Japan

Particulate matters (PM) in exhaust gas from automobile engines bring serious air pollution. PM is formed in regions of high density of fuel. The fuel stuck on walls of combustion chambers forms a high-density region of fuel in the air/fuel mixture, which becomes a source of PM. To decrease the amount of PM, the fuel stuck on the walls needs to be decreased by optimizing spray forms. We previously developed a fuel-spray simulation that combined the fuel-breakup simulation near nozzle outlets by using a particle method with the air/fuel mixture simulation by using a discrete droplet model. We furthermore modified our fuel-spray simulation in order to simulate the late fuel near nozzle outlets during closing the valve of fuel injector. In this study, we studied the effects of opening valve of fuel injector on fuel sprays. The simulation results were validated by comparing the simulated fuel breakup near nozzle outlets and air/fuel mixture in the air region with measure ones; the simulated liquid-column breakups near nozzle outlets and air/fuel mixture in the air region agreed well with measure ones. It was also found that the opening motion of valve changed the air/fuel mixture, especially the spray penetration.

#### SURFACTANT INDUCED LATERAL MIGRATION OF BUBBLES AND DROPS IN A SHEAR FLOW

Metin Muradoglu, Koc University, Turkey; Gretar Tryggvason, The University of Notre Dame, USA

Fully three-dimensional direct numerical simulations are performed to examine effects of soluble surfactant on the lateral migration of drops and bubbles in a pressure-driven channel flow. The interfacial and bulk surfactant concentration evolution equations are solved fully coupled with the incompressible Navier-Stokes equations. A non-linear equation of state is used to relate interfacial surface tension to surfactant concentration at the interface. Neutrally buoyant, upflow and downflow cases are considered. Extensive computations are carried out to investigate the effects of surfactant on the transverse migration of single drops and bubbles for a range of capillary (Ca), Reynolds (Re) and Eotvos (Eo) numbers. It is found that the surfactant induced Marangoni stresses can dominate over the shear-induced lift force and thus alter the dynamics of the drops/bubbles completely, i.e., the contaminated drops/bubbles drift away from the channel wall and stabilize at the center of the channel while the corresponding clean drops/bubbles move toward the wall and reach a steady motion near but with a certain distance from the wall.

\* Support from TUBITAK grant No. 112M181 and COST Action MP1106 is gratefully acknowledged.

#### USING CENTRIFUGAL FORCES IN A PITOT-TUBE JET-PUMP FOR CLEANING AND PUMPING OIL-WATER MIXTURES IN A CONTINUOUS LOOP: AN EXPERIMENTAL STUDY OF SEPARATION EFFICIENCY

Jan Meyer, University of Magdeburg "Otto von Guericke", Germany; Dominique Thevenin, University of Magdeburg "Otto von Guericke", Germany

In this study an unconventional radial pump (the Pitot-Tube Jet-Pump) is adapted for separating multiple fluids with the help of centrifugal forces. Instead of a spiral casing or a volute, a pick-up tube is used. This pick-up tube inside the pump is the key feature towards multi-fluid separation. To the author's knowledge this combination of pumping and separation is unique and cannot be found in any other separation device delivering a continuous loop for oil-water separation. Due to the promising results of previous studies, a first patent already has been filed together with an industrial partner.

In the current study a complete test-rig has been set up using tap water and sunflower oil as fluids. The focus is on quantifying the separation efficiency while varying the operation point (volume fractions at inlet, flow-rate and rotational speed). At the conference the impact of the key parameters on separation efficiency will be shown, leading to the identification of an optimal operation point for this new technology. Direct visualizations of the separation process will also be presented together with a quantitative analysis of separation times, showing the flow features inside the PTJ-Pump and the interaction between the pick-up tube and the fluid mixture leading to back-mixing.

#### COMPUTATIONAL STUDY OF THE ROLE OF SURFACTANTS IN SHEARED FOAMS FOR FOAM STABILITY

Andrea Titta, Université Claude Bernard Lyon 1, France; Francois Detcheverry, Université Claude Bernard Lyon 1, France; Marie Le Merrer, Université Claude Bernard Lyon 1, France; Anne-Laure Biance, Université Claude Bernard Lyon 1, France; Peter Spelt, Laboratoire de Mécanique des Fluides et d'Acoustique, France

The dynamics of sheared wet foams is investigated computationally in this study. For this purpose, an established 3D (parallel) implementation of a level-set method for incompressible two-phase flow has been extended to account for the presence of surfactants that are soluble in the liquid, and the associated modified stress conditions at interfaces. In particular, we account for surface rheological behavior (such as surface viscosity) beyond minimal surfactant models, to describe realistic systems. The results of 2D and 3D tests will be demonstrated to compare favourably with the literature. We shall report on the role of surfactants and their properties on T1 events, wherein adjacent bubbles are sheared past each other, and on the onset of bubble coalescence, with implications for foam instability.

\* Support from grant ARC Energie, by Rhône-Alpes region, is gratefully acknowledged.

#### STUDY OF THREE-DIMENSIONAL STRUCTURE AND COALESCENCE OF DISTURBANCE WAVES ON GAS-SHEARED LIQUID FILM IN A HORIZONTAL RECTANGULAR DUCT

Andrey Cherdantsev, Kutateladze Institute of Thermophysics, Russian Federation; Ian McCarthy, University of Nottingham, UK; David Hann, University of Nottingham, UK; Buddhikha Hewakandamby, University of Nottingham, UK; Barry James Azzopardi, University of Nottingham, UK

Disturbance waves are large lumps of liquid travelling with high speed over the surface of liquid film in presence of high-velocity gas stream. Knowledge of structure, properties and dynamics of disturbance waves is vital for creation of physically-based models predicting the overall characteristics of annular flow. The present work examines the transverse coherence and downstream evolution of these waves. Though they form full rings in narrow pipes, they lose transverse coherence in larger pipes, a property which returns further downstream. Other properties of these waves change with downstream distance by multiple coalescence. Brightness-based LIF technique was applied to study the processes of evolution and interaction of disturbance waves in a horizontal rectangular duct. Measurements were performed over an area of duct's bottom with length of 280 mm and width of 161 mm with frequency of 1 kHz and spatial resolution of 0.22 mm/pixel. It was shown that the disturbance waves are initially localized by the transverse coordinate and their fronts are curved. The decrease in transverse coherence with increasing pipe diameter and gas velocity was explained in terms of transverse size of disturbance waves. Two scenarios of 3D coalescence increasing the transverse coherence of film wavy structure were observed.

#### LIQUID-LIQUID ENTRAINMENT AND FILM WRAPPING AROUND A ROTATING CYLINDER IN STRATIFIED LAYER

Basanta Kumar Rana, Indian Institute of Technology Kharagpur, India; Yagya Sharma, Indian Institute of Technology Roorkee, India; Arup Kumar Das, Indian Institute of Technology Roorkee, India, India; Prasanta Kumar Das, Indian Institute of Technology Kharagpur, India

Entrainment and dynamics of wrapping film around the horizontal rotating cylinder in stratified liquid-liquid layer is investigated using volume of fluid (VOF) based finite volume solver. Due to imposed rotational field in the cylinder, finger of lighter liquid penetrates inside the heavier one and bends around the cylinder progressively. Subsequently it wraps the cylinder and encapsulates it from the heavier fluid. We observed that the film attains maximum thickness at the base and thins azimuthally in the direction of rotation of cylinder. Thickness of liquid film at any radial plane from the cylinder and depth of penetration of liquid entrainment depends on the liquid properties ( $\rho_1$ ,  $\rho_2$ ,  $\mu_1$ ,  $\mu_2$  and  $\sigma$ ) and size of the roller (D) respectively. Rotational speed ( $\omega$ ) allows control of the phenomenon and its proper selection (critical speed) is necessary for complete wrapping of the cylinder. At a speed lower than critical, liquid film makes from the stratified interface and breaks as tiny droplets after partial wrapping. Subtended angle of partial wrapping depends on rotational speed provided fluids and roller diameter is kept constant. Present numerical analysis develops better understanding about enhancement of heat and mass transfer through the film and provides alternate idea of encapsulation.



## MULTISCALE MODELLING OF HEAT TRANSFER FROM ARRAYS OF SPHERICAL PARTICLES

Arpit Singhal, NTNU, Norway; Schalk Cloete, SINTEF Materials and Chemistry, Norway; Stefan Radl, TU Graz, Austria; Shahriar Amini, SINTEF Materials and Chemistry, Norway

This work presents a modelling study of gas-particle heat transfer on two distinct scales. Firstly direct numerical simulations (DNS) are conducted in a geometry of spherical particles generated via the discrete element method (DEM). Simulations are completed on random particle arrays ranging from a void fraction of 0.9 to maximum packing over a range of Reynolds and Prandtl numbers. The geometry is meshed with a fine Cartesian cut-cell mesh both inside and outside the particles. These DNS results are then used to provide improved momentum and heat transfer closures to an unresolved Lagrangian modelling approach which can be used to simulate much larger particle beds. The unresolved Lagrangian approach also incorporates a 1D heat conduction model to directly simulate heat transfer inside the particles. This model is then verified against DNS data in geometries where wall effects and intra-particle heat transfer, both of which are directly accounted for by the Lagrangian approach, are important. Minor differences in results are discussed and the achievable computational speedup by this approach is quantified.

\* The research leading to these results has received funding from the European Union's Seventh Framework Programme (FP7/2007-2013) under grant agreement n° 604656.

## EULERIAN AND FULL LAGRANGIAN MODELING OF A FLOW AND HEAT TRANSFER IN A GAS-DROPLET SWIRLING FLOW IN A PIPE SUDDEN EXPANSION

Maksim Pakhomov, Kutateladze Institute of Thermophysics, Russian Federation; Viktor Terekhov, Kutateladze Institute of Thermophysics, Russian Federation

Swirling droplet-laden turbulent flows in a pipe with sudden expansion are widely encountered in spray dryers, and industrial combustors, where they are used to improve flame stabilization and minimize pollutant production. The effect of droplets evaporation on the flow structure, turbulence modulation and heat transfer in the swirling confined mist flow is numerically investigated. The numerical model is based on Eulerian by Zaichik (1999) and full Lagrangian by Osipov (2000) approaches. The set of steady-state 3D RANS equations is used for the computations of the gas phase. Gas phase turbulence is modeled with the use of the elliptic blending second moment closure. Particles mean flow interactions are described by a two-way coupling model. The full Lagrangian approach is used for computation of droplets velocities and its concentration field.

The results of comparative analysis between Eulerian and full Lagrangian models are presented. Difference between particles velocity fluctuations determined by Eulerian or Lagrangian approaches is up to 15 %. The Eulerian method underpredicts the value of particles mass fraction than Lagrangian method (up to 20 %) both in recirculation region and in the flow core. The effect Eulerian or Lagrangian approaches on heat transfer between wall and mist flow less than 15%.

\* The work was supported by the Russian Scientific Foundation (Project No. 14-19-00402).

## OBSERVATION OF NON-SPHERICAL DROPLET IMPACTS WITH VELOCITIES OF O(10 M/S)

Yutaka Kataoka, Hokkaido University, Japan; Yuusaku Hori, Hokkaido University, Japan; Masao Watanabe, Hokkaido University, Japan; Kazumichi Kobayashi, Hokkaido University, Japan; Hiroyuki Fujii, Hokkaido University, Japan; Toshiyuki Sanada, Shizuoka University, Japan

Droplet impacts on solid surface are key elements in technical applications such as semiconductor cleaning or ink-jet printing. Among the broad varieties of parameters that control splash formation after the droplet impact, impact velocity is one of the most important parameters: however, in the range of high impact velocities, splash formation has been hardly investigated partially because droplets with high impact velocity cannot keep spherical shapes due to the surrounding gas flow. We developed a high-speed droplet injector using a syringe to observe droplet impacts with velocities of O(10 m/s) on an acrylic solid surface with a high-speed photography. We succeeded to generate non-spherical droplets with velocities up to 23.8 m/s by using our high-speed droplet injector. The results show that the droplet dynamics after the impact is closely related with the droplet shape, especially the local curvature of non-spherical droplet that undergoes shape oscillation, immediately before the impact on the solid surface. We investigated the contributions of both the radius of curvature of droplet and the impact velocity on the splash formation. We also compared our results with the splash/non-splash boundary reported by Vandel Wal et al. (Exp. Fluids. 40 (2006) 53-59).

\* Support from JSPS KAKENHI 26289031 is gratefully acknowledged.

## 3D NUMERICAL SIMULATION OF UPFLOW BUBBLING FLUIDIZED BED IN OPAQUE TUBE UNDER HIGH FLUX SOLAR HEATING

Hadrien Benoit, PROMES-CNRS, France; Renaud Ansart, LGC, France; Pablo Garcia Trinanes, University of Surrey, UK; Daniel Gauthier, PROMES-CNRS, France; Gilles Flamant, PROMES-CNRS, France; Olivier Simonin, INP Toulouse, France

Current solar Heat Transfer Fluids (HTF) have a limited working temperature (< 600 °C). We proposed to use air-fluidized Dense Particle Suspensions (DPS), also called Upflow Bubbling Fluidized Bed (UBFB), in tubes as a new HTF and storage medium in the frame of the so-called CSP2 FP7 European project (<http://www.csp2-project.eu/>). UBFB can operate up to the solid sintering temperature, thus improving the plant efficiency and cost of produced kWh. The DPS capacity to extract heat from a tube absorber exposed to concentrated solar radiation was demonstrated and the first values of the tube wall-to-DPS heat transfer coefficient were measured. A stable outlet temperature of 750 °C was reached with a metallic tube, and a particle reflux in the near tube wall region was evidenced (H Benoit, I Pérez López, D Gauthier, JL Sans, G Flamant. Sol. En., 118:622-633, 2015). In this paper, the UBFB behavior is studied using the multiphase flow code NEPTUNE\_CFD. Hydrodynamics of SiC Geldart A-type particles and heat transfer imposed by a thermal flux at the wall are coupled in 3D numerical simulations. The convective/diffusive heat transfer between the gas and dispersed phase, and the inter-particle radiative transfer (Rosseland approximation) are accounted for. The simulations are compared to experiments. The reflux and the heat transfer mechanism (particle exchange between wall and tube center) are analyzed.

## LARGE EDDY & INTERFACE SIMULATION (LEIS) OF DISTURBANCE WAVES AND HEAT TRANSFER IN ANNULAR FLOWS

Junfeng Yang, Imperial College London, UK; Geoffrey Hewitt, Imperial College London, UK; Chidambaram Narayanan, ASCOMP AG, Switzerland; Djamel Lakehal, ASCOMP AG, Switzerland

A numerical method for forced convective boiling in an annulus needs to be developed in order to elucidate the reason for nucleation enhancement by disturbance waves. We first developed a numerical strategy to model the development of disturbance waves in annular flows where the highly turbulent gas core flow drives the laminar liquid flow upwards using advanced CFD tool TransAT. In which, the interface tracking method (e.g. Level-set) combined with a scale-resolving turbulence simulation technique (Large Eddy Simulation) was employed to capture dominant turbulence and interfacial scales using low-to-medium computational costs but reasonable accuracy. The method involves filtering continuity and Navier-Stokes equations a-priori defined for the one-fluid formulation. Then, the mass- and heat- transfer processes in the non-boiling annular flow was investigated to provide insight into the temperature gradient underneath the wave region. The modelling results are indicative and show that heat transfer is hindered in the wave region. The locally overheated zones underneath the disturbance wave could play key roles in activating the nucleation boiling sites.

Support from grant NURESAFE is gratefully acknowledged.

## EFFECTS OF SURROUNDING GAS TEMPERATURE ON ETHANOL DROPLET IMPACT ON SOLID SURFACE

Toshihisa Isono, Hokkaido University, Japan; Masao Watanabe, Hokkaido University, Japan; Kazumichi Kobayashi, Hokkaido University, Japan; Hiroyuki Fujii, Hokkaido University, Japan; Toshiyuki Sanada, Shizuoka University, Japan

Droplet impacts on solid surface are key elements in technical applications, such as rapid spray cooling, ink-jet printing and semiconductor cleaning. Although the broad varieties of parameters that control splash formation after the droplet impact, are proposed, they are not thoroughly explored; hence, dynamics of the droplet after the impact are yet fully understood. After droplet impacts on solid surface, characteristic flow called lamella which is the liquid film flow to progress along the solid surface is developed; then splash which consists of the secondary droplets ejected from the edge of the lamella may be generated. In this study, we examine the effects of the surrounding gas temperature on ethanol droplet impact. We experimentally observed the droplet impact on a glass surface in a vacuum chamber with a high-speed photography. We found that splash can be suppressed with the increase of the surrounding gas temperature. We measured the splash threshold pressure for a splash to appear as a function of the temperature ranging from 30 °C to 80 °C. The results show that the splash threshold pressure is a monotonically increasing function of the surrounding gas temperature.

\* Support from JSPS KAKENHI 26289031 is gratefully acknowledged.

## **FREEZING DELAY AND NUCLEATION RATES OF SUPERCOOLED WATER DROPS IMPACTING ONTO A COLD SURFACE**

Markus Schreimb, Technische Universitaet Darmstadt, Germany; Ilia V. Roisman, Technische University Darmstadt, Germany; Suad Jakirlic, Technische Universitaet Darmstadt, Germany; Cameron Tropea, Technische University Darmstadt, Germany

The freezing delay and nucleation rate of supercooled liquid droplets, which have been regularly investigated for droplets resting on a cold surface, are important parameters also in the case of impacting drops. Since the drop may continually alter its shape until freezing starts, the freezing delay time has a crucial influence on the overall surface area which is eventually covered by ice. To examine the role of the fluid temperature in the process of nucleation and freezing of supercooled water under motion, the normal impact of supercooled water drops onto a sand-blasted glass surface is investigated experimentally. The water drops and impact surface are cooled down to a temperature varying between -6 °C and -16 °C. Although the drop diameter and impact velocity are kept constant, the Reynolds number ranges from approx. 2000 to 4000 due to the strong temperature dependence of the viscosity of water. We use a high-speed video system to observe the impact process in top-view and to accurately detect the moment of nucleation. Freezing rates are calculated and analyzed based on a statistical model for heterogeneous nucleation. It is shown that the influence of the drop temperature on nucleation is twofold, with respect to alteration of both the hydrodynamics and thermodynamics of the process.

## **DEPOSITION CHARACTERISTICS IN PARTICLE-LADEN TURBULENT FLOWS**

Laura Nicolaou, Imperial College London, UK; Tamer Zaki, Johns Hopkins University, USA

The transport and deposition of particles in complex biological flows is a key consideration in the optimisation of drug delivery. Our work is motivated by aerosol delivery for the treatment of respiratory diseases. The complexity of the airways geometry and inter-subject variations lead to significant spread in deposition efficiency in experiments and simulations. Particle deposition in the airways is typically described as a function of the Stokes number based on a reference flow timescale. However, the velocity and length scales experienced by the particles as they are advected through the flow differ considerably from the reference quantities in many sections of the airways. Therefore, the use of an instantaneous Stokes number based on the local properties of the flow field is adopted instead. An effective Stokes number can then be defined as the average of the time-dependent value. We examine the evolution and distribution of the effective Stokes number in a canonical flow configuration in order to assess its suitability as a deposition parameter. Our results demonstrate that the effective Stokes number can deviate significantly from the reference value particularly in the intermediate Stokes number range. Particles of the same size experience very different effective Stokes numbers as they are advected through the flow, depending on their initial release position. In addition, the effective Stokes number shows a very clear correlation with deposition efficiency, and is therefore a more appropriate parameter to describe aerosol transport.

## **LATTICE BOLTZMANN MODEL FOR PREDICTING THE DEPOSITION OF INERTIAL PARTICLES IN TURBULENT CHANNEL FLOWS**

Guillaume Lepoutre, IMFT, France; Pascal Fede, INP Toulouse, France; Victor Sofonea, Romanian Academy, Romania; Richard Fournier, LAPLACE, France; Stephane Blanco, LAPLACE, France; Olivier Simonin, INP Toulouse, France

Deposition of inertial solid particles transported by turbulent flows is modelled in a framework of a statistical approach based on the particle velocity Probability Density Function (PDF). The particle-turbulence interaction term is closed in the kinetic equation by a model widely inspired from the famous BGK model of the kinetic theory of rarefied gases. A Gauss-Hermite Lattice Boltzmann model is used to solve the closed kinetic equation involving the turbulence effect. The Lattice Boltzmann model is used for the case of the deposition of inertial particles transported in an inhomogeneous boundary layer. The turbulent r.m.s. velocity and time scale profiles are obtained from experimental data. The presence of the wall coupled with particle-turbulence interactions leads to inhomogeneous particle distribution and non-equilibrium particle fluctuating motion. The deposition velocity are compared with Liu & Agarwal's experimental results.

## **DISCUSSION OF SURFACTANTS MIGRATION ON DYNAMIC INTERACTIONS BETWEEN DEFORMABLE DROPLET**

Hang Jin, China University of Petroleum, Beijing, China; Wei Wang, China University of Petroleum Beijing, China; Fangyuan Liu, China University of Petroleum, Beijing, China; Mengyu Ma, China University of Petroleum, Beijing, China; Kai Li, Institute of Mechanics, Chinese Academy of Sciences, China; Jing Gong, China University of Petroleum Beijing, China

The drainage and deformation of intervening film between two colliding drops has been investigated for decades, and a recent establishment of forces measurement between emulsion droplets (AFM) probes a new insight into the investigation of drop deformability and interfacial properties during film drainage. To reveal the nature of surfactants migration at interface, interaction force between two oil droplets in aqueous phase with trace amount of non-ionic surfactant (span80) is accurately measured and helped to evaluate the drainage process. Based on Reynolds lubrication theory and augmented Young-Laplace equation, theoretical models have been developed and the interaction force during drainage is successfully predicted, including the effects of interfacial deformation, hydrodynamic force and static surface forces. To illustrate the migration of surfactants at the interface, convection-diffusion equation is coupled with theoretical model where non-uniformed distribution of surfactants is successfully simulated with time-dependent drop deformations. Results indicate that predictions with mobile interface, surfactant convection and diffusion can well describe the interaction forces between droplets, revealing the migration of non-ionic surfactants absorbed on the interface.

\* Support from the National Natural Science Foundation of China(51422406) is gratefully acknowledged.

## **FINE PARTICLE DEPOSITION IN DUCT FLOW WITH STAGGERED BAFFLES**

Jiandong Wu, Peking University, China; Hongwei Shen, Peking University, China; Qiao Liu, Peking University, China; Hao Wang, Peking University, China

Particle pollution has become serious in developing countries. In the present work the particle deposition in a duct flow with repeated stagger baffles was investigated experimentally and numerically. In the experiment, deposition efficiency was obtained for particles of dimeters of 0.1-10 µm. In the simulation, Reynolds Stress Model with enhanced wall boundary condition was adopted to get air flow field, and Lagrangian method was employed to get the particle trajectory. Experiment showed when air flow velocity was 1 m/s, deposition efficiency increased as particle diameter increased. The simulation results are consistent with the experimental results. The influence of the flow speed and the particle size on the deposition efficiency are analyzed.

## **TENSORIALLY INVARIANT NORMALIZED LANGEVIN MODEL FOR PARTICLE DISPERSION AND DEPOSITION IN TURBULENT FLOWS**

Dmitrii Sikovsky, Novosibirsk State University, Russian Federation

We have developed the new stochastic normalized Langevin models for particle dispersion in non-homogeneous turbulent flows. In contrast to the previous normalized Langevin models of Boksell and Loth (2006), Dehbi (2010) and Sikovsky (2015) the proposed model is tensorially invariant and satisfies all the criteria established in Minier et al. ["Guidelines for the formulation of Lagrangian stochastic models for particle simulations of single-phase and dispersed two-phase turbulent flows", Phys. Fluids, 26, 113303 (2014)]. The model terms such as the diffusion matrix and the drift vector are determined from the asymptotic solution of the corresponding Fokker-Planck equation in the viscous sublayer of wall-bounded turbulent flows. We have shown that despite its strong inhomogeneity the particle statistics in the viscous sublayer is near-equilibrium even at large Stokes numbers of particles, so the Chapman-Enskog expansion can be utilized to obtain the needed closure relations for the terms of the Langevin model. The results of simulation of particle concentration, second moments of particle velocity and deposition rates in particle-laden channel and pipe flows obtained with the proposed model demonstrate the good agreement with both experimental and DNS/Lagrangian tracking data in a wide range of particle Stokes number.

#### PHENOMENOLOGICAL STUDY OF PHASE INVERSION IN A LIQUID-LIQUID DISPERSION PIPE FLOW

Schehinez Idoughi, INP Toulouse, France; Nathalie Le Sauze, Toulouse university, France; Emmanuel Cid, Toulouse university, France; Olivier Masbarnat, Tlemoine, France

Phase inversion phenomenon of a concentrated o/w dispersed flow in a vertical pipe has been investigated. Flow parameters are the phase ratio and the mixture velocity. A Planar Laser Induced Fluorescence technique is used to discriminate between the liquid phases and allows the visualization of the dispersion structure in a control volume. A configuration map of the dispersion is first drawn up on a phase ratio-mixture velocity graph. The transition towards phase inversion is characterized by the coexistence of multiple dispersions, with a large drop size and presence of aqueous inclusions in large oil drops. An accurate image processing is then requested to track the maximum size of dispersed oil phase and the number of aqueous inclusions due to phase inversion. It is shown that in the present configuration, phase inversion results from the trapping of aqueous phase drops in oil drops during the coalescence of oil drops, leading to a simple correlation between oil drop size and the number of trapped inclusions. It is therefore a continuous process driven by the coalescence rate, which is a growing function of phase ratio and total flowrate. A simple macroscopic model is proposed for this phase inversion mechanism.

#### AN EXPERIMENTAL STUDY OF UPWARD-VERTICAL OIL-WATER-GAS THREE-PHASE FLOW: FLOW PATTERN, PRESSURE DROP AND VOLUMETRIC PHASE FRACTION

Douglas Martins Rocha, University of São Paulo, São Carlos School of Engineering, Brazil; Carlos Henrique Monteiro De Carvalho, Petrobras, Brazil; Valdir Estevam, Petrobras SA, Brazil; Oscar Mauricio Hernandez Rodriguez, University of São Paulo, Brazil

The presence of water and gas is common in the offshore oil production scenario. Several techniques, such as the gas-lift and the water-assisted technique, utilize gas or water to increase oil-production productivity. Further research on natural or artificial three-phase flow in oil wells is in order. Several studies on three-phase flow have been published over the last years, but the results so far are not as conclusive as those on two-phase flow. In this work some characteristics of three-phase flow are investigated and compared with available data from the literature. New data of pressure drop and volumetric phase fraction are reported and different oil viscosities are tested. A classification of flow patterns in three-phase flows is presented as a function of superficial velocities. The hypothesis of no-slip between water and oil is evaluated and the benefits of water and gas injection in the reduction of the total pressure gradient are investigated.

\*The authors would like to thank PETROBRAS and ANP for the financial support and CNPq for the research grant.

#### BUBBLE GROWTH AND AGGLOMERATION IN DRYING PAINT FILMS

Nazli Saranjam, University of Toronto, Canada; Sanjeev Chandra, University of Toronto, Canada

Paint films with uniform thicknesses were applied on stainless steel and glass substrates using a model paint consisting of a resin dissolved in butanol. Test samples were cured in a natural convection oven at temperatures varying from of 100-140°C. Photographs of the paint surface were taken during drying and weight loss measured. Experiments were performed on the model paint and on glycerin-butanol solutions containing small air bubbles or tracer particles. Photographs of the surface of drying paint films showed regular undulations appearing on the surface, induced by surface-tension-gradients that occur as soon as the curing/drying process begins. Particles and micro bubbles moved in the drying paint layer as a result of surface-tension driven flows, and bubbles grew larger as the evaporating solvent diffused into them. Bubbles appeared to be drawn towards each other and formed clusters. Analytical solutions of the mass-diffusion equation were used to model solvent evaporation from the paint film surface, calculate the magnitude of concentration gradients, and estimate bubble growth rates. The concentration gradient between growing bubbles was used to predict the velocity with which bubbles approached each other to form clusters.

#### WIRE-MESH-BASED TOMOGRAPHY IN UPWARD-VERTICAL OIL-WATER-AIR THREE-PHASE FLOW

Hugo Fernando Velasco Pena, University of São Paulo, Brazil; Adriana Bonilla Riano, University of São Paulo, Brazil; Carlos Henrique Monteiro De Carvalho, Petrobras, Brazil; Valdir Estevam, Petrobras SA, Brazil; Oscar Mauricio Hernandez Rodriguez, University of São Paulo, Brazil

The imaging techniques based on wire-mesh sensor have been used successfully to the study of two-phase flows in several conditions. The technique has a good cost-benefit ratio, but the sensor is intrusive, which is its main disadvantage. There are many application that require fast tomographic imaging of flow inside pipes in the oil&gas, nuclear, food and chemical industries. There are two kinds of wire-mesh sensors that can be applied in two-phase flows, the conductive and the capacitive. Due to its low cost and flexibility, there are several studies to extend and upgrade the measurement techniques and allow it to work with three-phase flows. This work presents a wire-mesh-based tomograph (WMT) developed to be applied in oil-water-air flow. The primary measurement is based on a digital IQ demodulator, but the system is capable of including different types of primary measurements. We show the results of this new type of WMT applied to dynamic three-phase-flow tests in a big experimental setup (Multiphase-Flow Test Facility of the Laboratory of Thermal and Fluids Engineering of the University of Sao Paulo, Brazil). The experiments were conducted in a vertical glass pipe of 50-mm i.d. and 12-m height and the observed three-phase flow pattern was constituted of intermittent gas bubbles and dispersed oil droplets in a continuous water flow. The WMT data are compared with data of a synchronized high-speed video camera and quick-closing valves. The implemented WMT can reach 7000 fps with a 16×16 sensor.

#### EXPERIMENTAL STUDY OF HEAVY OIL-WATER CORE-ANNULAR FLOW PATTERN

Caio Cavicchio, State University of Campinas, Brazil; Antonio Carlos Bannwart, State University of Campinas, Brazil; Marcelo Souza De Castro, State University of Campinas - UNICAMP, Brazil; Jorge Luiz Biazussi, State University of Campinas, Brazil

It is well known that different flow patterns leads to different values of pressure drop. Therefore, it is important to study the application possibilities of these flow patterns when regarding production and transport of crude and/or petroleum mixtures, once implies in efficient energy/costs management for the industry. Following this as inspiration, the present work carried out a set of experimental tests with diesel-diluted heavy oil, at different viscosities. The flow pattern used for the experiments was the core annular flow – water in the annulus and heavy oil in the center of the pipeline, which is believed to give the lesser pressure drop amongst them for viscous and ultra-viscous oils. It already had been reported in the literature pressure drops almost 100 times smaller in respect to the oil single phase flow, being in the same order of water single phase flow in the pipeline.

The tests were done initially with two-phase flow (water-oil), then three-phase flow (water-oil-gas). The slip and holdup between the phases were obtained by a slow motion footage technique.

\* The support granted by PETROBRAS and ANP was undoubtedly acknowledged.

#### MODEL OF BUBBLE RISE AND COLLISION WITH COMPLEX DEFORMABLE INTERFACES

Derek Chan, University of Melbourne, Australia; Rogerio Manica, Institute of High Performance Computing, Singapore; Evert Klaseboer, Institute of High Performance Computing, Singapore

A point force model to describe the time dependent trajectory of the collision and rebound of a rising bubble with an initially flat but deformable fluid-fluid interface as well as a flat solid has been developed. The model encapsulates bubble-scale phenomenon such as the balance between buoyancy and drag forces that determine the terminal velocity. During bubble collision, flow within the thin film, with thickness that may be 3 orders of magnitude smaller than the bubble size that is trapped between the rising bubble and the deformable or rigid surface is treated by lubrication theory and coupled consistently to models for interfacial deformations. Complex small-scale interfacial structures and associated deformations and flow that would otherwise be a very challenging multi-scale multi-phase computational problem can also be treated by this approach. Excellent quantitative agreement with experimental data taken in water and alcohols has been obtained using very modest computational effort.

## DYNAMICS OF BUBBLE GROWTH AND DETACHMENT IN A SHEAR FLOW

Michel Lebon, *Institute of Fluid Mechanics Toulouse, France*; Julien Sebilleau, *INP Toulouse, France*; Catherine Colin, *INP Toulouse, France*

We study the quasi-static growth of an air bubble nucleated on a wall of a rectangular channel. The wall is made of Teflon ensuring a high contact angle. The dynamic of the bubble growth and detachment is recorded with a high-speed camera for different configurations: horizontal channel, inclined channel, with and without a shear flow. From these recordings, several characteristics of the bubble geometry (equivalent radius, foot radius, contact angles, ...) are determined from image processing and allow to quantify the capillary force and the drag and lift forces. A new modeling of the capillary force taking into account the bubble elongation on the wall is proposed. New expressions for the drag and lift coefficients for a deformable bubble attached to the wall are provided. With the new expressions of the forces, the force balance appears very well satisfied during the air bubble growth and can be used to characterize bubble detachment. The detachment occurs for maximum value of the advancing contact angle around 80° and for minimum value of the receding contact angle 20°. As long as the contact angles are in this range (20°-80°), the bubble remains attached to the wall. This modeling is also applied to previous experiments on vapor bubble nucleated on a small heater at the wall of a channel flow and in micro-gravity conditions.

## NUMERICAL SIMULATION OF BUBBLE FORMATION AND DYNAMIC CHARACTERISTICS AT A SUBMERGED MICRONOZZLE

Chen Qu, *Beijing Institute of Technology, China*; Jian Zhang, *Beijing Institute of Technology, China*; Yong Yu, *Beijing Institute of Technology, China*; Jun Hu, *Beijing Institute of Technology, China*

Abstract: This study investigates bubble growth on submerged 0.11mm diameter nozzle through the method of numerical simulation and VOF multiphase model is used in numerical simulation. Numerical simulations are conducted under low gas flow rate conditions (0.25ml/min-0.714ml/min), and detailed characteristics related to bubble growth such as bubble outer contour, bubble detached time and detached volume, the variation of the radius of contact line, the instantaneous contact angle and bubble height are obtained and compared with related literatures [1] and [2]. Then, different working conditions with varied static contact angles and surface tension coefficients are conducted, and related bubble parameters are obtained, and a completed analysis is conducted to explain the variations of related parameters with the relative contribution of buoyancy force, gas momentum flux, surface tension force and gravitational force.

Key words: bubble growth, numerical simulation, detached time, detached volume

## EFFECT OF LIQUID AND COLUMN DIAMETER OF THE FLOW REGIME IN BUBBLE COLUMNS

Josep Escrig Escrig, *University of Nottingham, UK*; Safa Sharaf, *University of Nottingham, UK*; Buddhikha Hewakandamby, *University of Nottingham, UK*; Barry James Azzopardi, *University of Nottingham, UK*

In some cases, it is advantageous to operate bubble columns in the homogeneous regime, i.e., where there is no large-scale mixing and where bubbles essentially retain the size they had at creation. The majority of research in this area has been carried out using water. Experiments have been carried out at Nottingham using spargers with 0.5 mm holes (open area of 0.19%) in columns of 67 and 127 mm diameter. Air and a silicone oil of viscosity 5 mPa s were used as the fluids and the flow was monitored using electrical tomography methods. Time and cross-sectionally resolved phase distributions were captured at 1 kHz for 60 seconds over a range of gas flow rates. From this, both average void fraction and its variation with time as well as bubble size distribution information were extracted. The results showed that with this liquid the homogeneous regime was limited to very low gas flow rates. This is in contrast with equivalent results obtained on the same gas distributor with air-water. The reasons for this are explained using changes in bubble size distributions as well as available literature data.

## HOMOGENEOUS FLOW IN A LARGE SCALE BUBBLE COLUMN: EXPERIMENTAL INVESTIGATION AND CFD MODELLING

Giorgio Besagni, *Politecnico di Milano, Italy*; Thomas Ziegenhein, *Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany*; Dirk Lucas, *Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany*; Fabio Inzoli, *Politecnico di Milano, Italy*

The homogeneous flow regime in a gas-liquid bubble column - 0.24 m inner diameter and 5.3 m height - is studied by using experimental and numerical investigations. The two-phase flow is studied experimentally by using gas holdup measurements, image analysis and optical probes. The holdup measurements are used for studying the column hydrodynamic and for identifying the regime transition. The image analysis and the optical probes are, then, used for quantify the bubble size distributions and the local flow properties for five superficial gas velocities in the homogeneous flow regime. A CFD Eulerian two-fluid modeling of the column operating in the homogeneous flow is proposed by using a customize version of the commercial software ANSYS CFX release 14.5.7. The 3D transient simulations have been performed considering a set of non-drag forces and polydispersity, accordingly with the Baseline formulation proposed by the Helmholtz-Zentrum Dresden-Rossendorf. It is shown that the CFD model predicts the global holdup and the local properties fairly well in the range studied.

## HYDRODYNAMICS OF BUBBLE COLUMNS IN THE HETEROGENEOUS REGIME

Pedro Maximiano Raimundo, *IFP Énergies Nouvelles and Université Grenoble Alpes, France*; Alain Cartellier, *Université Grenoble Alpes and CNRS, France*; Davide Beneventi, *Université Grenoble Alpes and CNRS, France*; Ann Forret, *IFP Énergies Nouvelles, France*; Frederic Augier, *IFP Énergies Nouvelles, France*

Bubble columns are widely used in industry for their mixing and mass transfer capabilities. Despite active research since the 70's, scale-up rules remain poorly understood, and simulations are not yet predictive. To better capture the inherent complexity of gravity driven bubbly flows, experiments were performed in columns 0.15m to 3 m in diameter, at void fractions up to 30-35%, with similar bubble sizes. Various measuring techniques were exploited including a new way to access the horizontal bubble size. The self-similarity of the mean flows in the heterogeneous regime was confirmed. Scaling were also revisited: notably, the liquid flow rate in the upward directed core region grows as  $D^2(gD)^{1/2}$ , i.e. the entrainment capability is set by the column diameter and surprisingly not by the gas flow rate. Besides, the mean relative velocity can be much higher than the terminal velocity. Correct predictions are recovered using an "apparent" drag coefficient that drastically decreases above 20-25% in void. Finally, some experimental evidences indicate that this enhanced relative velocity arises from collective effects that lead to strong concentration fluctuations at meso-scale.

## TWO-PHASE TURBULENCE IN LAMINAR BUBBLY PIPE FLOWS

Hyunmin Park, *Seoul National University, Republic of Korea*; Minki Kim, *Seoul National University, Republic of Korea*; Jun Ho Lee, *Seoul National University, Republic of Korea*

In the present study, the two-phase turbulence in laminar bubbly pipe flows at the Reynolds number of 750, at small-to-moderate void fractions, is experimentally investigated based on two-phase high-speed particle image velocimetry measurements. The bubble-induced liquid velocity is closely connected to the void distribution; i.e., it is accelerated (decelerated) where there are more (less) bubbles accumulated. Despite the small void fractions, the liquid-phase turbulence intensity in vertical and radial directions increases substantially with increasing void fraction, as a power of 0.4. Also, it becomes similar to the single-phase flow turbulence. Based on the investigation of bubble dynamics, on the other hand, we suggest a theoretical model for two-phase Reynolds stress, which includes the contributions of relative bubble rise velocity and void distribution to the generation of Reynolds stress, in addition to the mean liquid velocity gradient. While the present predictions show a good agreement with the measurements, further refinements are discussed to apply the suggested model to turbulent bubbly flows.

\*Supported by the NRF program (NRF-2013R1A1A1008373) of Korean government and Aspiring Researcher Program through Seoul National University in 2014.

## EXPERIMENTAL STUDY OF TRANSIENT FLOW BOILING HEAT TRANSFER UNDER EXPONENTIALLY ESCALATING HEAT INPUTS

Guan-Yu Su, *Massachusetts Institute of Technology, USA*; Matteo Bucci, *Massachusetts Institute of Technology, USA*; Thomas McKrell, *Massachusetts Institute of Technology, USA*; Jacopo Buongiorno, *Massachusetts Institute of Technology, USA*

This paper presents an experimental investigation of transient flow boiling heat transfer phenomena for water at atmospheric pressure under exponentially escalating heat inputs on plate-type heaters. Infrared thermometry and high-speed video were used to obtain high temporal and spatial resolution of wall superheat, heat flux, and vapor formation providing new insights into the physical phenomena. The database generated can be used for development and validation of accurate models of transient boiling heat transfer. Exponential power escalations with periods in the range from 5 to 500 milliseconds, and subcooling of 10, 25 and 75 K were explored. The Reynolds number was varied from 25000 to 60000, depending on the subcooling. Single-phase heat transfer, onset of the boiling driven (OBD) heat transfer regime and overshoot (OV) conditions were identified. The experimental data showed that during the single phase heat transfer regime, forced convection is the dominant heat transfer mechanism for long periods, whereas transient conduction is more important for short periods. To predict the single-phase heat transfer coefficient, a unique model was developed based on such competition between forced convection and transient conduction in the near surface fluid. For a given subcooling, the OBD and OV temperatures and the OBD heat flux increase with increasing Reynolds number. The same trend is observed for increasing subcooling at a given Reynolds number.

\* Support from CEA is gratefully acknowledged.

## LAGRANGIAN TRACKING OF DROPLETS EVAPORATING IN A HI TURBULENCE BY USING IN-LINE DIGITAL HOLOGRAPHY

Jean-Louis Marie, *CNRS, France*; Thibaut Tronchin, *Ecole Centrale de Lyon, France*; Nathalie Grosjean, *CNRS, France*; Loic Mees, *CNRS, France*; Michel Lance, *LMFA, France*

The evaporation of diethyl ether droplets dispersing in a HI turbulence is measured by following droplets along their trajectory. Measurements are performed at ambient temperature and pressure by using in-line digital holography. The holograms of droplets are recorded with a single high-speed camera (3kHz) and reconstructed with an "inverse problems approach" algorithm previously tested in Chareyron et al. *New J Phys* 14:43039 (2012) and Marié et al. *Exp in Fluids* 55:1708 (2014). The thermal-vapor concentration wakes developing around the droplets are visible behind each hologram. It is found that these wakes are aligned with the relative Lagrangian velocity seen by droplets at each instant. This relative velocity can be calculated from the dynamic equation of droplets motion and the positions and diameter of the droplets measured by holography. Sequences of the time evolution of droplets 3D positions, diameter, 3D relative velocity are presented. In a number of cases, the evaporation rate of the droplets changes along the trajectory and deviates from that calculated with a standard film model of evaporation. This shows that turbulence may significantly influence the phase change.

\* Supports from ANR program TEC2 and MORIN (ANR-11-IDEX-0007) are gratefully acknowledged.

## CFD SIMULATION OF CAVITATION IN FUEL NOZZLES

Alvaro Bretones, *Universidade Federal de Uberlândia, Brazil*; Francisco Jose De Souza, *Federal University of Uberlândia, Brazil*

Nozzle cavitation is a common phenomenon with relevant consequences on the fuel spray formation. It has been shown that simpler models, based on thermodynamics, have limitations for predicting cavitation. On the other hand, models based on fluid mechanics that account for non-equilibrium situations have the potential to provide more realistic predictions of the complex phenomena involved. In this work, the performance of a Rayleigh-Plesset VOF model is evaluated based on experimental fuel injector data. The model parameters are initially calibrated based on quasi-2D throttle experiments. Subsequently, simulations of realistic 3D fuel injectors are run, and the results are compared with the respective computer tomography (CT) and laser Doppler velocity (LDV) data. The cavitation structures and respective mass flows are compared and display reasonable agreement. Some of the discrepancies observed are attributed to bubble break-up, which is not accounted for in these simulations, although known to take place in such nozzles. It can be concluded that, despite their computational cost, CFD-based cavitation models offer advantages with respect to the phenomenon understanding and prediction.

## VORTEX STRUCTURES OF LOW STEAM QUALITY BOILING TWO-PHASE FLOW IN BUBBLE-TYPE PLATE CHANNEL

Chen Jiang, *Xi'an Jiaotong University, China*; Zonglin Yang, *Xi'an Jiaotong University, China*; Bofeng Bai, *Xi'an Jiaotong University, China*

Bubble-type plate heat exchanger with approximate semi-elliptical bubble is a new type plate heat exchanger which has many special advantages such as more uniform flow, lower pressure loss, less deposition and fouling. Due to the complexity of the channel structure and the lack of research, the characteristics and mechanism such as the vortex structures of the two-phase flow boiling in bubble-type plate channel are still not clear. In this article, a two fluid model of flow boiling in complex channel was built to investigate the distribution of velocity, pressure, vortexes and temperature of a boiling two-phase flow in bubble-type plate channel. Under the low heat flux, the heat is mainly transferred through the forced two-phase convection with disappeared and regenerated vortex. The blocking effect and flow separation of inward convex and horizontal vortex flow forms in outward convex increase the pressure loss. The boiling heat transfer coefficient increases dramatically with the increase of mass flow rate, which has greater effect on the coefficient than the heat flux. In addition, the pressure drop of boiling two-phase flow has positive correlation with the mass flow rate, while the effect of the heat flux on the pressure drop of flow boiling is little.

## EXPERIMENTALLY STUDYING OF THE EFFECT OF THE HEATING SURFACE MORPHOLOGY ON CHARACTERISTICS OF HIGHLY SUBCOOLED WATER BOILING

Nikolay Vasiliev, *Joint Institute for High Temperatures, Russian Federation*; Konstantin Khodakov, *Joint Institute for High Temperatures, Russian Federation*; Aleksei Varaksin, *Joint Institute for High Temperatures, Russian Federation*; Yuriy Zeigarnik, *Joint Institute for High Temperatures, Russian Federation*

The results of the experimentally studying of characteristics of highly subcooled water boiling are presented. The experiments were conducted under forced flow conditions in rectangular channel on flat surfaces: smooth, rough, and artificially coated by nanoparticles. The tests were carried out within the range of the fluid mass flow rate up to 800 kg/(m<sup>2</sup> s) water subcooling relative to saturation temperature 25-70°C, and heat flux density values from 1-2 MW/m<sup>2</sup> to those close to burnout. Using high-speed filming with a frequency up to 100 kHz the effect of the heating surface morphology on subcooled boiling characteristics was studied. This concerned size distribution of vapor bubbles, bubble life time, and duration of different studies of vapor evolution, spatial vapor distribution and density of nucleation sites population. It was recognized that certain phenomena typical of highly subcooled water boiling are retained at the boiling on surfaces with complex morphology (chaotic activation of nucleation sites) whereas the other characteristics changed (prevailing bubble dimensions at given heat flux, etc.)

\* The work was supported by the Russian Foundation of Basic Research, Grant No. 14-08-00740

## EFFECTS OF MASS TRANSFER ON THE OSCILLATIONS OF CAVITATION BUBBLES

Yuning Zhang, *North China Electric Power University, China*; Yuhang Gao, *North China Electric Power University, China*; Xiaozhe Du, *North China Electric Power University, China*

Cavitation is an essential topic of multiphase flow with many applications in fluid, chemical, biomedical engineering. In the present paper, effects of mass transfer on the oscillations of cavitation bubbles are numerically investigated in great details. Many important physical processes (e.g. heat and mass transfers across the bubble interfaces) involved during the dynamic oscillations of cavitation bubbles are incorporated into the present work. Liquid compressibility, surface tension and effects of non-condensable gas are also considered for completeness. Data analysis are performed in a large parameter zone including Peclet number and Sherwood number. Predictions of pure gas bubbles in the literature are shown for comparisons. Based on the data analysis, the regions dominated by mass transfer during the oscillations of cavitation bubbles are identified in terms of non-dimensional parameters. Physical mechanisms relating with the effects of mass transfer are also identified and discussed with the demonstrating examples. Comparing with gas bubbles, the effects of mass transfer on cavitation bubbles have many unique characteristics (e.g. in terms of time-scale). The importance of the present work on building a more sophisticated cavitation model is briefly discussed.

\*This work was financially supported by the National Natural Science Foundation of China (Project No.: 51506051) and the National Basic Research Program of China (973 Program) (No. 2015CB251503).

#### SYNCHROTRON X-RAY PHASE CONTRAST IMAGING OF CAVITATION IN FUEL INJECTOR NOZZLES WITH VARIOUS SIZES

Akira Sou, Kobe University, Japan; Rubby Prasetya, Kobe University, Japan; Seoksu Moon, Nat'l Inst of Advanced Industrial Science and Technology, Japan; Yoshitaka Wada, MAZDA Motor Corporation, Japan; Hideaki Yokohata, MAZDA Motor Corporation, Japan

Cavitation in a fuel injector for Diesel engines plays an important role in the characteristics of a discharged liquid jet and spray. Since images of cavitation in real-scale tiny nozzle are not clear, cavitation in large-scale nozzles have been visualized. In this study phase-contrast X-ray imaging of cavitation in two-dimensional nozzles with various widths of 4, 2, 1, 0.5 and 0.25 mm is conducted. As a result, we find that (1) the normalized cavitation width and length do not depend on nozzle size, (2) cavitation tends to be slightly thinner and, therefore, shorter in small nozzles due to the relatively large normalized radii of curvatures of the nozzle inlet edges, and (3) the shapes of cavitation bubbles are not always spherical, and the sizes of the cavitation bubbles are in proportional to on nozzle size.

#### TWO-PHASE FLOW SIMULATIONS OF THE INTER-BLADE VORTICES IN A FRANCIS TURBINE

Jonas Wack, University of Stuttgart, Germany; Stefan Riedelbauch, University of Stuttgart, Germany; Keita Yamamoto, EPFL, Switzerland; Francois Avellan, EPFL, Switzerland

Due to the extension of the operating range, Francis turbines are operating more and more at off-design conditions that are accompanied by cavitation. In the present study, the cavitation phenomena at deep part load conditions are investigated using two-phase numerical simulations. This operating point is characterized by the occurrence of inter-blade vortices in the runner. The pressure minimum in the vortex core might lead to cavitation requiring suitable turbulence models to properly capture those phenomena by numerical simulations. In a thorough study, the ability of the standard k- $\omega$ -SST and the SAS turbulence model, which is capable to resolve parts of the turbulence spectrum, to capture the inter-blade vortices is investigated. Comprehensive measurement data in the wicket gate, runner and draft tube are available and can be compared with the simulation results. The stochastic nature of this far off-design operating point is investigated at the locations of different pressure probes using a Fast Fourier Transformation (FFT). Furthermore, different visualization techniques have been performed in the measurements that enable to investigate the shape and dynamical behavior of the simulated cavitating vortices.

\* Support from ERC/FP7-ENERGY-2013-1-Grant 608532 is gratefully acknowledged.

#### A MATHEMATICAL ANALYSIS OF PARTICLE CLUSTERING IN TURBULENT FLOWS

Mahdi Esmaily Moghadam, Stanford University, USA; Ali Mani, Stanford University, USA

Inertial particles, characterized by their Stokes number, in interaction with turbulent flows form clusters. Experimental and numerical studies have shown that this clustering becomes significant at Stokes numbers near unity and disappears at very small and large Stokes numbers. However, it has yet to be found an analytically solution that characterizes particle clustering at the full range of Stokes numbers. To fill this gap, the present study offers a mathematical solution for prediction of particle clustering. In our analysis we consider fluid flow as simultaneous superposition of harmonic modes. Particle equations are analyzed in a Lagrangian frame and confined to a small cloud of particles, allowing representation of the flow field in terms of Taylor series. We show the size of this cloud can exponentially contract in time by interacting with flow harmonics, forming a concentrated cluster of particles. To verify our analysis, we performed direct numerical simulations of particle-laden isotropic turbulence over a wide range of Stokes and Reynolds numbers. The comparison of these results shows our analysis correctly predicts a Stokes number at which maximum clustering occurs as well as the rate of decrease in clustering at smaller and larger Stokes numbers.

\* Support from United States Department of Energy is gratefully acknowledged.

#### RAPID STRETCHING OF NEWTONIAN AND NON-NEWTONIAN LIQUID BRIDGES

Ilia V. Roisman, Technische University Darmstadt, Germany; Christina Weickgenannt, , Germany; Cameron Tropea, Technische University Darmstadt, Germany

In this experimental and theoretical study a rapid stretching of Newtonian and non-Newtonian liquid bridges, formed between two solid plates, is considered. One of the plates is fixed, while translation of another plate in the experiments is accurately controlled. Its constant acceleration ranges from 10 to 200 m/s<sup>2</sup>. The phenomena, associated with fast liquid bridge stretching are associated with the technology of gravure printing, car soiling, agglomeration of wet granular materials, ice accretion due to ice crystal impact, etc.

In our experiments the instant of the liquid bridge pinch-off is determined for various plate accelerations, liquid bridge volumes, initial gap thickness. The major role of the liquid rheology (in the case of the Newtonian liquid – viscosity) is demonstrated. A theoretical model for the breakup time is developed. Its predictions agree well with the experimental data.

Additionally, the numbers and the diameters of the liquid drops, formed by breakup, the volume of the residual liquid on the fixed and moved plates, the breakup length, are measured and modelled.

It has been shown that the breakup time of suspensions is similar to that of Newtonian liquids, but is always smaller. It indicates that presence of the solid particles in the liquid promotes breakup.

\* Support from Center of Smart Interfaces (Technische Universität Darmstadt) is gratefully acknowledged.

#### JOINT INVESTIGATION OF CLUSTERING AND SETTLING OF HEAVY SUB-KOLMOGOROV PARTICLES IN TURBULENT FLOWS

Sholpan Sumbekova, UGA, France; Alberto Aliseda, University of Washington, USA; Mickael Bourgoïn, CNRS, France; Alain Cartellier, Université Grenoble Alpes and CNRS, France

Particle-laden flows or flows with inertial particles are of a great interest of current research. Given different density and finite size of the inertial particles, they tend to form high concentration fields i.e. clusters in the regions of the flow with high strain rate and low concentration fields i.e. voids in the regions of high vorticity. Settling velocity of such particles is believed to be enhanced due to presence of velocity of clusters (Aliseda et al). In this work we present joint investigation of the clustering phenomenon and settling regime of the heavy sub-Kolmogorov particles in the turbulent flows with  $Re_\lambda = 200 - 500$ . Experimental techniques (PTV and PDI) used allow to obtain simultaneous information about velocity, size and particle's local concentration field. Results show that the level of clustering is maximum for both  $Re_\lambda = 200$  & Stokes number  $< 1$ , reaches a minimum at  $Re_\lambda = 300$  &  $1 < \text{Stokes number} < 2.5$  and increases again for  $Re_\lambda = 400 - 500$  & Stokes number  $> 2.5$ . The PTV data yield that both level of clustering and settling velocity depend on the volume fraction and Stokes number in non-trivial way. Clusters are found to be settling faster than voids. Presence of flow's drift velocity results in a saturation of settling velocity results at  $Re_\lambda = 400-500$ .

#### CLUSTERING, INTERMITTENCY OF RELATIVE VELOCITY AND CAUSTICS OF HEAVY PARTICLES UNDER GRAVITY IN ISOTROPIC TURBULENT FLOWS

Guodong Jin, Institute of Mechanics, Chinese Academy of Sciences, China; Jincai Chen, Institute of Mechanics, Chinese Academy of Sciences, China; Guo-Wei He, Institute of Mechanics, Chinese Academy of Sciences, China

Particle-laden turbulent flows widely exist in natural and industrial flows. Clustering and collision rates between particles affect the physical and chemical processes in these flows. For example, the rain droplet formations in cloud are accelerated by turbulence. The interactions of turbulence, particle inertia and gravity can change their statistical characteristics such as clustering, intermittency of relative velocity and caustics. Here, we will show that gravity suppresses clustering at small Stokes numbers while enhances clustering at large ones, and attribute these changes to the reduction in the timescale of the velocity gradient experienced by a nearby particle pair. We will then study the intermittency of relative velocity of heavy particles from the point view of backward-in-time separation and relate the current relative velocity of particles to the lengthscale and intermittency of turbulent motions experienced by particles during past  $3\tau_p$  ( $\tau_p$  is particle relaxation time). We find that gravity reduces the intermittency at small Stokes numbers and increases intermittency at large ones. The rates of caustic formation under gravity are studied using the Full-Lagrangian-Method by tracking evolution of the deformation tensor of a reference particle along its trajectory.

## NUMERICAL SIMULATION OF THREE-DIMENSIONAL MULTIPHASE FLOWS WITH ADAPTIVE UNSTRUCTURED MESHES

Zhihua Xie, Imperial College London, UK; Dimitrios Pavlidis, Imperial College London, UK; Zhizhao Che, Imperial College London, UK; Pablo Salinas, Imperial College London, UK; Christopher Pain, Imperial College London, UK; Omar K. Matar, Imperial College London, UK

Multiphase flows where two or more fluids have interfacial surfaces are often found in industrial and practical engineering applications, including bubbles, droplets, liquid film and waves. An adaptive unstructured mesh modelling framework is employed here to study three-dimensional (3D) two- and three-phase flow problems, which can modify and adapt unstructured meshes to better represent the underlying physics of multiphase problems and reduce computational effort without sacrificing accuracy. The numerical framework consists of a mixed control volume and finite element formulation, a 'volume of fluid' type method for the interface capturing based on a compressive control volume advection method and a force-balanced algorithm for the surface tension implementation. Numerical examples of multiphase flow problems, such as 3D falling liquid films in a long domain, 3D droplet impact on a flowing film, and three-phase flows are presented to demonstrate the ability of this method.

\* Support from EPSRC MEMPHIS multiphase Programme Grant (EP/K003976/1) and UK-Japan Civil Nuclear Energy project (EP/M012794/1) is gratefully acknowledged.

## DISCONTINUOUS GALERKIN FINITE ELEMENT METHOD APPLIED TO THE COUPLED NAVIER-STOKES/CAHN-HILLIARD EQUATIONS

Franck Pigeonneau, UMR 125 CNRS/Saint-Gobain, France; Pierre Saramito, CNRS and Grenoble university, France

To study numerically two-phase flows driven by the interfacial dynamics, a phase field model is chosen to tract implicitly interfaces. The phase field obeys a Cahn-Hilliard equation with convection. The fluid dynamics is described with the Navier-Stokes equations with an additional source term in the momentum equation to take into account the capillary forces. A discontinuous Galerkin finite element approximation is used to solve the coupled Navier-Stokes/Cahn-Hilliard equations. The Cahn-Hilliard equation is treated as a system of two coupled equations corresponding to the advection-diffusion equation for the phase field and an elliptic equation for the chemical potential (variational derivative of the free energy).

First, the weak formulation of the Cahn-Hilliard equation is presented for which the consistency is recalled as well as the error estimate. The accuracy of the numerical method is tested in various test cases such as the relaxation of an elliptic or a cross with and without convection. Second, the weak formulation of the Navier-Stokes equations is recalled in which the same space of approximation is used for the velocity and the pressure with an adequate stabilization technique. Finally, we will present numerical simulations to describe the capillary rising in a tube and the dynamics of the motion of contact line in a simple shear flow.

## FRAGMENTATION OF A POROUS VISCOELASTIC LIQUID BY RAPID DECOMPRESSION: IMPLICATION TO VOLCANIC ERUPTION

Masaharu Kameda, Tokyo University of Agriculture and Technology, Japan; Mie Ichihara, The University of Tokyo, Japan

Fragmentation of vesicular magma is a key phenomenon controlling volcanic eruptions. Solid-like brittle fragmentation due to rapid decompression is a trigger of explosive eruption. The fragmentation of magma, which is a viscoelastic fluid, occurs through a combination of viscoelasticity and rapid deformation. We conducted a rapid decompression experiment in order to clarify the viscoelastic effect on the fragmentation using a magma analogue, syrup containing gas bubbles. Through the experiment we demonstrated the existence of a transitional fragmentation behavior, which we refer to as brittle-like fragmentation, occurred even if the response of material should be in a ductile manner. Comparing the realistic decompression time with the viscoelastic relaxation time for magma, it is more probable that the fragmentation in the real volcanic system occurs in a brittle-like manner. We observed the interior of the specimen by synchrotron X-ray tomographic microscopy. The X-ray imaging was performed to observe initial structure of the specimen by three-dimensional tomography as well as its dynamic response by high-speed radiography. The results indicate that the crack is initiated from the interior of the specimen due to non-uniform spatial distribution of bubbles. Furthermore, we found that the fracture may occur when the inter-distance of neighboring bubbles is close to the order of the bubble radius, even if the rheological bulk properties of the specimen is close to the fluid state.

\* Support from JSPS grant in aid for scientific research KAKENHI No. 26630047 is gratefully acknowledged.

## DIFFUSE INTERFACE SIMULATIONS OF PHASE SEPARATION OF SHEARED BINARY MIXTURES USING A FINITE VOLUME SOLVER

Roberto Mauri, University of Pisa, Italy; Giuseppe Di Vitantonio, Università di Pisa, Italy; Chiara Galletti, Università di Pisa, Italy; Andrea Lamorgese, University of Pisa, Italy

Multiphase flows are generally modeled assuming that the different phases are separated by an interface of zero thickness, hence all physical properties are allowed to change discontinuously across the interface. This may be an issue when the length-scales of the systems are comparable to the real interface thickness, as in presence of micro-devices or when treating droplet break up. Conversely, the "diffuse interface" model or "phase field theory" assumes that the interfaces are diffuse; hence the model is naturally able to adapt to a large variety of multiphase problems, also in presence of a change of flow morphology, as for instance: physical phenomena of binary mixtures (mixing, spinodal decomposition and nucleation of macroscopically quiescent regular mixtures); droplet dynamics (deformation, coalescence and break-up, moving contact angles) as well as structure development of polymer blends.

So far the diffuse interface model has been developed using in-house spectral or finite element codes and simple geometries (boxes, channels...), and this has partly limited its applicability to cases of practical interest. The present work illustrates the implementation of the diffuse interface model in a finite volume solver able to deal with unstructured grids and thus more complex geometries, with the objective of enlarging the range of applicability of the model. The implementation is first validated using simple reference cases and then applied to model the phase separation of sheared binary mixtures.

## FAST SIMULATIONS OF RECURRENT MULTIPHASE FLOWS' LONG-TIME BEHAVIOUR

Thomas Lichtenegger, Johannes Kepler University, Austria; Stefan Pirker, Johannes Kepler University, Austria; Gerhard Holzinger, Johannes Kepler University, Austria

We present a novel, general technique to efficiently describe the long-time behavior of (particulate) multiphase flows characterized by recurring features, e.g. fluidized beds. Using the results of a full-CFD / full-CFD-DEM simulation of the system over some pseudo-periods, a recurrence plot that compares flow fields at two times is generated. We go beyond the well-known utilization of recurrence statistics for pure data analysis and employ it to generate a series of flow patterns which are a good approximation to the real process. This sequence of arbitrary length – in particular magnitudes of order longer than the initially simulated time – can be used to describe processes which are very slow in comparison to the short-time dynamics of the system, e.g. chemical reactions or heat transfer.

Even in the simplest implementations, the computational separation of these very different time scales leads to a significant speed-up in comparison with ordinary CFD / CFD-DEM simulations. We demonstrate our approach with the cooling of a hot particle bed fluidized with cold gas.

An outlook how to further improve efficiency and extend the scope of our method is given, current weaknesses and possible limitations are discussed.

## INFLUENCE OF ANISOTROPIC PERMEABILITY ON CONVECTION IN POROUS MEDIA

Marco De Paoli, University of Udine, Italy; Francesco Zonta, University of Udine, Italy; Alfredo Soldati, University of Udine, Italy

We use direct numerical simulation to examine the role of non-isotropic permeability on Rayleigh-Benard convection in a fluid-saturated porous medium. With this conceptual set-up we aim at mimicking the process of CO<sub>2</sub> sequestration into sedimentary rock reservoirs.

Simulations up to Rayleigh numbers of  $5 \times 10^4$  confirm previously found linear scaling of Nusselt number,  $Nu \sim Ra$ , which was obtained for isotropic media. Such linear scaling is maintained also for non-isotropic media with vertical-to-horizontal permeability ratios,  $\gamma$ , smaller than unity, typical of sedimentary rock reservoirs.

However, and contrary to intuition, we find that for  $\gamma < 1$ , Nusselt number increases significantly compared to the ideal case of isotropic media, indicating that in real reservoirs CO<sub>2</sub> dumping times can be much shorter than current predictions.

\* CINECA 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.

## SURFACING AND CLUSTERING OF GYROTACTIC SWIMMERS IN FREE-SURFACE TURBULENCE

Salvatore Lovecchio, Institut de Mécanique des Fluides de Toulouse, France; Cristian Marchioli, University of Udine, Italy; Gaetano Sardina, Royal Institute of Technology, Sweden; Luca Brandt, KTH Royal Institute of Technology, Sweden; Alfredo Soldati, University of Udine, Italy

Motile micro-organisms have developed different ways of adapting to aquatic flows. Some organisms are able to swim upward and reach fluid layers below the air-water interface where light is enough to activate photosynthesis, absorb CO<sub>2</sub> and produce oxygen. Although very simple, these organisms (called gyrotactic swimmers) have the center of mass below the center of buoyancy and therefore know the direction of gravity. Gyrotactic swimmers are almost neutrally buoyant and are strongly influenced by the local flow conditions: Velocity and vorticity can be such to facilitate or prevent the swimmers' natural rising motion. Knowing the rate at which swimmers reach water surface is important to estimate global CO<sub>2</sub> exchange across the air-water interface of large water bodies. In this work, we use DNS-based Euler-Lagrangian simulations to investigate the motion of gyrotactic swimmers in their final stretch towards the surface. Swimmers with different re-orientation times, reflecting the ability to react to turbulent fluctuations, are considered. We show, for the reference case of turbulent open channel flow, how these organisms can orient during surfacing and where they cluster once at the surface. Results show that vertical migration and surface clustering depend on the re-orientation time, which impacts on the stability of swimmers in the vertical direction. We also show that the regions where swimmers collect depend on the topology of surface turbulence.

\* Support from CINECA is gratefully acknowledged.

## LOCAL CONCENTRATION MEASUREMENTS IN THE WAKE OF BUBBLES BASED ON IN-SITU RAMAN SPECTROSCOPY AND STATISTICAL ANALYSIS

Daniela Schurr, Karlsruhe Institute of Technology (KIT), Germany; Jainabalkya Guhathakurta, University of Stuttgart, Germany; Yousef Baroud, University of Stuttgart, Germany; Sven Simon, University of Stuttgart, Germany; Gunter Rinke, Karlsruhe Institute of Technology (KIT), Germany; Roland Dittmeyer, Karlsruhe Institute of Technology (KIT), Germany

The concentration of chemical compounds in the wake of rising or moving bubbles due to physical or chemical absorption is of general interest in terms of industrial gas-liquid reactions, especially in bubble columns. A frequently used tool for concentration measurement is Raman spectroscopy. In order to measure the concentration locally resolved in the wake of moving bubbles, a novel spatially and temporally resolved Raman analysis setup is proposed in this paper. The spatial and temporal resolution is achieved using a high energy laser focused to a local spot to excite the Raman stray light with a short pulse duration in the microsecond range. Such high energy pulse lasers have a repetition rate of a few Hz, which is too slow to analyze the concentrations of the wake of a single bubble. However, the repeated analysis of bubbles at different positions provides a holistic view on the spatially resolved concentration field around a bubble. This requires the real-time computation of the bubble position for the statistical analysis of a plurality of bubbles using the presented Real-Time Raman Process Analysis System based on a dedicated high-speed camera with integrated real-time image processing.

\*We gratefully acknowledge the support by the German Research Foundation (DFG) within the Priority Program 1740 "Reactive Bubbly Flows".

## THE ROLE OF INLET FLOW INSTABILITIES IN LIQUID-LIQUID FLOW

Rhys Morgan, Forsys Subsea, UK; Christos N. Markides, Imperial College London, UK; Ivan Zadrazil, Imperial College London, UK

Horizontal flows of two immiscible liquids with matched refractive indices have been investigated using high-speed simultaneous Planar Laser Induced Fluorescence (PLIF), Particle Image Velocimetry (PIV) and Particle Tracking Velocimetry (PTV). The test section at which the laser-based optical diagnostic methods were applied was far downstream from the inlet to ensure sufficient development length; optical distortion due to the curvature of the (transparent) circular tube test section were corrected with the use of a graticule technique. The current study extends the database of liquid-liquid flows obtained using the aforementioned laser-based techniques by injecting the aqueous phase at the top of the channel and the oil phase at the bottom (an "inverted" inlet orientation). The main focus has been an evaluation of the effects of interfacial instabilities on flow patterns and multiphase flow parameters in comparison with our previous work in which the heavier phase was injected at the bottom of the channel and the lighter phase at the top of the channel. This change in inlet orientation imposes a Rayleigh-Taylor instability at the inlet; the effects of this instability have been shown to persist along the tube, increasing the propensity for oil droplets below the interface.

## CRUDE OIL PLUMES IN CROSSFLOW: EFFECT OF TURBULENT FLOW STRUCTURE ON OIL RESIDENCE IN PLUME

David Murphy, Johns Hopkins University, USA; Xinzhi Xue, Johns Hopkins University, USA; Kaushik Sampath, Johns Hopkins University, USA; Joseph Katz, Johns Hopkins University, USA

Oil well blowouts at sea create buoyant, immiscible jets and plumes of crude oil which are transported by ocean currents. In this experimental study, turbulent crude oil jets (2.5 m/s, Re=1100) are injected vertically in a towing tank of artificial seawater while being towed at 0.15 m/s to simulate crossflow. The jet Ohnesorge (Oh) number is altered by premixing dispersant (Corexit 9500A) with the oil at various dispersant to oil (DOR) ratios, which reduces the oil-seawater interfacial tension by up to two orders of magnitude. High speed visualizations observe the temporal and spatial development of the oil plume, and turbulent flow structures are measured using Particle Image Velocimetry. A submerged holographic microscopy system measures droplet size distribution (DSD) over time. Dispersant substantially changes the plume shape and DSD. Without dispersant, mm size droplets rise rapidly while smaller droplets remain trapped within a counter-rotating vortex pair (CVP) that develops as the plume bends horizontally. Dispersants cause a drastic reduction in droplet size and rise rate, thus increasing trapping by the CVP and narrowing oil plume width. Droplet residence within the CVP and the time evolution of DSD are linked to the vortex strength via a 'trapping function' derived from a dynamic force balance on buoyant droplets subjected to a vortex-induced pressure field.

\* Support from Gulf of Mexico Research Initiative is gratefully acknowledged.

## CONDITIONAL SAMPLING FOR THE ANALYSIS OF TWO-PHASE FLOW PIV/PTV EXPERIMENTAL IMAGES

Carlos E. Estrada Perez, Texas A&M University, USA; Junsoo Yoo, Idaho National Laboratory, USA; Yassin A. Hassan, Texas A&M University, USA

Visualization techniques have been a useful tool for the study of multiphase flow systems. The applicability of these techniques is of particular importance for liquid-gas systems on which the gas phase is the dilute phase. PIV/PTV are visualization techniques used to determine quantitatively the individual phases turbulence parameters, but also they can provide a qualitative perspective impossible to get directly from point measurement probes. This work is an attempt to exploit the added advantage of a high spatial resolution provided by visualization techniques. In this work, Particle Tracking Velocimetry (PTV) and conditional sampling were used to investigate the local "average" influence that a "characteristic" bubble has on the liquid turbulence surrounding it. Analogous to single-phase turbulence, the results of the conditional sampling approach can be described with three terms, a local average term due to the "characteristic" bubble influence, and two fluctuating terms representing the fluctuations induced by neighboring bubbles and by the liquid turbulence which will be present without the existence of the bubbles. It is expected that this approach can provide both: global averages and detailed local information only available with visualization techniques. Furthermore, this technique can help improve turbulent two-phase flow models which are based on single bubble or low void fractions measurements.

## DEVELOPMENT OF MEASUREMENT TECHNIQUE FOR BUBBLE VELOCITY VECTOR IN GAS-LIQUID TWO-PHASE FLOW BY USING 4-SENSOR PROBE

Yasushi Saito, Kyoto University Research Reactor Institute, Japan; Gen Ariyoshi, Graduate School of Energy Science, Kyoto University, Japan; Daisuke Ito, Kyoto University Research Reactor Institute, Japan

Precise modeling for two-phase flow requires a precise measurement technique which can obtain the phase distribution and bubble behavior with high accuracy. Various measurement algorithms for multi-sensor probe have been proposed so far to measure void fraction and interfacial area concentration. However, such algorithms are not compared well with uniform way. In this study, a new measurement method was proposed for 4-sensor probe. Its measurement error was compared numerically/experimentally with that by other existing measurement techniques. The new measurement method can measure void fraction, interfacial area concentration and bubble velocity vector by taking curvature effect of each bubble interface into account. Finally the proposed method was applied to air-water two-phase flow and gas-liquid metal two-phase flow experimentally.



## A LARGE SCALE MULTI-FLUID/DISPERSED PHASE APPROACH FOR SPRAY GENERATION IN AERONAUTICAL FUEL INJECTORS

Daive Zuzio, ONERA, France; Jean-Luc Estivalezes, ONERA, France; Ghislain Blanchard, ONERA, France

Atomization is usually neglected in industrial numerical simulations of aeronautical combustors, despite the increasing physical fidelity of LES approaches. The reason comes from the strong multi-scale nature of the problem: the typical space and time lengths are too small for their resolution to be afforded. The influence of the atomization process should however be taken in account when simulating unsteady processes like combustion instabilities. This work proposes a large scale numerical methodology (LSS, "Large Scale Simulation") to simulate the largest scales of the unsteady assisted atomization process to be integrated in industrial simulations of reactive combustion chamber. The LSS approach involves a multi-fluid resolution for the first stages of atomization, like primary oscillation of the liquid fuel, with a diffuse interface model. Subsequently, the unresolved liquid atomized phase is locally transferred to a dispersed phase solver in a fully coupled way. The coupling is supposed to take action when the interface becomes too smeared in respect to the local mesh, and the multi-fluid description of the liquid structures becomes unreliable. The atomization model acts as a volume source term for the multi-fluid solver removing liquid mass (like a chemical source term). On the other hand, it becomes an advanced adaptive injector for the dispersed phase solver. The methodology has been tested on two atomization configurations, a planar sheared liquid sheet and a round liquid jet in a non turbulent cross-flow.

## NOZZLE TIP WETTING OF MULTI-HOLE GASOLINE INJECTORS WITH VARIOUS NOZZLE OUTLET CONFIGURATIONS UNDER FLASHING AND NON-FLASHING CONDITIONS

Chansoo Park, Republic of Korea Advanced Institute of Science and Technology, Republic of Korea; Jinyoung Jung, Republic of Korea Advanced Institute of Science and Technology, Republic of Korea; Choongsik Bae, Republic of Korea Advanced Institute of Science and Technology, Republic of Korea; Heechang Oh, Hyundai Motors Company, South Korea; Juhun Lee, Hyundai Motors Company, South Korea

In the automotive industry, to meet the future emission regulations, various efforts to reduce particulate matter emission are being dedicated especially to the fuel injector. Injector nozzle tip wetting is one of emerging issues in gasoline direct injection engines. Wetted fuel on the surface of injector tip has been pointed out as a major source of particulate emission and injector tip coking. In this study, the degree of injector tip wetting of various injector nozzle outlet configuration under flashing and non-flashing conditions of gasoline fuel was evaluated. In order to measure the quantity of wetted fuel, LIF (laser induced fluorescence) technique was utilized. 266 nm wave length Nd:Yag laser was used as an excitation source and emission light was captured by using long distance microscope and intensified CCD (charge coupled device) camera. Backlight imaging was also used to explain the reason of the nozzle tip wetting. The result showed that smaller diameter nozzle outlet configuration reduced fuel wetting and increasingly widening shaped nozzle outlet increased fuel wetting. It was hard to conclude the result of increasingly narrowing shape cases, but it showed the potential of fuel wetting reduction. Flashing condition results showed increased injector tip wetting compared to non-flashing condition.

## INTERFACE TREATING TECHNIQUE FOR COMPRESSIBLE MULTI-MEDIUM FLOWS

Boo Cheong Khoo, National University of Singapore, Singapore; Hong Chen, Harbin Engineering University, China; Weibing Zhu, Harbin Engineering University, China

The Ghost Fluid Method and its variants (GFM) have encountered several difficulties when applied to the gas-water problem with a strong shock impacting and loading due to the complex breakup of the interface. We present a new and yet simple interface treating technique for the compressible multi-medium flow, which can easily solve and track faithfully the breakup of the gas-gas interface and gas-water interface in the presence of shock wave. In the mixed cells, the GFMs serve to construct the Riemann problem in order to get the states on both sides of the interface. We extend the HLLC-HLL with weighted average flux scheme which is used for strong shock wave [1] originally intended for single fluid medium to the multi-medium flow. Numerical examples show that the proposed method is robust for one and two-dimensional problems. In particular, this method can capture well the complex flow features of high density ratio multi-medium shock-bubble interaction problem.

Keywords:

Compressible multi-medium flows, Interface treating method, Level set method, Ghost fluid method, HLLC-HLL

Reference:

[1] Kim S D, Lee B J, Lee H J, et al. Robust HLLC Riemann solver with weighted average flux scheme for strong shock. Journal of Computational Physics, 2009, 228(20): 7634-7642.

## SIMULATING THE INTERACTION OF SPRAY AND SURFACE FILM FLOW WITH COMPLEX GEOMETRIES IN FIRE SUPPRESSION APPLICATIONS

Karl V. M. Meredith, FM Global, USA; Kristin Jamison, FM Global, USA; Jaap De Vries, FM Global, USA; Xiangyang Zhou, FM Global, USA; Yi Wang, FM Global, USA

Wetting the surfaces of a burning solid is the key to obtaining suppression in most industrial fire scenarios. The fire sprinkler is the main mechanism for applying water in the presence of a fire. The effectiveness of a sprinkler in wetting a surface depends on the injection pattern of the spray, the proximity and orientation of the sprinkler with the surface, and on the surface geometry and properties. These effects have been simulated in the fire suppression CFD code, FireFOAM using a Lagrangian particle method for the spray and an implementation of the thin-film equations for the surface flow. Injection models for three sprinklers have been developed based on experimental characterization of the near-field spray. These injection models, coupled with thin-film flow modeling, provide predictive capability for water coverage and flow rates over a series of standardized geometrical configurations. The three geometrical configurations in this study consisted of palletized commodities arranged in a rack-storage array. Validation of the water flow to the base of the array has been performed against measured experimental values, showing good agreement. In order to minimize water demands and installation requirements, the validated model was used to optimize sprinkler placement and flow rate in a typical rack-storage array for achieving ideal flow conditions for fire suppression.

## PRESSURE SWIRL NOZZLES WITH A CONVEX DEFLECTION BODY TO INCREASE THE SPRAY ANGLE

Jens Kamplade, TU Dortmund, Germany; Ivette Hohlfeld, TU Dortmund, Germany; Marcus Kelz, TU Dortmund, Germany; Markus Thommes, TU Dortmund, Germany; Peter Walzel, TU-Dortmund, Germany

Pressure swirl nozzles operating in the regime of aerodynamic wave breakup are frequently used in food or chemical industry. Due to the intensive swirl flow, an air core develops and multiphase flow occurs. The liquid emerges as a lamella through a ring gap at the orifice. In some applications a large spray angle and small drops are desired. At common nozzles the spray angle is influenced by the nozzles design, as intensive swirl causes large spray angles. However friction loss increases, esp. when spraying liquids with higher viscosity resulting in thicker sheets and subsequently in larger mean drop sizes.

One also commercially available option to increase the spray angle includes a convex deflection extension at the nozzle's opening. Depending on the design, the lamella attaches to the convex contour or propagates unaffected by the extension. Within the present work, the deflection at different nozzle geometries and process conditions is investigated to obtain deeper understanding of the deflection mechanism. Drop size distribution is measured to quantify the effect on the spray of stronger divergent lamellas. At defined process condition smaller drops are found compared to the commonly used, sharp-edged orifices.

\*Support from grant "Deutsche Forschungsgemeinschaft" is gratefully acknowledged.

## INTERFACIAL PERTURBATIONS IN TWO-PHASE OIL-WATER PIPE FLOWS INDUCED BY A CYLINDER

Kyeong Hyeon Park, University College London, UK; Maxime Chanaud, University College London, UK; Panagiota Angeli, University College London, UK

Droplet detachment from interfacial waves is an important phenomenon in two phase flows which has been related to the transition from separated to dispersed flow patterns. To study experimentally the detachment of drops and the characteristics of the waves that result in drop formation, it is necessary to localize the phenomenon. In this work a transverse cylindrical rod is used to generate interfacial perturbations and breaking waves in stratified liquid-liquid flows. Studies are carried out in an acrylic test section with 37mm ID. The rod has 5mm diameter and is located at 12.5D from the inlet, near the interface of the two liquids. Tap water and Exxsol D140 (density 830kgm<sup>-3</sup> and viscosity 5.5cP) are used as test fluids. It was found that the presence of the cylindrical rod generates waves shortly after from which drops detach. High speed visualization has been performed to acquire the flow pattern map for this system. The interface downstream the cylinder is subject to a competition between three different contributions: the vortices shed by the cylinder, the wall effects due to the pipe itself and the interface fluctuations due to the mixing of the two phases. PIV has been used to acquire velocity fields and quantify these contributions near the cylinder.

## HYBRID INTERFACE CAPTURING SCHEME FOR MIXED MULTIPHASE FLOWS

Mohit P. Tandon, CD-adapco, India; Vinesh H. Gada, CD-adapco, India; Jebin Elias, CD-ADAPCO, India; Simon Lo, CD-ADAPCO, UK

Multiphase flows appear in several industrial applications from nuclear energy industry, chemical process industry to biomedical industry. Topological classification of such flows is based on interface structures. They can be divided in three major groups: dispersed, stratified or mixed flows. Mixed flows are ones where flow is dispersed in one region and stratified in another and they occur across many industrial and natural phenomenon. Typically some flavour of interface tracking/capturing method or Euler-Euler two-fluid approach is used to simulate such flows, with former suitable for stratified flows and latter for dispersed flows. It is desired to amalgamate the desirable characteristics of both classes of numerical approaches into one to simulate mixed flows. Large scale interface (LSI) model in STAR-CCM+ has been developed to model such flows. This study intends to develop a hybrid interface capturing scheme within LSI framework which abandons tracking an interface if it is ascertained that it is too small to be resolved. The study evaluates curvature and gradient based criterion to identify the topology and uses that information for advection of volume fraction. The study also evaluates the interface detection algorithm from Coste et al. (Nuclear Engineering and Design, 2013) where interface is determined explicitly and its knowledge is used for the hybrid scheme developed in this study. The scheme is then validated across variety of mixed flow problems: Rayleigh-Taylor instability, dam-break simulation and bubble column with a gas-liquid interface in its top-section.

## THE EFFECTS OF NANOPARTICLES IN SUBCOOLED FLOW BOILING IN THE DIFFERENT CHANNELS WITH SAME HYDRAULIC DIAMETER

Shams Mehrzad, Khaje Nasir Toosi university of technology, Iran; Hasan Alimoradi, M.S. C student at K.N.T University of Technology, Iran; Ziba Valizadeh, Iran

A numerical simulation of the subcooled flow boiling of water and water-based nanofluid in different cross sections channels with the same hydraulic diameter was developed. First, Euler – Euler approach for subcooled flow boiling of water in the channels was used. Then, copper oxide nano particles were injected to the boiling fluid. A finite volume method was employed for numerical modeling. The nanofluid subcooled boiling was modeled by considering three phases, liquid, vapor and nanoparticles. The motion of nanoparticles in the continuous fluid was modeled by Euler – Lagrange approach. Water, vapor and nanoparticles were considered continuous fluid, dispersed fluid and dispersed solid, respectively. Volume fraction and temperature variations along the channels were obtained. The results showed that, at low concentrations of nanoparticles (0.001 kg/S) rectangular channel and at higher concentrations (0.005 kg/S) square channel has greater changes in vapor volume fraction compared to pure water boiling.

## A NEW MODEL FOR THE PREDICTION OF FRICTIONAL PRESSURE DROP OF AIR-WATER TWO-PHASE FLOWS IN VERTICALLY-DOWNWARD TUBES

Chao Yao, Xi'an Jiaotong University, China; Hui-Xiong Li, Xi'an Jiaotong University, China; Yuqing Xue, Xi'an Jiaotong University, China; Chenyu Hao, Xi'an Jiaotong University, China

In this paper, a series of experimental studies on the frictional pressure drop of air-water two-phase flows in vertically-downward tubes are conducted. Based on the experimental data, 19 correlations collected in existing literature are evaluated, and it is shown that the prediction accuracy of these 19 correlations is poor, and generally decreases with the increase in tube diameter, indicating strong dependence on the tube diameter. On the basis of our experimental data, a new model is established in this paper for prediction of the frictional pressure drop of air-water two-phase flows in vertically-downward tubes. A new parameter B is introduced to consider both the effect of bubble buoyancy force and the effect of the tube diameter on the frictional pressure drop in vertically-downward tube. The parameter is determined by the mechanistic analysis of the experimental data. The prediction results of the new model are compared with the experimental data, and it is shown that the prediction MARD and MAD of experiment data are 29.3% and 1.3% respectively, indicating that the new model is more accurate than those existing correlations and can be used for the calculation of frictional pressure drop of gas-liquid two-phase flows in vertically-downward tubes.

## FRONT-TRACKING SIMULATION OF ELECTROWETTING DROPLET MANIPULATION

Yasufumi Yamamoto, Kansai University, Japan; Ryoko Otomo, Kansai University, Japan; Takahiro Ito, Nagoya University, Japan; Tatsuro Wakimoto, Osaka City University, Japan; Kenji Kato, Osaka City University, Japan

Electrowetting (wettability is improved by applying voltage between liquid and solid wall) is receiving a lot of attention as useful techniques for microactuators optical devices, display devices, controlling liquid feed, and so on. The shapes of meniscus at equilibrium states before and after voltage applying can be estimated by energy minimization. However, for some kinds of devices desiring instantaneous responsibility for voltage change, the performance prediction is difficult without considering dynamics of liquid flow. In this study, droplet manipulation utilizing electrowetting on dielectric are numerically reproduced and whole flow field containing a drop are solved. Gas-liquid interface is represented by the front-tracking method and moving contact line is modeled by the generalized Navier boundary condition. Voltage dependency of wettability is accounted by Young-Lippmann equation. Capacitors' series connection model (positive-electrode/dielectric/water and water/dielectric/negative-electrode are considered as connected two capacitors) can reproduce droplet behaviors as being trapped between positive and negative electrodes and transported with electrode's switching, similar as observed in the experiments (Suzuki et al. J. Adv. Mech. Des. Syst. Manuf. 2010). The effect of switching frequency and voltage are evaluated and compared with experimental data. Furthermore, the effect of viscosity, which affects the response performance of drop manipulation, is examined.

## EXPERIMENTAL INVESTIGATION OF HIGH PRESSURE WATER-STEAM FLOW CHARACTERS IN SIMULATED CRACKS

Yang Zhendong, xi'an University of Technology, china; Qiaoling Zhang, xi'an University of Technology, china; Qincheng Bi, Xi'an Jiaotong University, China

This paper presents a critical leak flow from a simulated crack under the fluid condition of high-pressure water. An experimental study of critical leak flow has been performed with rectangle cracks of changeable cross-section and constant cross section. The experimental results could be applied in the leak-before-break analysis of pipelines and pressure vessels. Experimental evaluation of the leak flow rate using artificial slits simulating the through-wall crack. Rectangle cracks with length of 80mm, an opening displacement of 140  $\mu$ m in the narrowest cross-section were performed in the experiments. Pressure drop and flow rates were measured to analyze the flow characteristics. Subcooled flashing and water-steam two phase discharge tests were carried out with a vessel pressure from 6MPa to 16 MPa. The following three phenomena are described and discussed. 1) For the wedge-shaped crack, it is obvious that leak flow rates in critical condition were influenced by opening displacement of outlet rather than that of inlet. 2) A plateau domain of pressure occurred in convergent crack, which was not found divergent and constant cross section cracks. 3) when the parameters of inlet is close to saturation, pressure pulsation appeared in the changeable cross-section cracks, especially near the outlet of test sections.

## EFFECT OF FLOW OBSTACLES ON THE POOL BOILING CRITICAL HEAT FLUX OF DOWNWARD-FACING METAL HEATER

Hong Hyun Son, Hanyang University, Republic of Korea; Kyusang Song, Hanyang University, Republic of Korea; Uiju Jeong, Hanyang University, Republic of Korea; Gwang Hyeok Seo, Hanyang University, Republic of Korea; Gyoodong Jeun, Hanyang University, China; Sung Joong Kim, Hanyang University, Republic of Korea

Pool boiling CHF experiments were performed to examine the effect of flow obstacle, which has various shape (square, circular, elliptic) and size (10, 15, 20, 25 mm). Stainless steel 304 heaters (99×49×2 mm<sup>3</sup>) were inclined at 10° downward with gap channel of 10 mm in height. Pool boiling condition was set to saturated deionized water at atmospheric pressure. Compared to the CHF without flow obstacle, the CHF decreased slightly with an increase of obstacle size because larger obstacle induces delayed stagnation of the leading edge at the downstream region of the obstacle. The entrance of following bubble flow is postponed, and in turn, increases the residence time of vapor layer. The shape effect was intensified with maximum obstacle size of 25 mm. Especially with the square obstacle, the edge stagnation occurred in the same direction with slug flow, so the vapor layer after the square obstacle was remained as a stagnant region, hardly affected by following bubble flow. By contrast, the edge stagnation of circular and elliptic obstacles occurred along curved-obstacle boundary, inducing the interaction of the bubble flow with the vapor layer. Thus it showed that the obstacle of a curved-boundary along flow direction is advantageous to delay the occurrence of CHF compared to the right-angled obstacle.

\* Support from grant NRF-No. 2014M2B2A9032081 is gratefully acknowledged.

## **SURFACTANT EFFECTS ON DROPLET COALESCENCE WITH A LIQUID-LIQUID INTERFACE**

Panagiota Angeli, University College London, UK; Victor Voulgaropoulos, University College London, UK; Wehliye Wehliye Hashi, University College London, UK; Maxime Chinaud, University College London, UK

In this study, droplet coalescence on an oil-water flat interface is investigated experimentally in a vertical Hele Shaw cell with 1.5 mm width. In such a configuration, the flow field recorded perpendicularly to the plates of the cell can be assumed as a 2D flow. The flow within the water drop and the lower water layer are investigated by using high speed bright field micro-PIV, which provides easily time resolved velocity fields during the coalescence process. It has been shown in previous studies that after the coalescence, vorticity peaks appear near the interface breaking points which are then advected to the bulk of the water phase. The main objective of this study is to compare the hydrodynamic evolution for different concentrations of an oil soluble surfactant (SPAN80), chosen to be below the CMC value. The results indicate that when the surfactant concentration increases there is significant evolution of the interface motion during coalescence. The change of interface shape observed when surfactant is added is found to be linked to the dynamics and the properties of the vorticity peaks. Indeed, as the surfactant concentration increases, the size of the vorticity peaks increases while their magnitude decreases.

## **NUMERICAL SIMULATION OF SURFACTANT CONCENTRATION AT AN INTERFACE OF SINGLE SPHERICAL DROP**

Kosuke Hayashi, Kobe University, Japan; Yuya Masukura, Kobe University, Japan; Shigeo Hosokawa, Kobe University, Japan; Akio Tomiyama, Kobe University, Japan

The Frumkin-Levich model for the adsorption-desorption kinetics of surface-active agent (surfactant) has been utilized in interface tracking simulations of contaminated bubbles and drops. The adsorption and desorption fluxes in the model are assumed to be proportional to the interfacial surfactant concentration and the model constants have been measured for static interfaces. However the applicability of the model to moving fluid particles has not been examined yet due to the lack of experimental data such as the interfacial surfactant concentration, the interfacial velocity and the Marangoni force. The authors will present these data of single contaminated glycerol-water drops in silicone oil at several Triton X-100 concentrations in the conference (Hosokawa et al.). In this study the contaminated drops are therefore simulated using a level set method proposed by Hayashi and Tomiyama (Int. J. Multiphase Flow, 2012; J. Comp. Multiphase Flows, 2014) to examine the validity of the Frumkin-Levich model for moving interfaces. The numerical results show that the model gives good predictions of the surfactant effect in a limiting case, i.e. fully-contaminated drops. The numerical predictions however deviate from the measured data at intermediate surfactant concentrations, which implies that the adsorption and desorption fluxes linearly dependent on the interfacial concentration with the model constants for static interfaces in the Frumkin-Levich model are not applicable to moving interfaces.

## **MODELLING AND SIMULATION OF PARTICLE AGGLOMERATION AND DEPOSITION IN TURBULENT CHANNEL AND PIPE FLOWS**

Koen Schutte, Delft University of Technology, The Netherlands; Luis Portela, Delft University of Technology, The Netherlands; Aris Twerda, TNO, The Netherlands; Ruud Henkes, Delft University of Technology, The Netherlands

We present a model for the formation, break-up, deposition and re-entrainment of solid particle agglomerates in turbulent flows. The turbulent flow of the liquid carrier is represented through Direct Numerical or Large Eddy Simulations. The structure of the agglomerates, which are formed by inter-linked spherical primary particles, is explicitly taken into account in this model. While firstly still ignoring deposition and re-entrainment, we present results on the properties of the agglomerates formed both in channel flow, and in flow in a cylindrical pipe. The collision rate of the agglomerates is found to be under-predicted by the Saffman & Turner collision kernel by up to a factor of 10. We also present results on the modification of the turbulence due to the presence of the agglomerates. Furthermore, results on the deposition and re-entrainment rate of the agglomerates, as well as on the additional streamwise pressure drop induced by the deposition are presented. Ultimately, the results obtained this way are used to construct improved engineering models for predicting asphaltene deposition during crude oil production.

This research was carried out in the context of the Integrated Systems Approach to Petroleum Production (ISAPP) Knowledge Center. ISAPP is a joint project of the Netherlands Organization for Applied Scientific Research (TNO) and Delft University of Technology, sponsored by ENI, Petrobras, and Statoil.

## **EXPERIMENTAL EVALUATION OF SURFACTANT CONCENTRATION AT AN INTERFACE OF SINGLE SPHERICAL DROP**

Shigeo Hosokawa, Kobe University, Japan; Yuya Masukura, Kobe University, Japan; Kosuke Hayashi, Kobe University, Japan; Akio Tomiyama, Kobe University, Japan

Experimental evaluation of surfactant concentration at a fluid-fluid interface is in great demand for understanding surfactant effect on mass and momentum transfer between a drop/bubble and the continuous phase in a contaminated system. In this study, spatiotemporal filter velocimetry (SFV) is applied to flows about single spherical drops of glycerol-water solution falling in a stagnant silicon oil under contaminated conditions. Triton X-100 is used for the surfactant and the drop Reynolds number is less than unity. Accurate velocity distributions measured by SFV enable us to evaluate viscous stresses at the interface, and therefore, the Marangoni stress, i.e., the surface gradient of interfacial tension, can be evaluated from the shear stress jump at the interface. Since the gradient of the interfacial tension is caused by non-uniform distribution of surfactant concentration at the interface, the surfactant concentration can be evaluated from the velocity distributions in the vicinity of the interface. Adsorption process of the surfactant is discussed based on time-evolution of interfacial concentration of surfactant. The experimental results prove that SFV is of great use for evaluating the Marangoni stress and distributions of interfacial tension and surfactant concentration at the interface. The evaluated distributions of surfactant can be used for validation of numerical methods taking into account adsorption and desorption of surfactants at an interface.

## **EFFECT OF INTERFACIAL PROPERTIES ON DROPLET-INTERFACE COALESCENCE: A COMPUTATIONAL STUDY**

Kai Li, Institute of Mechanics, Chinese Academy of Sciences, China; Mengyu Ma, China University of Petroleum, Beijing, China; Nannan Liu, China University of Petroleum, Beijing, China; Hang Jin, China University of Petroleum, Beijing, China; Wei Wang, China University of Petroleum Beijing, China

The process of droplet coalescence with oil-water interface reveals various features due to the differences in physical properties of liquids and interface. With the existence of surfactants in such systems, the phenomenon would become more complicated, due to the diverse roles of surfactants adsorbed onto the interface film. For further understanding of the coalescence process, numerical simulations of the coalescence process between a droplet and the oil-water interface were carried out in present paper. Governing equations published in literature were introduced to account for the effect of surfactant on droplet coalescence and to better represent the process, from the perspective of capillary and Marangoni effects. Partial and total coalescence were distinguished according to the criterion established in literature as well as in our experimental results. The availability of the computation was discussed and the effecting mechanism of given surfactant on droplet coalescence was explored.

## **NUMERICAL INVESTIGATION OF PARTICLE DEPOSITION IN TURBULENT DUCT FLOWS**

Jun Yao, Xiamen University, China; Michael Fairweather, University of Leeds, UK; Yanlin Zhao, China University of Petroleum-Beijing, China

Particle deposition in fully developed turbulent square duct flows is simulated using large eddy simulation for Reynolds numbers, based on the bulk velocity and duct width, equal to 250k, 83k and 10,320. A particle equation of motion including Stokes drag, lift, buoyancy and gravitational forces is used for particle trajectory analysis. Results obtained for the fluid phase show good agreement with experimental data and the predictions of direct numerical simulations. Predictions for particles show that the secondary flow established in the duct cross-section plays an important role in the particle deposition process. Under the influence of this flow, high-inertia particles (particle Stokes number,  $St > 12.38$ ) tend to deposit close to the corners of the duct floor, while low-inertia particles ( $St < 6.43$ ) deposit near the floor centre. It is shown that the flow Reynolds number, particle size, drag force, shear-induced lift force and gravity all affect the particle deposition process. The influence of the lift force increases with particle size, and its effect becomes significant as particles approach the duct floor; hence, it can act as another important factor causing particles to accumulate at the corners of the duct. Generally, and for all particle populations in the three flows considered, the particle deposition process can be described by the free-flight model.

# THE EFFECTS OF NEAR WALL CORRECTIONS TO HYDRODYNAMIC FORCES ON PARTICLE DEPOSITION AND TRANSPORT IN VERTICAL TURBULENT BOUNDARY LAYERS

Michael Reeks, Newcastle University, UK; Chunyu Jin, Newcastle University, UK; Ian Potts, Newcastle University, UK

We examine the influence of near wall corrections on the drag and lift on particle deposition in a fully developed turbulent boundary layer to see if it will explain the significant underestimate in the deposition rate predictions of a 1D stochastic model for low inertial particle  $\tau^+ < 5$ . Whilst we find that these corrections are unable to explain this discrepancy they nevertheless do have a noticeable effect on the particle transport. Corrections to the drag are found to decrease deposition rates for small particles and to increase them for large particles. The inclusion of a Saffman lift force and a more recent formula reduced the deposition rates particles but the change was not significant (unlike previous predictions). In contrast, the corrected drag does yield significant differences between the particle and fluid mean streamwise velocities for large particles ( $\tau^+ \sim 20$ ) in the near wall region.

# TWO-WAY COUPLING EFFECTS IN PARTICLE-LADEN TURBULENCE: HOW PARTICLE-TRACKING SCHEME AFFECTS PARTICLE AND FLUID STATISTICS

Jeremy Horwitz, Stanford University, USA; Mona Rahmani, Stanford University, USA; Gianluca Geraci, Stanford University, USA; Andrew Banko, Stanford University, USA; Ali Mani, Stanford University, USA

We present a method for tracking small inertial particles in two-way coupled flows. The method was developed to resolve an issue with standard Lagrangian point-particle tracking whereby the disturbed fluid velocity at the particle location was used incorrectly in place of the undisturbed fluid velocity at the particle location. The latter quantity is the correct value to be used in computing the Stokes drag which affects momentum exchange between fluid and particles. Standard interpolation schemes which attempt to calculate the disturbed fluid velocity will make worse predictions about Lagrangian statistics, especially as the particle size to grid-spacing increases, and as the order of accuracy of the interpolation-projection scheme increases. The particle trajectories and miscalculation of particle-drag force will affect Eulerian statistics too, especially at appreciable mass loading. We consider homogeneous isotropic turbulence laden with particles of varying Stokes numbers and sizes, at different mass loadings. Mono- and poly-disperse flows are considered. For otherwise dynamically equivalent problems, we examine the consequences of different numerical schemes on the evolution of the energetics and structure of the fluid turbulence as well as on Lagrangian acceleration statistics. The goal is to report the range of observations that may simply be a consequence of change in numerical strategy for particle tracking.

\*Supported by DOE, J. H. Supported by NSF GRF

# OIL-WATER FLOW CHARACTERISATION IN HORIZONTAL PIPES USING SIMULTANEOUS LASER-INDUCED FLUORESCENCE AND PARTICLE IMAGE/TRACKING VELOCIMETRY

Roberto Ibarra, Imperial College London, UK; Ivan Zadrazil, Imperial College London, UK; Christos N. Markides, Imperial College London, UK; Omar K. Matar, Imperial College London, UK

The co-current flow of two immiscible liquids in pipes is commonly encountered in a wide variety of industrial processes from reactors to oil export pipelines at offshore facilities. The behaviour of these flows is a complex phenomenon not fully understood in which interfacial characteristics and droplet formation play an important role in the development of liquid-liquid segregated flow patterns and dispersion. Such phenomena need to be included in predictive models. The characterisation of these types of flows is commonly performed by impedance probes, conductive wire probes and other intrusive techniques which affect the behaviour of the flow near the probe. In this work, non-intrusive optical measurement techniques are employed to obtain detailed spatiotemporally information on interface behaviour, velocity profiles and turbulent measurements for water and Exxsol D140 in a horizontal 32 mm inside diameter pipe. Planar laser-induced fluorescence provides information on the phases configuration along with interface level and phase fraction. Particle image and tracking velocimetry are employed to obtain instantaneous velocity profiles of the flow which are central to quantifying the mechanism of drop break-up by turbulence and near wall velocity shear acting on the dispersed phase.

\* This work has been undertaken within the Consortium on Transient and Complex Multiphase Flows (TMF).

# MEASUREMENT OF WALL SHEAR STRESS IN CHURN FLOW IN A LARGE DIAMETER VERTICAL RISER

Mohammed Zangana, Koya University, Kurdistan Region- Iraq; Barry James Azzopardi, University of Nottingham, UK

Churn flow is one of the least understood flow patterns in gas-liquid flows in vertical pipes. Prior work, mainly on smaller diameter pipes has shown that part of the liquid travels as a film on the wall whilst the rest is entrained in the gas. There can be up and down oscillations in the direction of flow of the film. Here, we report on experiments in which film thicknesses and directionally resolved wall shear stress are measured on a 127 mm diameter pipe operating at a pressure of 3 bar. The calibration and testing of the wall shear stress technique are presented together with the measured data over ranges of gas and liquid superficial velocities of 5-17 and 0.02-0.14 m/s respectively. The link between wall shear stress and film thickness are explored. These data are combined to infer the fraction of liquid entrained in the gas. These results are compared with the predictions of published models. The shear stress results are examined in the light of prior data from pipes of different diameters.

# EFFECTS OF AIR INJECTION POSITIONS ON THE FLOW STRUCTURE OF HIGH VISCOSITY OIL IN A LARGE DIAMETER PIPES

Shara Mohammed, University of Nottingham, UK; Abbas Hasan, University of Nottingham, UK; Barry James Azzopardi, University of Nottingham, UK; Georgios Dimitrakakis, The university of Nottingham, UK

The majority of the literature on gas-liquid slug flow has been focused on low viscosity liquids often flowing in smaller diameter pipes whereas, studies describing the behavior of high viscosity liquids flowing in larger diameter pipes are limited. Herein we add to the limited work on high viscosity liquids in larger diameter pipes. Studies on the flow structure of high viscous silicone oil (300 Pa.s) in bubble columns of 240 and 290 mm diameter are presented. Particularly, the effects of different air injection configurations were investigated using Electrical Capacitance Tomography. Data were obtained on Taylor bubble and liquid slug lengths, bubble velocity, frequency and liquid film thickness. Mean void fraction, Taylor bubble velocity and frequency, and film thickness did not show a significant change with changing the air nozzle location, while there was a remarkable change for Taylor bubble and liquid slug lengths. Taylor bubble length increases significantly with increasing the distance of the air nozzle from the center of the smaller diameter pipe (240 mm) and increases slightly in the larger diameter pipe (290 mm). At the same time, liquid slug length decreases significantly in the smaller pipe, whereas it only increases slightly in the larger diameter pipe.

# ENTRANCE EFFECTS ON TWO-PHASE FLOW DEVELOPMENT IN A VERTICAL PIPE: EXPERIMENTS AND SIMULATIONS

Raad Issa, Imperial College, UK; Francesco Galleni, Imperial College, UK; Barry James Azzopardi, University of Nottingham, UK; Buddhikha Hewakandamby, University of Nottingham, UK; Josep Escrig Escrig, University of Nottingham, UK

The paper investigates the effect of inlet flow conditions on the development of intermittent (slug or churn) flow of gas and oil, both experimentally and numerically in a 67 mm ID, 6 m long vertical pipe. Two gas injection geometries, 25mm ID concentric pipe and a pipe with 98 holes (4 mm diameter) on a cylindrical surface were employed. Air and oil were introduced at superficial velocities ranging from 0.05 to 1 m/s and 0 to 0.4 m/s respectively. Cross sectional void fraction time series were measured using a Wire Mesh Sensor and Electrical Capacitance Tomography to obtain probability density functions, average cross section void fraction, structure velocities and structure frequencies. A one-dimensional two-fluid model utilising the "slug capturing" numerical methodology was applied to both inlet configurations. Like in horizontal flow, the method is found to be able to capture the formation of slugs automatically; slug characteristics such as frequency and velocity compare very well with the data.

## CONTROLLED BUBBLE FORMATION BY A FORCING MECHANISM IN A PLANAR CO-FLOW CONFIGURATION

Javier Ruiz-Rus, Universidad de Jaén, Spain; Rocío Bolanos Jimenez, Universidad de Jaén, Spain; Candido Gutierrez Montes, Universidad de Jaén, Spain; Alejandro Sevilla, Universidad Carlos III de Madrid, Spain; Carlos Martínez Bazan, Universidad de Jaén, Spain

We present an analytical description that models the bubbling regime of an air film discharging between two water streams with planar geometry. Based on the previous experimental and numerical characterization of this problem, the gas stream is modeled as a planar sheet divided into three different parts: a neck that moves downwards at the water velocity, a ligament upstream of the neck, and a forming bubble downstream of the neck, with uniform dimensionless half-thicknesses  $\eta_n(t)$ ,  $\eta_l(t)$ ,  $\eta_b(t)$ , and pressures  $\pi_n(t)$ ,  $\pi_l(t)$ , and  $\pi_b(t)=\pi_n(t)$ , respectively. Lengths are made dimensionless with the air thickness at the outlet,  $H_0$ , and pressures with twice the air dynamic pressure,  $\rho_a u_a^2$ . In a reference frame moving with the water velocity,  $u_w$ , and imposing a negative pressure caused by the sudden planar expansion of the air stream at the outlet of the injector, a set of differential equations are obtained, that can be numerically integrated to obtain the temporal evolution of the different thicknesses and pressures, showing a good agreement with numerical and experimental results for a given value of the initial velocity of the collapsing neck. The latter is the only free parameter of the model, that shows a weak dependence on the control parameters, namely,  $We=\rho_w H_0/\sigma$ , and  $\Lambda=u_w/u_a$ , where  $\sigma$  is the surface tension coefficient.

\* Support from Spanish grants MINECO DPI2014-59292-C3-1-P and DPI2014-59292-C3-3-P, Junta de Andalucía P11-TEP749 and Universidad de Jaén UJA2013/08/05.

## BUBBLES RELEASE FROM HYDROPHOBIC SITE UNDER MAGNETIC FIELD

Hongbo Liu, Chongqing University, China; Liangming Pan, Chongqing University, China; Tianming Chen, Chongqing University, China

The bubble management is an important issue which is related to the electrolysis efficiency and energy conservation in water electrolysis for hydrogen production. It has been proved that the external magnetic field has the effect of intensifying the two phase flow and reducing the electrode potential. The paper gave an experiment observation of hydrogen bubble growth and release from copper electrode partially covered with hydrophobic materials under magnetic field. It was found that the bubbles have periodic release from the hydrophobic site. The magnetic field has the effect of accelerating the bubble release no matter whether the field orientation is perpendicular or parallel to the electrode surface. The mechanism of magnetic field influence on bubble release is not all the same with what has been known before. Especially for the case of magnetic field perpendicular to the electrode surface, one hand, it is confirmed from the CFD analysis result that a relatively low pressure region is formed as the Micro-MHD effect. On the other hand, many unresolved foam of tiny microbubbles connect the big bubble and the electrode surface at the hydrophobic site, and the micro-MHD leads to the tiny microbubbles disturbance which accelerates the big bubble release.

## NUMERICAL SIMULATION OF MAGNETIC DRUG DELIVERY TO HUMAN MAXILLARY SINUS

Hamed Nikoogar, Shiraz University, Iran; Omid Abouali, Shiraz University, Iran; Mohammad Eghtesad, Shiraz University, Iran; Goodarz Ahmadi, Clarkson University, USA

Magnetic drug targeting (MDT) is an externally triggered drug delivery method based on attraction of ferromagnetic aerosols and hydrosols to high gradient magnetic source placed in target area. This method is suggested as an efficient strategy for drug delivery to tumors and infected regions in respiratory tract and blood vessels by several researchers in the recent years.

In this research, drug delivery to human maxillary sinus using MDT method is numerically analyzed. 3-D realistic models of nasal airway including nasal cavity and maxillary sinus before and after virtual endoscopic surgery are being injected with ferromagnetic micro-particles and exposed to an external magnetic source. Micro-particles consist of ferromagnetic core and drug shell. Static magnetic field generated by electromagnetic coil is applied as an external excitation for MDT. Effects of magnetic field strength and particle diameter on drug deposition are studied in the pre and post surgery models. Investigation of results proves the effectiveness of MDT method as a new approach for drug delivery to human maxillary sinuses.

## STUDY OF A BUBBLE PRODUCTION IN SHEAR LIQUID-GAS FLOWS USING THE BRIGHTNESS BASED LASER INDUCED FLUORESCENCE TECHNIQUE (BBLIF)

David Hann, University of Nottingham, UK; Andrey Cherdantsev, Kutateladze Institute of Thermophysics, Russian Federation; Buddhikha Hewakandamby, University of Nottingham, UK; Barry James Azzopardi, University of Nottingham, UK

The interaction between a shearing air flow and a liquid layer being driven by the air is of interest in a range of industrial areas. Liquid entrained in the gas as droplets is accelerated and then can impact with the surface to produce bubbles. This research uses the BBLIF technique to investigate the impact of droplets in a shear rig with gas velocities in the range 25-35 m/s. The results show that there are two types of impact; crater and furrow. The appearance of furrows is accompanied by massive entrainment of gas bubbles by the liquid film. The number and size of the bubbles appearance were measured and appear to increase during the impact before decreasing afterwards. It is shown that the additional bubbles are unstable and pop. The majority of bubbles created by furrows can be seen in the base layer between the disturbance waves that create the impacting droplets.

The overall number, size and velocity of the bubbles was analyzed and it was shown that the bubble density saturates as the liquid loading is increased. The speed of the bubbles is shown to be highly correlated to the local height of the film.

## ON THE PHYSICS OF FIZZINESS: HOW BUBBLE BURSTING CONTROLS DROPLETS EJECTION

Thomas Seon, CNRS, France; Elisabeth Ghabache, CNRS, France; Arnaud Antkowiak, UPMC, France; Christophe Josserand, Institut D'Alembert, France

Bubbles lying at the free surface of a liquid usually burst in ejecting myriads of droplets. Focusing on the bubble bursting jet, prelude for these aerosols, we propose a simple scaling for the jet velocity and we unravel experimentally the intricate roles of bubble shape, capillary waves, gravity, and liquid properties. We demonstrate that droplets ejection unexpectedly changes with liquid properties. In particular, using damping action of viscosity, self-similar collapse can be sheltered from capillary ripples and get closer to the singular limit, therefore producing faster and smaller droplets. These results pave the road to the control of the bursting bubble aerosols.

## MODAL BEHAVIOUR OF THREE-DIMENSIONAL ARTERIAL FLOW STRUCTURES

Lisa Pahl Wittberg, Royal Institute of Technology (KTH), Sweden; Kartik V. Bulusu, George Washington University, USA; Stevin Van Wyk, Royal Institute of Technology (KTH), Sweden; Laszlo Fuchs, Royal Institute of Technology (KTH), Sweden; Michael W. Plesniak, George Washington University, USA

Cardiovascular diseases such as atherosclerosis are localized to regions of curvature in the arterial network. The pathology of atherogenesis is widely considered an inflammatory response, hypothesized to be modulated by the interplay between Wall Shear Stress (WSS) variations and particulate transport mechanisms from the fluid core to the near-wall regions. The WSS is determined by the local flow characteristics as well as the rheological properties of the blood, in turn dependent on the bulk secondary flows. Development of planar secondary flow structures of blood-like fluids in a 180 degree curved artery model was investigated numerically and experimentally (using 2D-PIV). The time dependent fluid flow under various physiological conditions was investigated numerically, validated with a Newtonian fluid model from experiments and extended to non-Newtonian blood properties. In our previous studies, the complexity of the interaction of the Dean- and Lyne-vortices developing under pulsatile conditions at increasing peak Reynolds numbers and shear layer instabilities associated with non-Newtonian effects in the flow were described. In this paper, we build on knowledge gained from arterial flow structures in the planar regions and develop the 3D aspects of arterial coherent structures using Proper Orthogonal Decomposition and Dynamic Mode Decomposition techniques.

\* Support from the Swedish Research Council (Vetenskapsrådet) and NSF Grant CBET 0909678 is gratefully acknowledged.

## PLATELET MARGINATION AND THE RHEOLOGY OF WHOLE BLOOD

Niclas Berg, KTH, Sweden; Laszlo Fuchs, Royal Institute of Technology (KTH), Sweden; Lisa Prah Wittberg, Royal Institute of Technology (KTH), Sweden  
 Blood consists of various cells suspended in a viscous fluid, the plasma. The red blood cells (RBC) occupy the majority of the cellular volume and the rheology of whole blood is thus mainly governed by the interaction between the plasma flow and the RBCs. The motion of the RBCs also influences the transport of other cells, in particular the platelets, which play an important role in blood clot formation. It should however be noted that the RBC dynamics itself depends on the mechanical properties of the RBCs, which can change under pathologies. In this work we study how these changes affect the rheology of the blood and the platelet transport using numerical simulations. The flow field around the cells is resolved using an immersed boundary method coupled with the Lattice Boltzmann method. The RBCs are modelled as deformable, fluid-filled capsules and the platelets as rigid particles. Emphasis is on the dependency of shear modulus of the membrane and the viscosity difference between fluid inside and outside the RBCs.

\* The computational resources provided by Swedish National Infrastructure for Computing (SNIC) are gratefully acknowledged.

## EXPERIMENTAL INVESTIGATION OF THE FLOW BOILING HEAT TRANSFER CHARACTERISTICS WITH SUPER-HYDROPHILIC SURFACE IN NATURAL CIRCULATION THERMOSYPHON LOOP

Xiaopeng Zhang, Institute of thermal science and power systems, China; Liang Zhang, Institute of thermal science and power systems, China; Meng Hua, Institute of thermal science and power systems, China; YiLing Zhang, Zhejiang University, China; LiWu Fan, Institute of Thermal Science and Power Systems, China; Zi-Tao Yu, Institute of thermal science and power systems, China  
 An experimental study was carried out to investigate the effects of super-hydrophilic surface on flow boiling enhancement in a natural circulation thermosiphon. The super-hydrophilic boiling surface in evaporator was obtained by a wet chemical reaction method. A heat load of 1.0kW was applied to heat a 0.8m long evaporator tube with an inner diameter of 15.0 mm. The results show that the super-hydrophilic has enhanced the average heat transfer coefficient by more than 200%. Moreover, besides of the decrease of wall temperature, a drop of the temperature difference resulted from the gas-liquid stratification flow was also observed. The mechanisms of the enhanced boiling heat transfer were also analyzed.

\* Support from the National Natural Science Foundation of China (NSFC) under Grant No. 51406177, the China Postdoctoral Science Foundation (CPSF) under Grant Nos. 2014M551734 and 2015T80613 is gratefully acknowledged.

## NEW NUCLEATION BOILING MODEL DEVOTED TO HIGH PRESSURE FLOWS

Stephane Mimouni, EDF R&D, France; William Benguigui, EDF, France; Mathieu Guingo, , France; Jerome Lavieville, EDF R-D, France; Namane Mechitoui, , France; Nicolas Merigoux, , France

Extensive efforts have been made in the last five decades to evaluate the boiling heat transfer coefficient and the critical heat flux in particular. Boiling crisis remains a major limiting phenomenon for the analysis of operation and safety of both nuclear reactors and conventional thermal power systems. As a consequence, models dedicated to boiling flows have been improved. For example, Reynolds Stress Transport Model, polydispersion and two-phase flow wall law have been recently implemented. Nevertheless, these models are still being improved.

Moreover, most of nucleate boiling models implemented in the CFD codes are based on the calculation of the site density where bubbles are emitted. This quantity is evaluated by an empirical correlation which depends on thermal-hydraulics conditions and material surface. Such formulation leads to large uncertainties.

The objective of this paper is to propose a new mechanistic model devoted to CFD scale and allowing similar results for the thermal-hydraulics variables (velocity, liquid temperature and void fraction near the wall) but improved results for the wall temperature. The new nucleate boiling model does not depend on the site density and has been validated against 20 test cases devoted to high pressure flows. Furthermore, it was found that the sensitivity to the grid refinement was acceptable.

## EXPERIMENTS AND MODELS FOR TRANSIENT BOILING

Nicolas Baudin, IRSN, France; Pierre Ruyer, IRSN, France; Catherine Colin, INP Toulouse, France; Julien Sebilliau, INP Toulouse, France

When a large power is rapidly supplied to a heater immersed in water, the onset of boiling regimes and the heat transfer can differ substantially from classical steady state situations. This can be a matter of study to evaluate some risk of failure of materials in the context of nuclear safety. The relaxation of the wall to fluid heat flux toward steady state values depends on heating, flow, and boiling regime conditions.

We consider the use of an experimental device that allows supplying power to a thin wall in contact with a sub-cooled flow of an organic fluid. It operates over a wide range of conditions: power pulses, steps and ramps or wall temperature steps and ramps. For each regime, it leads to either steady state or transient heat transfer. Those regimes are: convection, sub-cooled nucleate boiling, and inverted annular film boiling. Wall temperature and power are measured, that allows deducing the wall to fluid heat flux.

For single phase convective heat transfer, the relaxation time of the heat transfer coefficients depends on the flow rate and can be determined from experimental results. Numerical simulations and theoretical study support this point. A similar behavior is observed from experimental transient nucleate boiling regimes. Different asymptotic behaviors of the nucleate boiling heat transfer during power transients are analyzed: the conditions for the heat transfer coefficient to converge toward the steady-state value are modeled.

## PERFORMANCE EVALUATION OF CAPILLARY-ASSISTED PLATE-TYPE EVAPORATOR WITH CU-CNT-TIO2 COMPOSITE COATING

Edward Joshua Pialago, Chonbuk National University, Republic of Korea; Hyun-Ho Ha, Chonbuk National University, Republic of Korea; Dong An Cha, Republic of Korea Institute of Industrial Technology, Republic of Korea; Kyung Jin Bae, Korea Institute of Industrial Technology, Republic of Korea; Min Soo Kim, Chonbuk National University, Korea; Oh Kyung Kwon, Republic of Korea Institute of Industrial Technology, Republic of Korea; Chan Woo Park, Chonbuk National University, Republic of Korea

Ternary Cu-CNT-TiO<sub>2</sub> composite powders were fabricated by ball milling. These composite powders were applied on Cu plate substrates as coating. The wettability of the coatings in water is investigated at atmospheric condition by measuring the contact angles between water droplets and the surface. The coating with the best wettability in water was then applied on Cu plates, which were used in an evaporator. Evaporation heat transfer tests on the plates were conducted at saturated water vapor pressures ranging from 6 Torr to about 8 Torr. The evaporator has a single-pass single-plate-per-pass setup. The influence of immersion depth and hot water velocity as well as evaporation pressure were investigated. The heat transfer characteristics of the coated plates were compared to those of the plain plate without coating and plate with fins. The results revealed that high heat transfer performance could be achieved when the evaporator plates were partially immersed in water with levels that were about half of the plate height.

## AN EULER-LAGRANGE METHOD WITH VOLUMETRIC COUPLING FOR CAVITATING FLOWS

Michel Schumichen, Technische Universität Dresden, Germany; Frank Rudiger, Technische Universität Dresden, Germany; Jochen Frohlich, Technische Universität Dresden, Germany

Euler-Lagrange models become more and more popular for the simulation of cavitating flows. Most of the work reported in the literature applies the one-way coupled Euler-Lagrange model. This model neglects the impact of the cavitation bubbles on the liquid phase, which is only valid for cases with small dispersed bubbles and low bubble growth rates. In oil hydraulic spool valves, for instance, under common operating points cavitation is characterized by a high percentage of small bubbles together with few very large bubbles. In this case the consideration of the volumetric coupling via the displacement of the liquid due to bubble volume variation is important. To the best of the authors' knowledge no satisfactory Euler-Lagrange method presently exists which is able to account for this effect. The contribution presents such an Euler-Lagrange method with volumetric coupling which is able to handle arbitrarily large bubble growth rates. This is achieved by modifying the underlying Finite-Volume solver in a non-trivial way and an appropriate time stepping scheme. The method is validated by an academic configuration where a reference solution is available. Also, the difference in result with respect to standard methods is highlighted. Finally, the impact of the volumetric coupling is demonstrated for a cavitating nozzle flow.

#### VOID FRACTION FLOW FIELD MEASUREMENTS OF CAVITATING FLOW IN THE WAKE OF A CIRCULAR CYLINDER

Harish Ganesh, University of Michigan, USA; Juliana Wu, University of Michigan, USA; Tiezhi Sun, Harbin Institute of Technology, China; Steven L. Ceccio, University of Michigan, USA

Cavitation in the wake vortices behind objects is common in many naval applications. Cavitation dynamics in the wake of a circular cylinder has been studied extensively due to its simplicity and different regimes of wake vortical flow. From previous studies it has been found that the nature of the shear flow in the wake has an important role to play on the cavitation dynamics. In the present study, cavitation in the wake of a cylinder is investigated using high-speed video cameras and X-ray densitometry. Using synchronized top and side views from high speed video cameras, different types of cavities forming on the wake of the circular cylinder is studied for a range of cavitation numbers, at a Reynolds number of  $1 \times 10^5$ , which lies at the transition region between sub-critical to critical regime of wake transitions. X-ray densitometry based void fraction flow field measurements of different types of cavities at various cavitation numbers are then analyzed to understand the role of vapor production in the observed dynamics. Both instantaneous and time averaged void fraction flow fields are presented and discussed.

\* This work was funded by U.S. Office of Naval Research, grant number N00014-14-1-0292, under program manager Dr. Ki-han Kim.

#### RELATIONSHIP BETWEEN MOTION OF ACOUSTIC CAVITATION-ORIENTED BUBBLES AND FLOCCULATION PATTERNS, DEPENDING ON COMPONENTS OF DISSOLVED GASES IN WATER

Koji Nomata, Shizuoka University, Japan; Sayuri Yanai, Shizuoka University, Japan; Hiroya Muramatsu, Shizuoka University, Japan; Takayuki Saito, Shizuoka University, Japan

Particle manipulation techniques using MHz-band ultrasound have been investigated for long years, and the target diameter of the particle was smaller than  $\mu\text{m}$ -order due to the wavelength limitation. We used kHz-band ultrasound and have overcome the limitation. Flocculation of sub-mm-order particles dispersed in water under irradiation of ultrasound was found out. It was caused by Acoustic Cavitation-Oriented bubbles (ACOBs) adhering to the particle surface. The ACOBs moved with the particle toward the flocculation position by the Acoustic radiation force acting on the bubbles.

We discovered the flocculation patterns were categorized into two types: spherical or chain-like; and the key factor was the dissolved gas components. The spherical flocculation resulted from the dissolved air gas, and the chain-like one resulted from the dissolved  $\text{CO}_2$ . However, the mechanism of the particle flocculation with the interaction between the ACOBs and particles and effects of the surrounding liquid motions are still unknown. In the present study, to reveal the mechanism of the particle flocculation, we visualized the bubbles, particles and its surrounding liquid motions, under precise control of the dissolved gas components. Furthermore, we discussed the relationship between the flocculation patterns and the bubble motions.

#### THE BREAKUP OF A VISCOUS LIQUID DROP IN A HIGH REYNOLDS NUMBER SHEAR FLOW

Chin Hei Ng, University of Washington, USA; Alberto Aliseda, University of Washington, USA

The breakup of a viscous liquid drop in a sheared turbulent flow of an immiscible liquid is a key open problem in multiphase flows. Despite significant understanding from intense experimental and computational research in recent decades, the formation of high aspect ratio ligaments makes this process more complex than the Hinze-Kolmogorov paradigm of a single eddy-particle collision, and allows only for limited predictive capabilities from the various breakup models based on this paradigm. We investigate the break-up of a high-viscosity, equal-density drop in a high Reynolds number (up to 250,000) submerged round jet. Unlike in the H-K theory, applicable to the break-up of inviscid fluid particles, the breakup of viscous droplets occurs through a sequence of eddy collisions at multiple scales, with the formation of ligaments that grow beyond the integral-scale and eventually break into many inertial-scale droplets. High speed imaging of this process is statistically analyzed to establish the scaling of droplet deformation, ligament stretching and break-up times. A phenomenological model for the breakup time, the breakup probability and the breakup frequency is validated from the experimental results, for different Weber and Ohnesorge numbers. Additionally, a resulting particle size model is formulated and compared against the experimental measurements when break-up is no longer occurring.

\* NOAA support for this research through grant NA10OAR4320148 is gratefully acknowledged

#### NUMERICAL SIMULATIONS OF THE UNSTEADY CAVITATING FLOWS AROUND A CIRCULAR DISK AND CONICAL CAVITATOR

Minjae Kim, Agency for Defense Development, Republic of Korea; Kurnchul Lee, Agency for Defense Development, Republic of Korea

In the present study, the unsteady cavitating flows around a circular disk and conical cavitator are numerically simulated using RANS equations and mixture model to predict the cavity shape and drag of those cavitators. The predicted cavity length, diameter and drag of the disk and cone are compared with other numerical results, semi-empirical formula, theoretical and experimental data at different cavitation numbers and show good agreements with those data. The numerical simulations for an underwater body with the circular disk and conical cavitator are also performed to investigate the influence of the body on cavity size. It is found that the cavity size is slightly shortened due to the re-entrant jet generated at cavity closure.

#### BEHAVIOR OF DROPLETS IMPACT ON DRY AND WET WALLS: A NUMERICAL STUDY

Ming Chen, Institute of High Performance Computing, Singapore; Baili Zhang, Institute of High Performance Computing, Singapore; Jing Lou, Institute of High Performance Computing, Singapore

The impact of a liquid drop on a surface has fascinated researchers for long time. One of reasons is because of the numerous physical phenomena accompany the interaction of drop with surface and their relevance in larger number of applications, such as, inkjet-printing, spray-cooling, soil-erosion due to rain-drop-impact and fuel injection in internal-combustion-engines. Furthermore, many artists use drop to create drop arts. A lot of experiments show the outcome of drop impact on a dry surface can be dramatically altered, depending both on the physicochemical characteristics of drop and on those of the surface itself.

In this work, a lattice Boltzmann model for high-density ratio two-phase flow is developed as an alternative to experimental approach to investigate the behavior of droplets impact on a dry wall and a wet wall. We study drop impact dynamics by quantitatively analyze the evolutions of interface shape, velocity fields, and other data generated at different Weber and Reynolds numbers. The present numerical simulations not only reproduce the bouncing and splashing behavior observed in experiments on the dry surface and on the wet wall, respectively, but also reveal the influence of the impact angle on those dynamic characteristics

#### AERODYNAMIC BREAKUP OF NON-SPHERICAL DROPLETS

Yannis Hardalupas, Imperial College London, UK; Konstantinos Bergeles, Imperial College London, UK; Georgios Charalampous, Imperial College London, UK; Alex M.K. P. Taylor, Imperial College London, UK

The breakup of liquid fragments through aerodynamic atomisation is commonly occurring while the fragments are non-spherical. Most computational breakup models are based on semi-empirical correlations of regimes and outcomes from the breakup of originally spherical droplets. However, the effect of non-sphericity on droplet or ligament breakup is not known. An experimental study is presented investigating the breakup morphology of non-spherical water droplets in a range of We and Oh numbers. The experimental layout consisted of an ultrasonic levitator holding the non-spherical droplet, which was aerodynamically atomised. The initial shape of the droplet was controlled by adjusting the aspect ratio of the ellipsoidal, through the amplitude of the ultrasonic wave. A high speed camera with acquisition rate of 10kHz and image magnification  $12\mu\text{m}/\text{pixel}$  with a novel image processing allowed the imaging of the temporal evolution of the droplet deformation. The droplets were impulsively accelerated by a sudden gust of air, released from a pressure chamber by a solenoid valve. The results indicate that the conventional definition of the We number is not sufficient to categorise the various breakup regimes and a modified We number ( $\text{WeAR}$ ), based on the aspect ratio of the non-spherical droplet, fitted the experimental data to the standard morphology charts. The droplet acceleration and its drag coefficient don't remain constant with time and novel drag coefficient expressions as a function of time are proposed.

## PARTICLE GROWTH IN TURBULENT FLOW UNDER DYNAMICALLY CRITICAL STOKES CONDITIONS

Michele Guala, *university of minnesota, USA*; Jiarong Hong, *university of minnesota, USA*

Following field scale observations of snowflake distribution in an atmospheric turbulent flow, we proposed a model for inertial particle growth resulting from dynamic clustering at scales both similar and larger than the Kolmogorov length scale. Particles able to aggregate or coalesce are expected to grow in size in flow regions where preferential concentration is predicted by a critical Stokes number  $St = (\rho/\mu)f \approx 1$ , due to increased collisional probability. We assume that, during growth,  $St$  stays critical, with the particle response time ( $\tau_p$ ) growing according to an evolving flow time scale ( $\tau_f$ ) defined by the vortices around which progressively larger particles end up orbiting. This mechanism leads to the prediction of the distribution of particle size at different turbulent scales, within the inertial range, as well as the limiting size of aggregating particles in a turbulent flow. The latter limit is defined by the extent of the turbulent inertial range, and it can be formulated as a function of measurable integral-scale quantities, such as the r.m.s. velocity or the integral time scale (see Guala and Hong ArXiv 2015). The mechanism of particle size growth, under dynamically critical Stokes number, provides a first order framework to study particle aggregation, size growth and particle cluster growth in various geophysical multi-phase flows. Large scale preferential sweeping was observed in the wake of a utility-scale wind turbine during a snowstorm (Hong et al. Nature COMM 2014). Predictions on rain droplets and snowflakes sizes in atmospheric flows are consistent with observations in the literature.

## THE SIMULATION OF MULTILAYERED MEDIA OF FLUID AND SOLID USING THE GHOST METHOD

Zhi Wei Feng, *Beijing Institute of Technology, China*; Boo Cheong Khoo, *National University of Singapore, Singapore*; Ji Li Rong, *Beijing Institute of Technology, China*

In this work, we simulate the multilayered media of fluid and solid using the ghost method in an Eulerian-Lagrangian system. The modified ghost fluid method is employed for the interaction between the different fluids and is able to account for all possible wave patterns. The ghost solid-fluid method is used to deal with interaction between the fluid and solid undergoing elastic-plastic deformation, and also the ghost solid method is used to deal with the elastic-plastic solid-solid interface. Using the above methods, some 1D cases with different material combinations have revealed that the method works for various problems like shock-tube problem, strong shock wave incident on the interface, etc. Then the different ghost methods are combined and used to study the properties of the lamination which is composed of several different solid layers under the influence of shock wave. Lastly, underwater explosions near the elastic-plastic solid under different domain setups and geometries are simulated to study the effect of the material property and domain setup on the underwater shock refraction, the bubble shape and the transient response of the solid.

\* Support from NSAF grant No.U1530147 is gratefully acknowledged.

## A PARALLEL ADAPTIVE FINITE ELEMENT METHOD FOR SIMULATING THE INCOMPRESSIBLE FLOWS WITH FREE SURFACE

Long Ngo, *Seoul National University of Science and Technology, Republic of Korea*; Byoungjin Jeon, *Seoul National University of Science and Technology, Republic of Korea*; Hyoung Choi, *Seoul National University of Science and Technology, Republic of Korea*

Parallel simulations of the incompressible flows with free surface were conducted by using unstructured adaptive meshes. The incompressible Navier-Stokes equations were discretized by a splitting finite element method and the free surface was captured by solving the advection equation of a level set method with a least-squares weighted residual method. We employed the direct re-initialization techniques for unstructured meshes to reinitialize level set function near the interface. The meshes were adapted during the computational process by a local mesh refinement strategy based on position of free surface so that the computational time was reduced without the loss of solution accuracy. Benchmarks free surface problems were simulated to assess the accuracy and efficiency of the present code. Present numerical results were comparable with existing experimental and numerical data. Parallelization of the present interface capturing code is essential to simulate a large scale free surface flows. The present adaptive finite element method was parallelized by using domain decomposition method and MPI library.

\* Support from National Research Foundation of Korea grant NRF-2014R1A2A01004879 is gratefully acknowledged.

## A THREE-PHASE NUMERICAL MODEL FOR LIQUID-GAS-GAS FLOWS WITH RELAXATION PROCESSES

Marica Pelanti, *ENSTA ParisTech, France*

We are interested in three-phase flows involving the liquid and vapor phases of one species and a third inert gaseous phase. We describe these flows by a single-velocity multiphase flow model composed of the phasic mass and total energy equations, the volume fraction equations, and the mixture momentum equation. The model includes stiff mechanical and thermal relaxation source terms for all the phases, and chemical relaxation terms to describe mass transfer between the liquid and vapor phases of the species that may undergo transition. The homogeneous hyperbolic portion of the equations is solved numerically via finite volume wave propagation schemes. Relaxation terms are treated by routines that exploit algebraic equilibrium conditions for the relaxed states. We present numerical results for a three-phase cavitation tube test, showing that the behavior of the predicted sound speed for different levels of activation of stiff mechanical and thermo-chemical relaxation processes agrees with the theoretical results for the hierarchy of relaxation models. The simulation of a two-dimensional underwater explosion is also presented.

\* Support from French DGA Grant N. 2012.60.0011.00.470.75.01 and Norwegian RCN Grant N. 234126/E30 is gratefully acknowledged.

## A VARIATIONAL APPROACH TO DESIGN A NUMERICAL SCHEME ON AN ARBITRARY MOVING GRID FOR N-FLUID FLOW WITH THERMODYNAMIC CONSISTENCY

Thibaud Vazquez Gonzalez, *Commissariat à l'Energie Atomique, France*; Antoine Llor, *Commissariat à l'Energie Atomique, France*; Christophe Fochesato, *Commissariat à l'Energie Atomique, France*

In some highly demanding fluid dynamics simulations, as for instance in inertial confinement fusion applications, it appears necessary to simulate multi-fluid flows involving numerous constraints at the same time, such as (and non-limitatively): large numbers of fluids (typically 10 and above), both isentropic and strongly shocked compressible evolutions, large heat sources, large deformations, transport over large distances, and highly variable or contrasted EOS stiffness. Fulfilling such a challenge in a robust and tractable way demands that thermodynamic consistency of the numerical scheme be carefully ensured. This is addressed here over an arbitrarily evolving computational grid (ALE or Arbitrary Lagrange-Euler approach) by a three-step mimicking derivation: i) to ensure a compatible (approximately symplectic) exchange between internal and kinetic energies under isentropic conditions, a variational least action principle is used to generate the proper pressure forces in the momentum equations; ii) to generate the conservative internal energy equation, a tally is performed to match the kinetic energy, and iii) artificial dissipation is added to ensure shock stability, but other physical terms could also be included (drag, heat exchange, etc.). Varied single-, two- and multi-fluid test cases show satisfactory behavior, including the 2D super-sonic, high volume-fraction convergence of eight Gaussian packets of different fluids in a background of perfect gas (under the sole pressure coupling).

## FLOW MEASUREMENT IN OPAQUE FLUIDS USING ULTRASOUND IMAGING VELOCIMETRY

Christian Poelma, *Delft University of Technology, The Netherlands*; Arati Gurung, *Delft University of Technology, The Netherlands*

The majority of modern experimental flow measurement techniques are based on optics: the common approaches involve the use of lasers and cameras, which necessitate a transparent fluid. This means that these methods are not viable for multiphase flows with an appreciable volume load of a dispersed phase ( $>0.5$  vol%), unless they are studied in nearly two-dimensional geometries. With the recent advent of ultrasound imaging velocimetry (a.k.a. "echo-PIV"), a tool has become available that finally allows the study of more densely-laden flows. Although the resolution may not be able to compete with light-based implementations, recent work has shown that it is feasible to accurately determine velocity fields in a wide range of multiphase applications, including e.g. in vivo blood flow measurements and rheology studies of drilling mud. These studies, and others reported in the literature, are generally based only on time-averaged velocity fields. In the current contribution, we investigate the viability of obtaining instantaneous velocity fields that will allow further insight in, for instance, turbulent two-phase flow statistics. We will provide examples of suitable model systems (i.e. combinations of liquid/solid) that can be measured using echo-PIV, such as the aforementioned drilling mud, ketchup (as an example of applications in the food processing industry) and water with glass particles for more fundamental two-phase investigations.



#### **X-RAY RADIOSCOPY AS A POWERFUL TOOL FOR VISUALIZATION OF LIQUID METAL BUBBLY FLOWS**

*Olga Roshchupkina, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Natalia Shevchenko, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Erik Strumpf, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Dietrich Hoppe, Institute of Fluid Dynamics, Germany; Sven Eckert, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany*

Many technical applications in metallurgy and the quality of continuous casting rely on liquid metal two-phase flows. Injection of the Argon gas became an integral part of continuous casting since it prevents clogging of the casting nozzle and also separates alumina particles from the melt. On the other hand, injection of gas has many side effects as for example induction of highly turbulent complex two-phase flows. There exist many numerical simulations and water models, but due to large differences in physical properties between water and liquid metals water models and experiments cannot be fully extended to liquid metals. Therefore, direct investigation and understanding of liquid metal two-phase flows became critical. In the present work we demonstrate that X-ray radiography can be used as a powerful tool for the visualization of liquid metal two-phase flows. Here we present an experimental study of ascending bubble chains over a wide range of gas flow rates in GalSn alloy at room temperature. We report on differences in bubble release frequency, shape, size, velocity etc. and additionally compare with experiments in water. The efficiency of the corresponding measurement technique is primarily validated in water experiments.

\* This research is supported by the German Helmholtz Association in form of the Helmholtz-Alliance "LIMTECH".

#### **A STUDY OF LIQUID-LIQUID SECONDARY FRAGMENTATION WITH SOLIDIFICATION**

*Miloud Hadj-Achour, University of Lorraine, France; Nicolas Rimbert, University of Lorraine, France; Sebastian Castrillon-Escobar, University of Lorraine, France; Michel Gradeck, University of Lorraine, France*

The present work focuses on the fragmentation of a single molten alloy droplet. The liquid droplet interacts with a water pool at instantaneous contact interface temperature between 50°C to 90°C and according to its Weber number, it will be fragmented in a different way. A drop-on-demand device was designed and built for this purpose. For each experiment, a single droplet has been visualized using a high-speed camera (at 8000 fps). All measurements (drop size distribution, velocity, impact parameter and geometrical properties of the drops after the penetration) into the pool are evaluated using an open source image processing. The analysis of the different tests has shown that the solidification conditions can have a strong influence on the final size distribution of fragments. A new mechanism of bag formation is revealed for liquid-liquid fragmentation and the experimental results are compared with those obtained by DNS using the open-source software Gerris.

#### **REMOVAL OF FINE PARTICLES ON A WALL BY HIGH-SPEED AIR JET EJECTED FROM A CAVITY NOZZLE (OPTIMIZATION OF NOZZLE SHAPE)**

*Tatsuro Wakimoto, Osaka City University, Japan; Atsushi Nakagawa, Osaka City University, Japan; Kazuhiko Soemoto, Sinko Co., Ltd., Japan; Kenji Katoh, Osaka City University, Japan*

We developed a cleaning device which removes fine particles adhering on a wall by a high-speed impinging jet with strong fluctuation. In general, it is difficult to remove micrometer-sized particles by a simple air jet because they strongly adhere to the surface by van der Waals forces and remain immersed in a viscous sublayer. In this study, strong velocity and pressure fluctuations are added to the jet using a nozzle with triangular cavities to overcome the problem. In order to clarify the removal mechanism by the fluctuated jet and optimize nozzle shape, we considered hydrodynamic drag and local pressure gradient forces acting on a sphere particle and evaluated removal moment by numerical simulation. The removal moment evaluation for various types of nozzle revealed that the nozzle shape with the shortest slit length and one step cavity is the best for removal due to the strongly enhanced removal moment by velocity and pressure fluctuations.

#### **GAS AND LIQUID DYNAMICS IN BUBBLE COLUMNS: ADVANCED FLOW IMAGING WITH ULTRAFAST X-RAY AND RADIOACTIVE PARTICLE TRACKING**

*Salar Azizi, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Ashutosh Yadav, Indian Institute of Technology, India; Yuk Man Lau, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Shantanu Roy, Indian Institute of Technology, India; Markus Schubert, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany*

The performance of bubble columns depends strongly on the gas and liquid phase flow dynamics. Interactions between the two phases result in certain bubble size distributions and turbulence spectra defining the transfer rates as well as the mixing behavior. Models and experiments, which treat these systems linearly, tend to over-simplify their behavior by resorting to only time and spatially averaged descriptions. Although the employed experimental techniques are rather powerful, today's literature commonly reports hydrodynamic data for gas or liquid phases separately, often for different dimensions, operating conditions, etc. In this work, a body of experimental data is consolidated for a laboratory bubble column using both the complementary techniques of ultrafast X-ray tomography (UXT) and radioactive particle tracking (RPT). Sizes and velocities of the dispersed gas bubbles are measured with UXT, while RPT provides velocity distribution and turbulent kinetic energy of the liquid phase. The results of both techniques have a high spatiotemporal resolution and provide a unique benchmark of experimental validation data of bubble column reactors for CFD models. In particular, the effects of gas flow rate and sparger configuration determining the inlet bubble sizes will be discussed.

\* Support from ERC grant XFLOW (FP-7-307360) is gratefully acknowledged.

#### **EXPERIMENTAL AND COMPUTATIONAL INVESTIGATION OF FLOW INSIDE A MILLING CIRCUIT HYDROCYCLONE**

*John Furlan, GIW Industries, USA; Renjie Ke, Case Western Reserve University, USA; Chinmay Shingote, Case Western Reserve University, USA; Jaikrishnan Kadambi, Case Western Reserve University, USA; Robert Visintainer, GIW Industries, USA*

Hydrocyclone separators are widely used in various industrial, oil and mining operations to sort, classify and separate solid particles or liquid droplets. Limited information on their performance is available in the open literature. The available models are not applicable to many useful complex cyclone geometries. Nearly all industrial hydrocyclones have an underflow which is open to atmosphere, resulting in an air core along the central axis. In this study, the two phase flow system inside a transparent acrylic model (12.7 cm diameter and 94 cm height) of a solid-liquid milling circuit hydrocyclone, with test conditions having a central axis air core, was investigated. Particle Image Velocimetry (PIV) in conjunction with refractive index matching technique was used to obtain the flow field measurements. Star CCM+, a commercial Computational Fluid Dynamics (CFD) code was used for the computational study. Large Eddy Turbulence model was applied with the Volume of Fluid Multi-phase model. The computationally predicted velocity field, shape and diameter of the air core are in reasonably good agreement with the experimental results. The physical time of the air core generation calculated from the simulation approximated the time scale observed in the experiments. These results will be discussed in this paper.

#### **NONINVASIVE TEMPERATURE DISTRIBUTION IMAGING OF HIGH VISCOSITY PLASTIC FLOW BY ELECTRICAL CAPACITANCE TOMOGRAPHY**

*Yusuke Hirose, Chiba University, Japan; Kazuaki Hata, Chiba University, Japan; Masahiro Takei, Chiba University, Japan*

This study has launched a concept to measure real time two-dimensional temperature distribution non-invasively by a combination of electrical capacitance tomography (ECT) technique and a permittivity-temperature equation for melting plastic (polymethyl methacrylate). The concept has two steps which are the relative permittivity calculation from the measured capacitance among the many electrodes by ECT technique, and the temperature distribution calculation from the relative permittivity distribution by permittivity-temperature equation.

ECT sensor with 8-electrode was located an injection molding machine, which is designed to measure and visualize the cross sectional relative permittivity distribution. The images of the relative permittivity distribution are successfully reconstructed at every time step during the process. The images indicate that the relative permittivity of wall side is higher than the center area.

\* Support from grant Adaptable and Seamless Technology Transfer Program through Target-driven R&D, Japan Science and Technology Agency (#AS251Z00116K) is gratefully acknowledged.

## A NUMERICAL CLOSE-UP ON THE SPRAY FORMATION FROM A GAS-LIQUID MIXING LAYER

Yue Ling, *Université Pierre et Marie Curie (Paris 6), France*; Daniel Fuster, *CNRS UMR 7190, France*; Greta Tryggvason, *The University of Notre Dame, USA*; Ruben Scardovelli, *University of Bologna, Italy*; Stephane Zaleski, *Université Pierre et Marie Curie (Paris 6), France*

Spray formation from a gas-liquid mixing layer is commonly seen in nature and industry. In order to understand the detailed mechanisms of spray formation, very large-scale 3D direct numerical simulations (DNS) of a gas-liquid mixing layer are performed. The Navier-Stokes equations for incompressible flows with sharp interfaces are solved using the Volume-of-Fluid method. Despite a wide breadth of theoretical, experimental and numerical studies on spray formation, three-dimensional DNS is less advanced due to the extreme computational cost. In particular questions like whether the spray characteristics (such as size distribution) are fully numerically resolved are often unanswered. In this work, a gas-liquid mixing layer is introduced by two coflowing gas and liquid streams, and simulations of different grid resolution (up to 4 billion cells) are conducted. Particular attention is focused on examining whether we can capture various droplet formation mechanisms and achieve converged results on the statistical spray characteristics. It is observed that the crest of the wave on the interface develops a quasi-singularity, where a very thin sheet and tiny fingers are formed. Tiny droplets are detached from these small fingers. But as time evolves, the small fingers merge forming larger ligaments, which in turn breaks into larger droplets. This mechanism is reminiscent of an inverse energy cascade or of droplet impact, in which small structures appear first and larger ones later.

\* Support from the MODEMI project (ANR-11-MONU-0011) is gratefully acknowledged.

## A CONSISTENT MOMENTUM TRANSPORT IN CLSVOF METHOD: VALIDATION ON JET ATOMIZATION

Alain Berlemont, *CNRS, France*; Thibaut Menard, *Normandie Université, France*; Wojciek Aniszewski, *Normandie Université, France*; Geoffroy Vaudor, *France*

Many atomization processes are characterized by large density ratios coupled with large shear. Numerical simulations are known for becoming unstable in these physical conditions. We thus developed a new mass and momentum consistent algorithm for CLSVOF method. Based on Rudman's work, the originality of our approach is to avoid using a sub-grid by splitting flux calculation between neighboring cells. The method is validated by comparing simulations and theoretical results on droplet transport with 106 density ratio, spurious current and 2D air assisted atomization of a liquid layer.

We then focused on air assisted atomization: the studied case is 7.6 mm cylindrical water jet surrounded by a 1.7mm thick air flow. Liquid velocity is 0.27m/s and gas velocity is 22.6m/s. The grid is 512x512x1024, the mesh is equal to 66µm and massively parallel computing is used (8192 procs). Boundary layers are taken into account in the initial profiles and proved to be of large influence. Qualitative agreement is observed by comparing snapshots from the experiment and the simulation. Quantitative comparisons are carried out on the wave length of the primary instability, the jet expansion and the length of the liquid core. We then consider high speed (200m/s) Diesel jet atomization in quiet air. Comparisons concern the interface velocities which are experimentally measured by an ultrafast shadow imaging (double pulsed femto second system). Improvements of the new scheme are clearly validated.

## EFFECT OF THE ALIGNED FLOW OBSTACLES ON SUBCOOLED FLOW BOILING IN AN INCLINED RECTANGULAR CHANNEL

Uiju Jeong, *Hanyang University, Republic of Korea*; Hong Hyun Son, *Hanyang University, Republic of Korea*; Kyusang Song, *Hanyang University, Republic of Korea*; Gwang Hyeok Seo, *Hanyang University, Republic of Korea*; Joon Goo Jeon, *Hanyang University, Republic of Korea*; Sung Joong Kim, *Hanyang University, Republic of Korea*

In the core catcher system suggested for APR1400, a key design component is the engineered corium cooling system, consisting of an inclined engineered cooling channel made of a single channel between the corium spreading compartment and inside wall of the cavity, and many short columnar structures, called stud, supporting the loading on the core catcher body. To investigate the heat transfer capability of the channel with the aligned studs, scaled experiments for subcooled flow boiling heat transfer with downward facing heat transfer surface have been conducted. This study is mainly focused on a visual study on water flow boiling in an inclined (100) rectangular channel containing two studs in a line. A test section scaled down by one-fourth has been designed and manufactured. The experimental conditions adopted are mass fluxes (50-300kg/m<sup>2</sup>s) and subcooling of 5-20K at the channel with dimensions of 30×118×215mm using indirectly heated flat plate, made of SS316, with active heated area of 107.5×215 mm. With these experiments, effect of the aligned studs with variation of shape and mass flux on the boiling heat transfer characteristics including CHF has been examined carefully. It is observed that the studs hinder the bubble venting process, and the CHF of rectangular-type stud is lower than the bare CHF. The resulting stagnant region of the vapor between the aligned studs is the most probable cause of the significantly decreased CHF.

\* Support from grant NRF-2014M2B2A9032081 is gratefully acknowledged.

## INFLUENCE OF DISSOLVED GASES ON ATOMIZATION PROCESSES

Emmanuelle Tinon, *IMFT, France*; Charles-Etienne Martin, *Airbus Operations SAS, France*; Rudy Bazile, *Université de Toulouse, IMFT, CNRS, France*; Laurent Selle, *IMFT, France*

The atomization of a liquid is a crucial process for many industrial applications. For power generation and transportation, atomization controls the evaporation of the fuel and ultimately, combustion performance and pollutant emissions. In other fields (painting, fire protection, disinfection, etc.), the control of the spray characteristics (droplets' sizes and velocity) is mandatory for the overall efficiency of the process.

For this purpose there is a wide range of atomizer geometries, from simple holes to swirling or impinging jets. Moreover, the use of a secondary airflow (co- or cross-flow) may enhance shear forces and favor the destabilization of the liquid jet. Injecting a gaseous phase prior to the nozzle exit (a.k.a. aerated atomization) also modifies the spray characteristics.

Here, we explore experimentally the behavior of a simple-hole atomizer, fed with a liquid saturated with gas. Its high solubility allows for more than 20% in mass to be dissolved in the liquid to be atomized. The resulting spray is massively affected by the rapid release of the dissolved gas during the discharge. The governing parameters of this phenomenon, referred to as 'effervescent atomization', are sought. The flow both upstream and downstream the nozzle exit is scrutinized with high-speed imaging and laser diagnostics.

## ON THE GAS-LIQUID ENTRAINMENT BY IMPINGEMENT OF LIQUID JET ONTO A POOL

Vatsal Sanjay, *Indian Institute of Technology, Roorkee, India*; Arup Kumar Das, *Indian Institute of Technology Roorkee, India, India*

Air entrainment due to impingement of liquid jet in a pool is studied extensively by full scale experiments and volume of fluid (VOF) based numerical simulations. Based on high speed image analysis and conductivity probe signals, entrainment pattern is examined for a wide range of jet velocity and diameter (10000<Re<50000). Transition boundary between continuous and no entrainment has been characterised from experimental observations. Onset of entrainment and its unsteady dynamics have been revealed using experimental snapshots and complementing simulation contours. Signals from electrical conductivity probe reveals that though the entrainment is dynamic in nature, progressively increasing length of steady entrainment is observed for increasing Reynolds number. Triangular entrained region is found to be a three dimensional cluster of bubble population continuously breaking and making with neighbours due to turbulent dispersion induced by jet. During entrainment, free surface of the pool shows undulations which subsequently releases droplets of different sizes around the jet. The trajectories of the droplets are driven by jet inertia and subsequent gravitational interactions. Findings from the present study can be extrapolated for understanding the knowledge of gas-liquid reactors, heat and mass transfer applications as well as mixing dynamics.

## HEAT TRANSFER CHARACTERISTICS OF SUBCOOLED WATER FLOW BOILING IN CIRCULAR CHANNELS WITH NONUNIFORM HEATING FLUXES

Ge Zhu, *Xi'an Jiaotong University, China*; Qincheng Bi, *Xi'an Jiaotong University, China*; Jianguo Yan, *Xi'an Jiaotong University, China*; Qizheng Yuan, *Xi'an Jiaotong University, China*; Haicai Lyu, *Xi'an Jiaotong University, China*; Hui Pan, *Xi'an Jiaotong University, China*

Experiments of heat transfer characteristics of subcooled water flowing in vertical circular channels, which were offcenter in rectangular blocks, were carried out under high heat fluxes up to the ITER requirements. The heating fluxes distributions of the channels were nonuniform in the circumferential direction, which were obtained by electrically heating the blocks directly. Two types of channels were used: smooth channel and swirl tapes channel. The surface temperature of the rectangular blocks was measured by infrared camera and thermocouples. Effect of the system pressure, mass flow rate, inlet subcooling, and equivalent heat fluxes on heat transfer were all investigated. The main attentions were paid to the subcooled water heat transfer under nonuniform heating flux, and the effect of swirl tapes. Results shows that subcooled boiling is more likely to become the dominant factor under the conditions of lower mass flow rates, higher heat fluxes and lower system pressures. Swirl tapes can enhance the heat transfer, which is more clear in high heat fluxes. The temperature fields in the block were calculated with a Computational Fluid Dynamics(CFD) method to obtain, which were consistent with the experimental results.

Keywords: heat transfer, swirl tape, nonuniform heat flux, ITER

#### A STUDY ON THE BEHAVIOR OF MECHANISTIC CRITICAL HEAT FLUX MODELS FOR FLOW BOILING AT AN ISOTHERMAL WALL USING A REFRIGERANT

Moritz Bruder, Technische Universität München, Germany; Norbert Heublein, Technische Universität München, Germany; Thomas Sattelmayer, Technische Universität München, Germany

A study on the behavior of mechanistic critical heat flux models is presented. Established boiling models were numerically implemented in a Matlab programming environment. Their behavior was analyzed both for original sources used by the authors in their original publications and for own experiments. Experiments were conducted near critical heat flux and during the critical heat flux transient in a rectangular flow channel with a large copper heater mounted flush into one of the channel's walls. The fluid used is 3M's Novec 649, a fluorinated ketone with a low boiling point. A range of subcooling of 5 K to 27 K at different mass fluxes was investigated. In this paper, the critical heat flux data from these experiments is compared to the models' predictions. Selected predicted parameters for flow topology and vapor behavior are compared to experimental results using synchronized high-speed videometry and fiber optic needle probes. It was observed that the models' performance is deteriorated when using a fluid different than the original fluids for which the respective models were developed. In contrast to the general performance of each model, several parameters of individual submodels showed reasonable agreement with the experimental data.

This work was funded by the German Federal Ministry of Economic Affairs and Energy under grant number 1501473A on the basis of a decision by the German Bundestag.

#### INVESTIGATION ON THE DEFORMATION BEHAVIOR BETWEEN OIL DROPLETS IN AQUEOUS SOLUTION

Fangyuan Liu, China University of Petroleum, Beijing, China; Wei Wang, China University of Petroleum Beijing, China; Hang Jin, China University of Petroleum, Beijing, China; Kai Li, Institute of Mechanics, Chinese Academy of Sciences, China; Hongli Chang, China University of Petroleum, Beijing, China; Jing Gong, China University of Petroleum Beijing, China

The study of deformation behavior between two dispersed drops can provide insight into the stability of emulsion. An experiment using AFM to measure the force between two oil droplets at different concentrations of oil soluble surfactant (Sorbitan monooleate, Span 80) in water is carried out and the influence of surfactant concentration and electrical double layer (EDL) repulsion is discussed in detail. The range of Span 80 concentrations from far below the Critical Micelle Concentration (CMC) to above the CMC is analyzed. The slope of force curve can reflect the deformability of droplets. It is found that the slopes first increase and then decrease with the increase of the surfactant concentration, which displays similar tendency with the effect of surfactant on interfacial dilational rheology. Since the oil surface in de-ionized water is negatively charged, it will retard drop coalescence and influence the drop deformation behavior. To characterize the coupling effect of EDL and surfactant on drop deformation, sodium chloride is added to quantitatively adjust the EDL repulsion force, where a trigger drop coalesce is observed at higher sodium chloride concentration reflecting the inner nature of emulsion stability.

\* Support from the National Natural Science Foundation of China(51422406) is gratefully acknowledged.

#### DROP DEFORMATION IN VISCOUS COMPRESSIONAL FLOW: FLAT AND TOROIDAL DROPS

Avinoam Nir, Technion - Israel Institute of Technology, Israel; Olga Lavrenteva, Technion - Israel Institute of Technology, Israel; Irina Smagin, Technion - Israel Institute of Technology, Israel; Michael Zabarankin, Stevens Institute of Technology, USA

We report the deformation of viscous drops embedded in an immiscible viscous fluid under linear compressional flow. The dynamics of the deformation can lead to flat drops or to toroidal shapes. For phases with equal viscosity, it is shown that stationary shapes are predicted when the capillary number  $Ca < 0.2$ . Thus, the two shapes present two branches of the same flow problem. It was found that, in this range of  $Ca$ , stationary flat drops are stable, while stationary toroidal drops are unstable. The dynamics of the toroidal deformation can lead to collapse torus toward the center or to its indefinite expansion. Comparison of the prediction is made with available experimental observation of toroidal collapse in the limit of vanishing capillary number. At  $Ca > 0.2$  no stationary shapes are predicted in the compressional flow. All drops are unstable. The flat drop either breaks up or evolves into a toroidal shape, the radius of which expands indefinitely or until torus disintegration occurs.

\*Supported by Israel Science Foundation Grant 327/14

#### CONVECTIVE LOCAL HEAT TRANSFER AROUND ELONGATED BUBBLES PROPAGATING IN DOWNWARD LIQUID PIPE FLOW

Adam Fershtman, Tel-aviv University, Israel; Dvora Barnea, Tel-aviv University, Israel; Lev Shemer, Tel-aviv University, Israel

The motion of an axisymmetric Taylor bubble propagating in stagnant and upward-flowing liquid is well documented. However, only a number of studies have been conducted in downward liquid flow. In this complex flow regime, the bubble shape can become asymmetric. In previous experiments carried out in our laboratory 3 different bubble propagation regimes were identified for a given downward background flow: (1) negative translational velocity with a symmetric bubble nose, (2) fast upward moving bubble that has a strongly distorted nose that almost touches the pipe wall, and (3) an intermediate case with a slower upward bubble velocity and a nose that is distorted and shifted away from the pipe axis. The first two regimes are unstable; initially the 1st regime undergoes transition to the 2nd one; subsequently the bubble nose moves away from the wall, so that the transition to the stable 3rd steady regime is observed. The different bubble shapes and correspondingly notably different bubble velocities have a considerable effect on the flow field in the bubble's wake and thus on the convective heat transfer rate. Thermography technique has been applied to measure the local instantaneous heat transfer rate for all three bubble propagation regimes in downward flow.

#### EFFECT OF LIQUID PROPERTIES ON THE INTERFACIAL INSTABILITY AND ATOMIZATION BEHAVIOR OF THE ACOUSTICALLY LEVITATED DROPLET

Kengo Kono, Kogakuin University, Japan; Koji Hasegawa, Kogakuin University, Japan; Hiroaki Imai, Kogakuin University, Japan; Hiroyasu Ohtake, Kogakuin University, Japan; Atsushi Goda, University of Tsukuba, Japan; Yutaka Abe, University of Tsukuba, Japan

Acoustic levitation is based on the production of a standing wave with equally spaced nodes and antinodes between the horn and reflector. This levitation technique is expected to be used in the field of the analytical chemistry and manufacturing new materials in order to prevent the heterogeneous nucleation and contamination by container-less processing. It is vital to understand quantitatively the non-linear dynamic behavior of the interface deformation and translational motion of the acoustically levitated droplet in order to apply acoustic levitation method for such various scientific fields. The purpose of our study is to investigate experimentally the interfacial behavior and atomization of the droplet. In this paper, we recorded the interfacial deformation and atomization process of an acoustically levitated droplet by a high-speed video camera. We estimated the dynamics of the droplet quantitatively with water, glycerin, and ethanol and compared with the existing experimental correlation by Danilov. et al (1992). It is shown that the mechanism of the atomization of the levitated droplet is more complicated than the existing model indicated by the theory and moreover, the surface tension and viscosity of the levitated droplet would be related to its atomization behavior.

#### HOMOGENIZATION: SIMPLIFIED DROP DEFORMATION ESTIMATE DURING TRANSITION THROUGH ORIFICES

Peter Walzel, TU-Dortmund, Germany

In high-pressure homogenization usually a pre-emulsion with large primary droplets is forced through circular orifices or slots with high overhead pressure. The drops are stretched on their way through the orifice due to the extensional flow and depending on their size, they may be deformed to about prolate ellipsoids or even to threads. The local deformation depends on the flow forces acting on the drop surface. Depending on the viscosity of the continuous phase liquid, typical Reynolds numbers of the flow around the droplets for practical cases lie in the range of  $10 < Re < 500$ . This leads to comparable magnitudes of viscous as well as inertia forces acting on the drop surface. Neglecting internal flow within the drops, the local external forces can be determined and are in equilibrium to the capillary forces of the deformed drops. The calculation of the forces requires knowledge of the flow field upstream to the orifice. The calculations were performed for rounded nozzle inlets assuming non-detached sink flow. It can be shown that the ratio of primary drop to orifice diameter plays a significant role in the deformation at given overhead pressures, i.e. corresponding transition velocities. Small drops are hardly deformed while large drops are significantly deformed and may fall apart into secondary droplets upon transition. Too small drops may relax their shape without breakup and a minimum elongation seems to be necessary for breakup into secondary droplets. Even though some constraints had to be implemented in the simplified calculation, new insight into the breakup process is presented.

## TWO WAY COUPLING IN PARTICLE-LADEN TURBULENT JET

Francesco Battista, *Sapienza Università di Roma, Italy*; Paolo Gualtieri, *Sapienza Università di Roma, Italy*; Giorgia Sinibaldi, *University of Rome La Sapienza, Italy*; Carlo Massimo Casciola, *Sapienza Università di Roma, Italy*  
 Particle-laden turbulent jets are typical of several natural phenomena and technological applications. In particular, when the particles are much smaller than the characteristic fluid dynamical scale, i.e. the Kolmogorov scale in a turbulent flow, and when their mass loading is large enough, the disperse phase eventually effects the particle-to-fluid momentum exchange (two-way coupling regime).

Aim of the present work is to address the effects of the fluid-particle momentum coupling on the dynamics of a turbulent jet. In fact, the jet spreading angle and jet mean velocity decay rate are expected to be controlled by the particle inertia and by the overall mass loading. To this purpose, the Direct Numerical Simulation (DNS) of a turbulent jet laden with inertial particle is performed. The incompressible Navier-Stokes equations are solved in a cylindrical domain to obtain the evolution of the turbulent carrier flow, while a classical Lagrangian approach is used to track each single particle seeding the jet. Since the particles are assumed to be smaller than the Kolmogorov scale, their dynamics is controlled by the Stokes number. The particle-to-fluid inter-phase momentum coupling is modelled by the Exact Regularized Point Particle (ERPP) method. The talk will address both data obtained by a reference DNS in the one-way coupling regime (no back-reaction on the fluid) and data pertaining the two-way coupling regime to highlight the modification of the turbulent flow due to the particle's back-reaction.

## TWO-WAY INTERACTIONS IN PARTICLE-LADEN TURBULENT CHANNEL FLOW

Cheng Peng, *University of Delaware, USA*; Lian-Ping Wang, *University of Delaware, USA*

In this work we present results on two-way interactions of finite-size particles and fluid turbulence in a channel flow. The lattice Boltzmann approach is used to resolve both the channel flow and the disturbance flows around moving particles. Three aspects will be reported. First, results of single-phase turbulent channel flows are compared to published benchmark DNS results to validate the lattice Boltzmann approach. Second, the accuracy of the simulation method in treating the physics at the solid-fluid interfaces is discussed using several benchmark cases including the Segre-Silberberg effect in a laminar channel flow and a fixed particle in a turbulent channel flow. Finally, results on turbulent particle-laden channel flow are analyzed at several levels: whole-field, phase-partitioned, and profiles as a function of distance from the surface of solid particles. We examine the effects of finite particle size on the mechanisms of energy production and dissipation. Specifically, the two-way interactions near the channel wall are contrasted with those away from the walls. We find that finite-size particles enhance mixing of turbulent fluctuations in the viscous sublayer and the bulk flow in the center region of the channel. Results are compared to those based on the point particle approach. We will also discuss how the results change with particle size, particle-to-fluid density ratio, and particle volume fraction.

## MODELLING OF CHURN-ANNULAR FOAM FLOWS

Jos Westende, *TNO, The Netherlands*; Pejman Shoeibiromani, *TNO, The Netherlands*; Frank Vercauteren, *TNO, The Netherlands*; Erik Nennie, *TNO, The Netherlands*

Foam assisted lift is a deliquification method in the oil and gas industry, which aims to prevent or postpone countercurrent gas-liquid flow in maturing gas wells or to assist in removing downhole accumulated liquids. According to Nimwegen (2015), who performed experiments with foam flows, foam accomplishes this, since the creation of foam postpones the transition from annular to churn flow to lower gas velocities, which improves the upward transport of liquid carried by foam. This paper presents a model predicting the foam flow behavior in annular flow conditions down to the transition from annular to churn flow. The model treats the foam flow as a film flow, similar to that described by, e.g., Whalley (1987). Closure relations for the foam density, foam viscosity and interfacial friction factor are derived from the measurements performed by Nimwegen. The dependencies found in these closures are coupled to physical phenomena that occur in annular flows or compared to models available in the literature. The model and the newly found closure relations are used to reconstruct the pressure gradient as a function of the gas velocity for different foamer concentration and compared with experimental data.

\* Support from grant TKIG01022 and participating partners is gratefully acknowledged.

## AN EXPERIMENTAL INVESTIGATION OF THE THREE-WAY COUPLING BETWEEN TURBULENCE, PARTICLES, AND RADIATION

Andrew Banko, *Stanford University, USA*; Laura Villafane, *Stanford University, USA*; Christopher Elkins, *Stanford University, USA*; John Eaton, *Stanford University, USA*

We study the coupled dynamics of a fully developed turbulent particle-laden flow subjected to radiative heating. The flow of air with dispersed Nickel particles in a channel of square cross-section is exposed to monochromatic near infrared radiation through one wall of the channel. While the gas and channel walls are nearly transparent, the particles absorb radiation and heat the gas with a spatial distribution influenced by the local particle clustering. With sufficient radiation intensity and particle concentration, the fluid momentum and thermal energy become cyclically coupled to the local particle dynamics and their radiation absorption. In the present experiment, the bulk Reynolds number is 104, the particles are smaller than all flow length scales and have a Kolmogorov based Stokes number of order unity. The mass loading ratio of particles to air varies from 0.01 to 0.4, spanning the one-way and two-way coupled regimes. The illumination system is a custom laser diode array outputting up to 10 kW of optical power. Integral measurements of total energy absorption, mean gas temperature and velocity profiles, and particle velocity and clustering statistics are reported using a variety of probe based and optical techniques for several particle loadings and illumination intensities

## MEASUREMENTS OF PARTICLE SETTLING VELOCITY IN HOMOGENEOUS TURBULENCE WITH NO MEAN FLOW

Filippo Coletti, *University of Minnesota, USA*; Alec Petersen, *University of Minnesota, USA*; Douglas Carter, *University of Minnesota, USA*; Lucia Baker, *University of Minnesota, USA*

The fall speed of heavy particles suspended in a turbulent flow is an important parameter in numerous natural phenomena and industrial applications. Nevertheless, the question of how the presence and strength of turbulence affect the particle settling process remains still unanswered. Several mechanisms have been identified which may either hinder or enhance the settling rate, but different studies often show qualitative and quantitative discrepancies. We study experimentally the case of a dilute suspension of size-selected microscopic particles falling through air. The mass loading is low enough to neglect the influence of the dispersed phase on the air motion. We use a novel apparatus in which randomly actuated air jets create a volume of homogeneous turbulence with no mean shear and negligible mean flow. In contrast with most previous experimental studies, the homogeneous region is several times larger than the integral length scale, allowing the particles to sample eddy motions of all scales. The turbulence intensity is gradually increased by altering the jet actuation pattern, and because the integral length scale is kept approximately constant, the Kolmogorov scale become progressively smaller. We use particle imaging to measure the average and fluctuating settling velocities for the different cases, and discuss the compatibility of the observations with the various proposed mechanisms of settling alteration by turbulence.

## SOME CHARACTERISTICS OF LIQUID DROPLETS IN VERTICAL UPWARD ANNULAR-MIST FLOW

Yuchen Wang, *Tokyo University of Marine Science and Technology, Japan*; Samwel Andayi, *Tokyo University of Marine Science and Technology, Japan*; Tatsuya Hazuku, *Tokyo University of Marine Science and Technology, Japan*; Tomoji Takamasa, *Tokyo University of Marine Science and Technology, Japan*

Accurate knowledge of flow characteristics of the entrained droplets in annular two-phase flow is essential for proper thermal-hydraulic design and safety analysis of plants. While a majority of two-phase flow characteristics can now be predicted numerically due to recent advances in numerical simulation techniques, reliable experimental data concerning the instantaneous local characteristics of two-phase flow structures are required to confirm the validity of these simulations. In this study, measurements of droplet size and velocity in vertical upward annular-mist flow flowing up a vertical pipe with 10 mm-ID were performed using an image-processing based on a shadowgraph technique. Transient characteristics of droplet flux and velocity were also evaluated by means of the present technique.

Total 9 datasets were obtained at the predetermined superficial liquid and gas velocities, ranging from 20 to 40 m/s for gas phase and from 0.1 to 0.4 m/s for liquid phase. The measured droplet size distribution varied greatly with change in gas flow rate, while its weak dependence on liquid flow rate at the fixed gas flow rate was confirmed. The measured droplet velocity profiles in pipe cross-section tended to obey the one-seventh-power law. Insignificant local slip velocity between gas phase and droplets was confirmed, while its average value tended to slightly increase with droplet size.

## IMPACT OF GAS VELOCITY PROFILE ON SIMULATED WAVE FREQUENCIES IN VERTICAL AIR-WATER CHURN FLOW

Matej Tekavcic, Jožef Stefan Institute, Slovenia; Bostjan Koncar, Jožef Stefan Institute, Slovenia; Ivo Kljenak, Jožef Stefan Institute, Slovenia

In two-phase flow of gas and liquid in vertical pipes, the churn flow regime can be viewed as a transitional regime between slug and annular flow, where large waves of liquid travelling upwards can typically be observed. The mechanisms of the flow are related to the onset of flooding phenomena which is of particular interest during loss-of-coolant accidents in pressurized water nuclear reactors. One of our previous attempts to model such flooding waves with multiphase computational fluid dynamics approach showed a systematic over-prediction of wave frequencies compared to the values from the experiments available in the open literature. This can be attributed to the modelling of the gas flow, i.e. the distribution of the gas velocity in the pipe. In the present study, the impact of gas velocity profile on the wave dynamics is therefore investigated. A short section of a vertical pipe around the liquid inlet was simulated representing a typical experimental section, where large waves of liquid are formed and travel downstream. In order to quantify the impact of gas flow modelling on the simulated wave frequencies, a sensitivity study was performed using different imposed gas velocity distributions at the inlet ranging from a flat uniform profile, turbulent logarithmic profile, up to the parabolic velocity profile of the laminar flow.

## ON THE DYNAMICS OF NON-CONDENSABLE GAS IN A DIESEL INJECTOR

Marco Arienti, Sandia National Laboratories, USA; Mark Sussman, Florida State University, USA

Recent experimental observations of internal flow in Diesel injectors reveal that, at the end of injection, gas from the combustor chamber may be sucked into the sac in a mechanism that could be relevant to cycle-to-cycle variations in the engine. The dynamics of gas bubbles at these conditions is illustrated with the sharp-interface capturing method CLSVOF with the addition of compressible effects in both the liquid and the gas phases and the capability to simulate moving wall boundaries. For instance, high-fidelity simulations that seamlessly connect the internal to the external flow of the injector show that, at the beginning of the following injection cycle, the trapped gas is expelled in an under-expanded jet and contributes to the formation of precursor droplets in the orifice, therefore determining the delay with which the fuel jet appears at the injector's exit.

\* Support by Sandia National Laboratories' LDRD (Laboratory Directed Research and Development) is gratefully acknowledged.

## INTERACTION OF CLEAN OR CONTAMINATED SPHERICAL PAIR BUBBLES IN QUIESCENT LIQUIDS

Toshiyuki Sanada, Shizuoka University, Japan; Hiroaki Kusuno, Shizuoka University, Japan

We experimentally investigated the motion of a pair of bubbles initially positioned in-line configuration in ultrapure water or an aqueous surfactant solution. The bubble motion were observed by two high speed video cameras. The Reynolds number of bubbles were ranged from 50 to 300, and the bubbles hold a spherical shape in this range. In ultrapure water, initially the trailing bubble deviated from the vertical line on the leading bubble owing to the wake of the leading bubble. And then, the slight difference of the bubble radius changed the relative motion. When the trailing bubble slightly larger than the leading bubble, the trailing bubble approached to the leading bubble due to its buoyancy difference. The bubbles attracted and collided only when the bubbles rising approximately side by side configuration. This motion is completely different with particles which is approaching by in the wake of leading bubble. On the other hand, the trailing bubble was drafted to the wake region of leading bubble when the bubbles were rising in an aqueous surfactant solution. We consider that these phenomena are caused by the changes of lift force direction in the wake region owing to the surface contamination.

## PRESSURE GRADIENT VARIATIONS IN GAS-LIQUID DOWNWARD FLOWS IN A VERTICAL LARGE DIAMETER PIPE

Aliyu Aliyu, Cranfield University, UK; Liyun Lao, Cranfield University, UK; Hoi Yeung, Cranfield University, UK

In comparison to upwards two-phase flows, the behaviour of downwards gas/liquid flows in large pipes is poorly understood and data is scarce in the open literature. To attempt to address this, experiments were conducted in a large diameter flow loop with a serpentine configuration. Measurements showed that the pressure gradients over the upper and lower pipe sections of the downward leg produced a unique and consistent pattern when plotted against the superficial gas velocity. Large variations from negative to positive gradients (a so-called "pressure drop reversal") was observed between 5–10 m/s superficial gas velocities. At this critical range, measured liquid film thickness peaking was observed plus a distinct signature of the gas void fraction probability density function. An examination was carried out on whether these indicate flow regime transitions or merely a change in shear occasioned by changing slip at the gas-liquid interface. Further insight was gained by performing spectral analysis on the time-varying pressure signals. The findings here could have important implications for the design/operation of downflow large pipe systems deployed in the oil and gas/nuclear /heat exchange process industries.

\*Support from PTDF Nigeria for Aliyu's PhD is gratefully acknowledged.

## MODELLING DROP OR BUBBLE BREAKUP IN TURBULENT FLOWS

Benjamin Lalanne, LGC - University of Toulouse, France; Olivier Masbernat, Tlemoine, France; Frederic Risso, IMFT- CNRS/Université de Toulouse, France

We consider the problem of drop breakup in turbulent carrier flows, with initial diameters lying in the inertial range. Drop deformation is due to continuous interactions with turbulent vortices; the drop responds to these interactions by performing shape-oscillations, and breakup occurs when the deformation is critical. This coupling between turbulent fluctuations and drop interface dynamics describes the deformation process, provided (i) the turbulent fluctuations at the scale of the drop and (ii) the time scales of its shape-oscillations are known.

Following this idea, we model the deformation process of a drop in a homogenous turbulent flow by a forced oscillator. Our model requires the interface dynamics time scales to be known: they are given by the linear theory of Miller and Scriven for a drop in the absence of gravity or any adsorbed surfactants, and they can be measured or obtained by DNS in more complex configurations.

We show that our deformation model is able to predict accurately drop breakup statistics in terms of breakup probability, by comparison with experimental data. Based on this model, the drop breakup frequency is shown to scale as a power law of a Weber number based on the power spectral density of the turbulent forcing taken at the eigenfrequency of the drop.

## ENTRAINMENT CAUSED BY A SINGLE BUBBLE THROUGH LIQUID-LIQUID INTERFACE IN UNCONFINED SPACE

Abhimanyu Kar, IIT Kharagpur, India; Prasanta Kumar Das, Indian Institute of Technology Kharagpur, India

Rise of small gas bubbles through liquid-liquid interface has been numerically simulated only in a handful of studies, none of which have focused on the entrainment phenomena. In this article, experimental investigation on the rise of single bubbles of different size across the interface of two liquids, water and PDMS contained in a sufficiently large tank is described. As the bubble passes from water (the heavier liquid) to PDMS(the lighter liquid), it entrains a mass of water. The entrained water forms a thin film around the bubble and also forms a stem that connects to the bulk of water below. The entrained mass stretches and narrows with the rise of the bubble. Given sufficient length of rise, a necking forms towards the top of the stem eventually separating the top portion which rises upwards, pendulously swinging, as the bubble rises its wavy path. The entrainment volume of water increases with increasing bubble volume for small bubbles and then plateaus. However, as the entrained mass grows wider with the bubble size, even for larger bubble volumes, it decreases in height eventually to maintain the entrained volume (and mass) invariant of bubble size. In the full length paper the unique hydrodynamics has been revealed with the help of image processing. The process of entrainment has been captured and effort has been made to explain the mechanism.

## A PENALIZATION METHOD FOR THE SIMULATION OF BUBBLY FLOWS

Antoine Morente, EDF R&D / Institut de Mécanique des Fluides de Toulouse, France; Jerome Lavieville, EDF R-D, France; Dominique Legendre, Institut de Mécanique des Fluides de Toulouse, France

We present a  $L^2$  penalization method developed for the simulation of two-phase liquid-bubble flows. The chosen Euler-Lagrange framework involves spherical and non-deformable bubbles represented as moving penalized obstacles interacting with the fluid. A Volume of Fluid method is used to track the phase function while a discretization of the incompressible mass and momentum Navier-Stokes equations is performed with slip conditions at the liquid-bubble interface. The numerical validation process of the built tool is twofold. We first consider static configurations with different penalized obstacles shapes in which either an analytical solution is given or results from simulations carried out by directly meshing the obstacle. Then we consider dynamic configurations: omitting the action of the fluid on the bubble, penalized objects with different shapes are moving through the fluid at a given velocity. In some cases, an analytical solution is available. The results show the compliance of our numerical closures. We propose a formulation of the hydrodynamic forces acting on a bubble based on a direct use of the penalized problem. To verify the accuracy of the coupling method, numerical simulations of bubbles rising in a quiescent liquid are performed and compared with a DNS approach.

## TURBULENT FLOWS WITH COALESCING MICROBUBBLES: AN EULER-LAGRANGE APPROACH BASED ON LARGE-EDDY SIMULATION

Felix Hoppe, Helmut-Schmidt University Hamburg, Germany; Michael Breuer, Helmut-Schmidt University Hamburg, Germany

In order to simulate complex turbulent flows eddy-resolving methodologies such as the large-eddy simulation (LES) technique have become quite popular for single-phase flows. Since an accurate description of the continuous flow is also a main ingredient for an accurate two-phase flow prediction, the LES approach is extended by a Lagrangian method in order to track a huge number of microbubbles within a turbulent flow field. Based on an efficient bubble tracking scheme and an efficient detection scheme for the bubble collisions in a recent study the coalescence of microbubbles is incorporated into simulation methodology. The question whether a collision leads to coalescence or not is modeled by estimating the film drainage time and the bubble contact time. If the latter is sufficiently long to allow the complete drainage of the film, coalescence is assumed. Thus the entire simulation methodology is based on first principles and does not rely on empirical assumptions for the collision rate and the coalescence efficiency as typically used in population balances. It is first applied to study upward and downward turbulent channel flows with varying volume loadings of microbubbles. The disperse and the continuous phase are analyzed in detail with special emphasis on the coalescence rate.

## SIMULATION OF BUBBLY FLOW IN AN INDUSTRIAL ELECTROCHEMICAL REACTOR

Mehdi Arjmand, AkzoNobel, Sweden; Bert Vreman, AKZO Nobel, The Netherlands; Albert Bokkers, AkzoNobel, The Netherlands; Johan Lif, AkzoNobel, Sweden; Kalle Pelin, AkzoNobel, Sweden

The hydrodynamics of a bubbly flow in an electrochemical reactor of industrial size is investigated. The generated gas bubbles induce a recirculation of liquid inside the reactor, due to the so-called airlift principle. Numerical simulations of the flow have been performed using the two-fluid modeling approach. An important input parameter of the model is the bubble diameter. The dependency of the flow on the prescribed bubble diameter has been investigated in detail. Bubbles in electrolyte systems are relatively small (typically 100  $\mu\text{m}$ ), since the presence of strong electrolytes inhibits coalescence. It appears that for a realistic gas production, gas accumulates and no statistically stationary state is reached in the simulations, if the bubble diameter is set to 100  $\mu\text{m}$ . This problem is attributed to the standard degassing boundary conditions at the free surface of the reactor, in combination with a small bubble size and large gas production. Therefore a new procedure is proposed to model the degassing process in the top region of the reactor. With the new procedure, a statistically stationary state is reached in the simulations, while a significant amount of gas recirculation is maintained. The simulations have been performed with two different commercial computational fluid dynamics packages: ANSYS-CFX and COMSOL. The sensitivity to the grid size and the drag closure model has also been studied.

## EFFECT OF INLET CONDITIONS ON TWO PHASE FLOW PATTERN PREDICTION IN SMALL TUBES

Jurij Gregorc, University of Ljubljana, Slovenia; Iztok Zun, University of Ljubljana, Slovenia

To better predict transitions between the various two-phase flow patterns, it is necessary to update our way of thinking from one-of-a-kind flow pattern maps of limited applicability to a generalized approach that includes multiscale information about the flow itself. This would require combined numerical and experimental approach to identify and understand the governing phenomena. In our previous work published in 2013 (Chem Eng Sci 102:106-120), a comparison of the distribution of bubbles with equivalent diameter revealed the inlet mixer's strong impact on bubble size and bubble distribution along the horizontal mini-tube of 1.2mm ID and thus, the bubble to slug flow transition. In order to further clarify the effect of inlet conditions on flow patterns we performed a systematic study on tubes with 10, 6, 3, 1.2, 1 and 0.8mm ID. Focus was on description of inlet conditions on resulting flow patterns evolution downstream the tube at different air and water flow rates at ambient pressure and temperature. Series of numerical simulations were performed to check the numerical capabilities for corresponding flow pattern prediction utilizing VOF in commercial code ANSYS Fluent. Three major flow patterns are considered: bubbly flow, slug flow and semi-annular flow. Flow pattern development length due to hydrodynamic effects and unavoidable numerical transients is discussed in details.

## INFLUENCE OF CONVECTION IN TUBE BUNDLES DURING BOILING

Andrea Luke, University of Kassel, Germany; Joseph Addy, University of Kassel, Germany; Bjoern Mueller, University of Kassel, Germany

Heat transfer in boiling is used in flooded evaporators for instance in different process steps during the liquefaction of natural gas. To understand the fundamentals of nucleate boiling and to get a better understanding of the boiling characteristics it is necessary to identify the influencing parameters. The operation parameters are saturation pressure and heat flux. The microstructure of evaporator tubes supports the bubble formation by providing stable nucleation sites, which are cavities that hold the necessary amount of vapour. The macrostructure enhance heat transfer in boiling by nucleation and by additional convective effects. The upstreaming bubbles increase the heat transfer on the tubes above. These parameters are all a function of the thermo-physical properties of the boiling liquid.

The flooded shell and tube evaporators are designed by pool boiling data for single tubes, which is inaccurate due to the convective effects from rising vapor bubbles in tube bundles. Therefore, the bubble flow and the additional convection evoked by them is analyzed in video evaluation of boiling hydrocarbons for different pressures. It is shown, that the convective effects by the bubble flow are important for low to moderate heat fluxes. Furthermore, a short overview on different calculation methods for the integral heat transfer coefficient in horizontal tube bundles and convective effects in boiling is given and evaluated by new experimental data.

## SYNTHESIS OF CRUD AND ITS EFFECTS ON POOL AND SUBCOOLED FLOW BOILING

Carolyn Coyle, MIT, USA; Bren Phillips, MIT, USA; Sean Lowder, MIT, USA;

Jacopo Buongiorno, Massachusetts Institute of Technology, USA; Thomas McKrell, Massachusetts Institute of Technology, USA; Michael Rubner, MIT, USA  
 Previous studies have demonstrated the potential of porous, hydrophilic surfaces to lead to more efficient boiling. CRUD (Chalk River Unidentified Deposits) is a naturally occurring porous, hydrophilic layer with characteristic boiling chimneys that forms on fuel rods during nuclear reactor operation. Unique features of these deposits are the characteristic boiling chimneys that form as vapor leaves the surface. It has been hypothesized that the presence of these chimneys, by providing a clear path for vapor escape, can further enhance boiling properties such as critical heat flux (CHF) and heat transfer coefficient (HTC). An investigation of such effects has been conducted by preparing a porous, hydrophilic layer with boiling chimneys on indium tin oxide coated sapphire heaters. These heaters were tested in pool and flow boiling facilities in MIT's Reactor Hydraulics Laboratory. A porous matrix emulating CRUD was created using layer-by-layer deposition of 100 nm silica nanoparticles to form porous, hydrophilic thick films. Photolithography was used to manufacture posts that were then dissolved to create characteristic boiling chimneys. Features such as thickness, wettability, pore size, and chimney diameter and pitch were verified. During pool and flow boiling testing, IR thermography and high-speed video were used to obtain two dimensional temperature profiles of the active heater area to quantify properties such as HTC, nucleation site density, bubble departure frequency, and bubble departure diameter. Initial data has shown that HTC increases with increasing layer thickness and chimney diameter while the chimney pitch has relatively no effect.

\* Support from the Consortium for Advanced Simulation of LWRs is gratefully

## TRANSIENT MASS TRANSFER FROM A SPHERICAL DROPLET EVAPORATING IN GASEOUS ENVIRONMENT

Simona Tonini, *Università degli studi di Bergamo, Italy*; Gianpietro Elvio Cossali, *Università degli studi di Bergamo, Italy*

Analytic modelling of drop evaporation is usually approached under quasi-steady approximation and with fixed boundaries, disregarding the inherent unsteadiness of such phenomenon and the fact that drop radius shrinking due to evaporation settles a moving boundary problem. Such approximations yield simple and very useful analytical solutions of the species conservation equations. However it is known that after the sudden immersion of a drop in a gaseous environment a relaxation time is needed to reach quasi-steadiness, and the evaporation rate during this period is expected to be much higher than that under steady conditions. The present work is aimed to define the analytical problem of evaporation in a gaseous environment relaxing the above mentioned approximation. The spherically symmetric species conservation equation for vapour transport in a steady gas environment is developed in non-dimensional form accounting for moving boundaries. Numerical solution allows to evaluate the relaxation time as a function of the Spalding mass number and to quantify the evaporated mass during this time lapse. Algebraic correlations are proposed to estimate these effects providing simple implementation of correction factors into the available models.

## EFFECT OF TRIPLE LINE ON SPHERICAL-SUBSTRATE-MEDIATED CONDENSATION

Sanat Kumar Singha, *Indian Institute of Technology Kharagpur, India*; Prasanta Kumar Das, *Indian Institute of Technology Kharagpur, India*; Biswajit Maiti, *Indian Institute of Technology Kharagpur, India*

A novel physical model has been developed for spherical-substrate-mediated condensation considering the effect of both the routes of monomeric addition combined with the peripheral tension. In the kinetic study of the seed-aided condensation, two mechanisms of monomeric addition are employed, i.e. direct interfacial accumulation in the neighborhood of the droplet-vapor interface and peripheral addition through the vicinity of the triple line. A rigorous energetic derivation has been formulated based on the principle of the free energy extremization. Moreover, a kinetic model associated with the detailed balance analysis has also been considered to determine the several important kinetic parameters, viz. the rate of monomeric addition and deduction, the rate of cluster growth etc. The energy barrier for heterogeneous condensation can be divided into the geometric contribution and the component related to the peripheral tension. Considering the growth rate, it is evident that condensation promotion becomes more effective for larger hydrophilic convex substrate and smaller hydrophobic concave substrate. However, higher apparent substrate wettability restrains condensation in hydrophilic regime for convex spherical substrate and in hydrophobic regime for concave spherical substrate.

## BECKER-DÖRING MODEL FOR NON-EQUILIBRIUM STEAM CONDENSATION IN A SUPERSONIC NOZZLE

Rob Hagmeijer, *University Twente, The Netherlands*; Edwin Van Der Weide, *University of Twente, The Netherlands*; Vitaly Kalikmanov, *Twister b.V., The Netherlands*

Condensation in steam turbines degrades the efficiency of power generation. Usually, condensation is described by means of the method of moments (MOM) which is based on classical nucleation theory (CNT) assuming quasi-steady state nucleation. This assumption does however not hold in rapidly expanding nozzle flows and the nucleation rate may be off by several orders of magnitude. In addition, moment methods only approximately describe the statistics of droplet size distributions, e.g. the number of droplets and the averaged mass of droplets. We have used a much more detailed model for condensation consisting of the so-called Becker-Döring (BD) equations, i.e., balance equations for each individual droplet-size that do not require the CNT assumption. The largest droplets typically consist of ten million molecules leading to ten million balance equations. To reduce the system one can safely describe the larger droplets in groups and still describe the smaller sizes individually leading to one or two thousand equations. We have solved the inviscid flow equations coupled with the Becker-Döring equations in two-dimensions by means of a standard second order finite volume discretisation scheme with a limiter and MUSCL reconstruction. The source term is advanced in time by a point implicit method. As a result we obtain highly detailed droplet size distribution dynamics which are strongly coupled to the fluid dynamics through the release of latent heat during condensation. Several examples are shown and the sensitivity to surface tension modeling is examined.

## SUPPRESSION MEASURES OF STEAM EXPLOSIONS WITH AND WITHOUT EXTERNAL PRESSURE PULSE

Masahiro Furuya, *Central Research Institute of Electric Power Industry, Japan*; Takahiro Arai, *Central Research Institute of Electric Power Industry, Japan*

Steam explosion has been causing industrial disasters in metalwork and other industries. We had reported that Polyethylene glycol (PEG) is a reliable retardant for spontaneous steam explosions without external triggers. In order to investigate controllability of steam explosion with PEG, experiments were conducted with and without a pressure pulse generator. Steam explosion was suppressed with a 0.03 wt% PEG solution for molecular weight of 4 million both with and without the pressure pulse generator. This is because the cloudy-point phenomenon stabilizes vapor film and prevents the solution from mixing finely by the precipitated solute near the steam-water interface. The molecular weight must be selected in reference to the cloudy-point temperature to be lower than saturation temperature by a certain degrees at the target pressure. The effective concentration became denser when large share stress and/or external force act on the vapor film.

## EFFECT OF THE RECESS GEOMETRY ON CRYOGENIC CAVITATING FLOW THROUGH THE FLOATING-RING CLEARANCE

Mamoru Oike, *Ishinomaki Senshu University, Japan*; Masataka Kikuchi, *Japan Aerospace Exploration Agency, Japan*; Hitoshi Takada, *Japan Aerospace Exploration Agency, Japan*; Takayuki Sudo, *Japan Aerospace Exploration Agency, Japan*; Tomoyuki Takano, *Japan Aerospace Technology, Japan*

In the present study, a flow visualization experiment in the clearance between a clear floating-ring and a rotating journal was carried out in order to clarify the effect of the recess configuration on the flow characteristics of the cryogenic hybrid journal bearing for a rocket turbopump. Four types of the floating-rings with different recess geometries, i.e., the circular Young Leaf Marks, the plane circle and non-recess, were operated at a rotational speed of up to 45,000 rpm using liquid nitrogen as the working fluid to observe the interaction between a source flow from the recess and a rotating flow induced by the journal rotation in the clearance. The cavitation cloud inside the clearance was induced by viscous frictional heating and the pressure drop. The influence of the geometry of the recess leading edge on the flow characteristics was investigated by comparing flow coefficient and the cavitation cloud area ratio obtained from the visual image. Based on the experimental results, the dependence of the flow characteristics on the rotational speed was confirmed to be affected by the sweepback angle of the recess leading edge. Furthermore, It can be considered that the value and the gradient of the pressure distribution within the clearance of the floating-ring tend to decrease with increasing sweepback angle at the same cavitation number

\* Support from the Japan Society for the Promotion of Science under Grant-in-Aid for Scientific Research (c) 25420128 is gratefully acknowledged

## INTERFACE CAPTURING SIMULATIONS OF SHOCK-ACCELERATED BUBBLE PAIRS

Ratnesh Shukla, *Indian Institute of Science, India*

The collapse dynamics of a pair of shock-accelerated air bubbles in a water-like medium is investigated numerically with an interface capturing finite-volume multiphase compressible flow solver. The numerical method employs coupled color function and level set descriptions of the interface to suppress spurious numerical oscillations at the air-water interface and to counter the excessive numerical diffusion caused by the standard shock- and interface-capturing treatment. This combined approach leads to a more accurate modeling of the wave interface interactions and robust handling of topological changes including merger and breakup of the interface even in the presence of large density and pressure variations and strong shocks. Simulations reveal that an upstream bubble located initially in close proximity to the one downstream effectively shields it from the incident shock wave thus delaying its collapse significantly. However, the blast wave emitted from the collapse of the upstream bubble triggers and accelerates the collapse of the one downstream thus substantially increasing its collapse intensity, in agreement with previous experimental work on shock-induced collapse of void arrays. A range of complex jetting patterns that result from the large interface deformations and bubble splitting including formation of a reverse jet are found. The dependence of the jetting patterns, the maximum pressure and the collapse times on the inter-bubble spacing is established.

### NUMERICAL STUDY OF WATER DROPS BEHAVIOUR IN OIL UNDER ELECTRIC FIELD

Bin Chen, Xi'an Jiaotong University, China; Meng Huang, Xi'an Jiaotong University, China

In order to reveal the mechanism of electric demulsification, the behaviors of aqueous drops among continuous oil in electrostatic field are studied by numerical simulation. For the electric-driven multiphase flow, Lorentz force is employed as the electric driving force to get charge distribution. Coupled with surface tension, two-phase flow in electric field was solved by a projection Method on unstructured grid with a PLIC-VOF method for the water-oil interface tracking. After the validation by visualization experiment, the behaviors of single water drop and double drops in oil are simulated under electric field. An exponential relationship between deformation rate of the water drop and the Electric Weber Number has been found before the breakup of droplet under critical electric intensity. Water drops positioned along direction of electric field will coalesce more easily. Breakup occurs at the end of major axis of an ellipsoidal drop in form of releasing secondary drops rather than being divided from the middle, because of the stress concentration produced by the high charge density at the drop end. Alternating electric field in a regular hexagon domain with three pairs of electrodes shows a remarkable advantage for demulsification.

\* Support from NSFC grant 51176152 is gratefully acknowledged.

### EXPERIMENTAL OBSERVATION OF THREE-DIMENSIONAL PARTICLE CLUSTERING IN TURBULENT CHANNEL FLOW

Andras Nemes, University of Minnesota, USA; Filippo Coletti, University of Minnesota, USA; Kee Onn Fong, University of Minnesota, USA; Nicholas Sloan, University of Minnesota, USA

Although it is well known that inertial particles preferentially concentrate in specific regions of a turbulent velocity field, experimental observations of the three-dimensional nature of this phenomenon are lacking. Here we consider a vertically oriented, fully developed, turbulent channel flow, in which the working fluid is air laden with size-selected glass particles. The volume and mass loading are kept low and the particle diameter is smaller than the viscous length scale, so that the turbulence is unaffected by the dispersed phase. Tomographic Particle Image Velocimetry is used to reconstruct the position and velocity of the inertial particles. Specifically, the tendency of the particles to concentrate in long streamwise streaks and to drift towards the wall (turbophoresis) are quantitatively characterized by the instantaneous and mean concentration fields. The findings are discussed in relation to the results of previous studies which have used one-way coupled direct numerical simulations, and on which the current understanding of these class of flows is based. We then increase the volume fraction and enter the two-way coupled regime. We show how the change in turbulence structure, caused by the particle mass loading, changes the three-dimensional spatial distribution of the particle themselves.

### 3D NUMERICAL SIMULATIONS OF MACROSCOPIC FLOWS WITH MOVING CONTACT LINES

Zlatko Solomenko, Laboratoire de Mécanique des Fluides et d'Acoustique, France; Peter Spelt, Laboratoire de Mécanique des Fluides et d'Acoustique, France; Michel Lance, LMFA, France; Pascal Alix, IFP Energies nouvelles, France; Christine Dalmazzone, IFP Energies nouvelles, France; Yacine Haroun, IFP Energies nouvelles, France

Flows with moving contact lines involve a large range of length scales. To alleviate a non-integrable singularity in the wall shear stress that otherwise arises at the contact line, a slip condition is often used, wherein the slip length is typically nanometrical. A priori, the flow in this immediate vicinity of a contact line affects the large-scale flow, for instance, it strongly affects the velocity of the contact line. In practice, it is not feasible to resolve the flow down to such length scales. In this work, a 3D macroscale model is developed to avoid having to resolve the flow down to the slip length. A well-tested level-set method (the freeware TPLS) is extended for this purpose. The computational strategy is based on resolving the flow down to an intermediate length scale, and to use asymptotic theory for the unresolved part of the flow in the immediate vicinity of the contact line. This work generalizes prior work in this area that is restricted to axisymmetric problems using level-set. The 3D code is validated against Direct Numerical Simulations (wherein the flow is resolved down to the slip length) and experimental results of axisymmetric droplet spreading in viscous and inertial regimes. Truly three-dimensional flows have also been simulated; the code is tested against experimental results for sliding droplets, wherein contact-angle hysteresis is accounted for. Simulations of falling film flows are performed and tested against previous numerical and experimental results of fingering instabilities.

### NUMERICAL SIMULATIONS ON COPPER DROPLET COLLISIONS IN THE SLAG CLEANING PROCESS

Hongkun Yang, Forschungszentrum Jülich, Germany; Jorg Wolters, Forschungszentrum Jülich, Germany; Philipp Pischke, RWTH Aachen, Germany; Helmut Soltner, Forschungszentrum Jülich, Germany; Jochen Frohlich, Technische Universität Dresden, Germany; Sven Eckert, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany

The copper slag cleaning process contains complex physicochemical phenomena, among which the collisions between liquid metal droplets possess an enormous influence on the cleaning efficiency. An Euler-Lagrange approach was chosen to numerically study the slag-copper droplet system with the CFD code FLUENT. For the dispersed phase a new hybrid collision algorithm was implemented in FLUENT to overcome the mesh-dependency problem of the most commonly used stochastic algorithm. It provides better predictions of the collision probability among the inhomogeneously distributed droplets. The outcome of collisions in the numerical simulations was firstly determined by currently available regime maps for liquid-gas system. But due to the high viscosity and density of both phases, the slag-droplet collision system differs from the conventional collision systems. Therefore the VOF (volume of fluid) method is used to study the binary collision process between copper droplets in the slag and to verify the new regime maps. The impact of employing the new regime maps on the modeling the slag cleaning process will be discussed.

\* Support from the German Helmholtz Association in the framework of the Helmholtz Alliance LIMTECH is gratefully acknowledged.

### DETERMINISTIC AGGLOMERATION MODELS FOR THE EULER-LAGRANGE APPROACH USING LARGE-EDDY SIMULATION

Naser Almohammed, Helmut-Schmidt University Hamburg, Germany; Michael Breuer, Helmut-Schmidt University Hamburg, Germany

In the present work, the particle agglomeration in turbulent flows is investigated based on an Euler-Lagrange approach relying on the large-eddy simulation technique. In the framework of a hard-sphere model with deterministic collision detection two different models for the agglomeration process are employed, an energy-based (EAM) and a momentum-based agglomeration model (MAM) for rigid, dry and electrostatically neutral particles. The comparative study is based on particle-laden plane channel flows and turbulent shear flows. The investigations show that both agglomeration models predict similar trends of the physical behavior of the agglomeration process, but their results deviate slightly. The most important reasons for the differences observed between the results of both models are discussed showing that the deviations can be mainly attributed to the different formulations of their agglomeration conditions. Furthermore, the influence of various simulation parameters (i.e., normal restitution coefficient, diameter of the particles, wall roughness) and simulation setups (i.e., feedback of the particles on the fluid, subgrid-scale model for the particles, inclusion of the lift force) is analyzed. Finally, the advantages and drawbacks of both agglomeration models are highlighted.

### DETAILED MODELING OF SLOSHING IN SATELLITES TANK FOR LOW BOND NUMBERS

Mathieu Lepilliez, Institut de Mécanique des Fluides de Toulouse, France; Sebastien Tanguy, INP Toulouse, France

Consumption of ergols is a critical issue regarding the whole lifetime of a satellite. During maneuvers in mission phases, the Helium bubble used to pressurize the tank can move freely inside, thus generating movement of the center of mass, and sloshing which can disrupt the control of the satellite. As accelerations are small compared to capillary forces, theoretical work is not recommended, and there are very few experiments available for these setups (long damping periods in microgravity).

In this study we present numerical results obtained from CFD computation, using an Immersed Interface Method to model the tank with a level-set approach for both liquid-gas interface and solid-fluid interface. A parametric study is proposed to observe the influence of the Bond number on resulting forces and torques generated on the tank. One can observe different steps during the maneuvers under microgravity: the first part is dominated by accelerations and volume forces, which flatten the bubble on the hydrophilic tank wall. When the forcing stops, the bubble bounces back, generating sloshing by moving under the influence of inertia and capillary effects. Finally viscous effects damp the sloshing by dissipating the kinetic energy of the bubble. Those results are compared to actual in-flight data for different typical maneuvers on forces and torques, allowing us to characterize the period and damping of the sloshing. Support from grant by Airbus Defence & Space and CNES (French Space agency) is gratefully acknowledged.



## LBE-DEM SIMULATION OF GAS-SOLID TWO-PHASE IMPINGING STREAM

Wenkai Xu, China University of Petroleum-Beijing, China; Nan Gui, Institute of Nuclear and New Energy Technology, China; Zhenlin Li, China

The gas-solid two-phase impinging stream is investigated using the Euler-Lagrangian framework in this study. The lattice Boltzmann equation (LBE) method is applied to simulate gas phase, and the discrete element method (DEM) is utilized to track particle motion and collision. Three Reynolds numbers of  $Re=250$ ,  $500$  and  $750$ , four dimensionless nozzle-to-nozzle distance of  $Dn=1$ ,  $2$ ,  $3$  and  $4$ , and three Stokes number of  $St=1$ ,  $10$ ,  $100$  are considered. The dynamic and thermal characteristics of the gas and solid phases are investigated, especially for the interactions, both momentum and heat, between gas and particles. Moreover, the influences of Reynolds number, ratio of distance between nozzles on both velocity and temperature field are explored. The effects of the Stokes number on particle motion and heat transfer are discussed. The results showed that higher Reynolds number, smaller nozzle-to-nozzle spacing, and smaller Stokes number improve the performance of mixing and heat transfer properties between gas and particle.

\* The support from the national natural science of foundation (Grant No. 51576211 & 51106180) is gratefully acknowledged.

## FLAPPING OF AN AXISYMMETRIC LIQUID JET

Antoine Delon, Université Grenoble Alpes, France; Jean-Philippe Matas, Université Claude Bernard Lyon and CNRS, France; Alain Cartellier, Université Grenoble Alpes and CNRS, France

Air blast atomization of liquid jets is commonly used in propulsion systems, but the mechanisms governing the drop formation are still not fully understood. We therefore undertook experiments on the atomization of a water jet by a co-current air stream, focusing on the large scales instabilities. Experiments were performed over a wide range of liquid ( $0.1$ - $0.5$  m/s) and gas ( $10$ - $150$  m/s) velocities and for various liquid jet diameters ( $5$  to  $20$  mm) and gas annulus thicknesses ( $2$  to  $24$  mm). A number of new features were unveiled concerning the global flow structure. In particular, the intact length happens to be controlled by the lateral displacement associated with the large-scale instability and not by stripping as proposed in the literature. The origin of these large-scale instabilities and their connection with Kelvin-Helmholtz (KH) instability will be also discussed and compared with available investigations on liquid sheets. Finally, we will show how the selection of the flapping mode (i.e. sinuous or varicose) and of its frequency can be controlled in a certain range, and how the mean drop characteristics and their spatial distribution change according to the mode selected.

## DEVELOPMENT OF AN EXPERIMENTAL APPARATUS AND MEASUREMENT TECHNIQUES FOR THE STUDY OF QUASI-REAL SLUG-FLOW IN VERTICAL DUCTS

Rafael Cerqueira, SINMEC / Federal University of Santa Catarina, Brazil; Emilio Paladino, SINMEC / Federal University of Santa Catarina, Brazil; Ronei Brito, SINMEC / Federal University of Santa Catarina, Brazil; Clovis Maliska, SINMEC / Federal University of Santa Catarina, Brazil

This paper presents the description of the experimental apparatus and instrumentation developed for the study of quasi-real slug flow using Particle Image Velocimetry and Laser Diode Photocell techniques. Most experimental studies of the flow fields around Taylor bubbles, aiming a deeper insight into the slug flow, analyze the situation of isolated Taylor bubbles rising in stagnant or co-current liquid flow. However, in most industrial situations slug flows are characterized by the presence of small dispersed bubbles in the liquid stream flowing together with large Taylor bubbles. On the other hand, the chaotic nature of a real slug flow would not allow an adequate characterization of the flow around single Taylor bubbles, mainly, due to the lack of the repeatability of the flow field measurement, making difficult the calculation of the averaged fields. Thus, in this work, a specific experimental apparatus and instrumentation was developed, which allows the study of the flow around Taylor bubbles with and without dispersed small bubbles in the liquid stream under controlled conditions. This allows the PIV measurements be done under repeatable conditions and thus, the determination of the averaged flow fields. The laser diode photocell technique is used to synchronize the PIV system with the passage of Taylor bubbles at the test section as well as to measure the terminal velocity and length of Taylor bubbles, under different flow conditions.

## NUMERICAL INVESTIGATION OF DUAL BUBBLE SYSTEM

Mustapha Al-Behadili, University of Nottingham, UK; David Hargreaves, Faculty of Engineering, University of Nottingham, UK; Buddhikha Hewakandamby, University of Nottingham, UK; Barry James Azzopardi, University of Nottingham, UK

This paper describes the numerical modelling of air-water dual-bubble systems in which two initially spherical bubbles are allowed to rise, move around each other and possibly coalesce. The numerical model uses Computational Fluid Dynamics (CFD) and will eventually be validated against experiments, which are progressing in parallel.

Here, we describe the techniques used to track the bubbles. The ANSYS-Fluent software is used with a standard multiphase model called the Volume of Fluid (VOF) method. In order to achieve a desirable level of refinement of the mesh around the bubbles, the standard refinement/coarsening algorithm in Fluent is used. However, to mark the region around the bubbles that needs to be refined, a scalar Lagrangian equation is solved where a source of the scalar is applied inside the bubbles and at the walls of the domain, a Dirichlet condition is used where the scalar is fixed at zero. The mesh is then refined in the region where the scalar is greater than one quarter of its maximum value. As the bubbles rise, the scalar equation is repeated solved and so the refinement region is automatically modified around the bubbles with little appreciable increase in the number of cells in the mesh.

The goal of the research is to identify the effect that the initial separation of the bubbles (both in the horizontal and vertical planes) has on the rise of the bubbles and also on the likelihood of coalescence.

## DROPLET SIZE MEASUREMENT IN STATIC MIXERS USING REFRACTIVE INDEX MATCHING AND LASER-INDUCED FLUORESCENCE

Philipp Rudolf von Rohr, ETH Zurich, Switzerland; Richard Hafeli, ETH Zürich, Switzerland; Oliver Ruegg, ETH Zürich, Switzerland; Marco Altheimer, ETH Zurich, Switzerland

A liquid-liquid two-phase flow through a foam-like porous structure and a Sulzer SMX static mixing element is studied and compared. Optical measurements are performed in a system of two immiscible liquids, which are index matched to the solid material enabling optical access. The disperse phase is seeded with a fluorescent dye to measure the droplet size and position within the internal structures using laser-induced fluorescence. It is observed, that at low flow rates, droplets follow preferred paths, whereas at higher flow rates, they are more homogeneously distributed within the structures. The droplet size distribution is found to be well represented by the Sauter mean diameter. A correlation is proposed correlating the droplet size as a function of Weber and Reynolds number, being in good agreement with data found in literature. Measuring along the axis of the two static mixers, the foam-like porous structure is found to better disintegrate the disperse phase and to produce smaller droplets than the Sulzer SMX. From this observation, it is concluded that the change in the free cross section is an important parameter, as both geometries have the same porosity and hydraulic diameter but very different free cross sections along the axis.

\* We gratefully acknowledge the financial support from the Swiss National Science Foundation (SNF) under grant number 200021 132552.

## EXPERIMENTAL CHARACTERIZATION OF INTERFACIAL WAVE STRUCTURE OF A FALLING LIQUID FILM IN A VERTICAL LARGE PIPE DIAMETER

Abbas Hasan, University of Nottingham, UK; Barry James Azzopardi, University of Nottingham, UK; Buddhikha Hewakandamby, University of Nottingham, UK; Yousuf Alaoui, University of Nottingham, UK; Nicholas Watson, University of Nottingham, UK; Shara Mohammed, University of Nottingham, UK

There is considerable literature on the thicknesses of falling liquid films, much of which is focussed on either flat plates or small diameter pipes. Many studies provide time series data of the film thickness at one or two points on the pipe wall. This paper reports on an investigation of the interfacial structure of falling liquid films (liquid Reynolds numbers,  $Re_L=618$ - $1670$ ) in a large diameter pipe ( $127$  mm) using a Multiple Pin Film Sensor (MPFS) which is capable of providing measurements of film thickness and interfacial waves with excellent resolution in time and in the circumferential and axial directions. Parameters, such as film thicknesses, wave velocities and frequencies were extracted. 3D interfacial wave structures were reconstructed from the film thickness data. The waves seen were much localized unlike those in smaller diameter pipes which are characterised as coherent rings. The mean film thicknesses are generally in good agreement with published models. The mean film thickness obtained from MPFS was also compared with the output of two other sensors, Ultrasound and conductance ring pairs. There is good agreement between the three methods particularly when the fact that the ring pair technique provides a circumferentially averaged value.

#### VOID FRACTION CHARACTERIZATION THROUGH AN OPTICAL METHOD FOR AIR-WATER UPWARD FLOW ACROSS TUBE BUNDLE

Fabio Toshio Kanizawa, University of São Paulo, Brazil; Gherhardt Ribatski, University of São Paulo, Brazil

This paper presents a study focused on the experimental determination of superficial void fraction during air-water upward flow across triangular tube bundle. An optical method using a high-speed camera, optical settings and diluted fluorescent dye in the liquid phase was developed and used for void fractions measurement between two adjacent tubes in the center of the test section. A regular triangular tube bundle test section was used for the experimental campaign, counting with 19 mm OD tubes and transversal pitch of 24 mm. The results obtained with the optical system was previously compared with gravimetric method, and reasonable agreement is observed. Experiments were performed during bubbles flow, comprising superficial liquid and gas velocities up to 0.6 and 0.14 m/s, respectively. Superficial void fraction increases with mass velocity and gas mass fraction. The effects of using Rhodamine B, as fluorescent dye, on the flow pattern characteristics was carefully examined.

\* Support from São Paulo Research Foundation grant number 2014/06902-9 is gratefully acknowledged.

#### TOWARDS THE NUMERICAL SIMULATION OF TWO-PHASE FLOW AND INTERFACIAL SPECIES TRANSFER IN STRUCTURED PACKINGS

Simon Hill, Linde AG, Germany; Thomas Acher, Linde AG, Germany; Daniel Deising, Technische Universität Darmstadt, Germany; Rainer Hoffmann, Linde AG, Germany; Andreas Rarrek, Technische Universität München, Germany; Holger Marschall, Technische Universität Darmstadt, Germany; Dieter Bothe, Technische Universität Darmstadt, Germany; Harald Klein, Technische Universität München, Germany

To optimize the separation performance of structured packings a thorough understanding of the hydrodynamics and transport processes inside these column internals is vital. This study represents an approach to the numerical simulation of separation processes using the Volume-Of-Fluid (VOF) method for two-phase hydrodynamics and the Continuous-Species-Transfer (CST) method to consistently incorporate interfacial species transfer.

The first part of this study shows the results gained by investigating the hydrodynamics only. Thereby the gaseous flow is analyzed, focusing on the visualization and understanding of the transient flow structures occurring at higher flow rates. Also, a two-phase flow simulation is conducted to identify the distribution of the liquid phase. The results of both, single-phase as well as two-phase flow simulations, are validated against experimental results.

For interfacial heat and species transfer the CST method by Marschall et al. (2012) and Deising et al. (2015) is enhanced to cope with mesh-induced discretization errors. Therefore semi-implicit correction schemes are developed and validated using meshes with well-defined distortion. Furthermore, suitable boundary conditions for the transported species are established.

All code extensions are implemented in the CFD software package OpenFOAM.

#### A NUMERICAL AND ANALYTICAL STUDY ON THE MICRO-JET INDUCED BY A SHOCK

Xiao Bai, Beijing Computational Science Research Center, China; Xiaolong Deng, Beijing Computational Science Research Center, China

A micro-jet can be induced by a shock passing by a cambered or spherical cap liquid-air interface inside a capillary. Peters et al. 2013 gave a model of maximum jet velocity with inviscid, irrotational, and incompressible hypothesis, and applied it to fit the experimental results by Tagawa et al. 2012, by adjusting four parameters. Lack of definite relationships of these parameters makes it hard to be applied in practice. In our work, starting from Peters' model, a more definite relationship between maximum velocity and Mach number is obtained from theoretical consideration, with compressible hypothesis. The effects of some parameters are investigated numerically with the cut cell based sharp-interface method developed by Chang, Deng & Theofanous 2013, which give some relationships of the undetermined parameters within some domains. This new model can give the maximum jet velocity when the shock Mach number and contact angle are given. Simulation results also show that when the shock wave is strong enough, Peters' model is not valid anymore, no matter how to adjust the parameters.

\* Support from Natural Science Foundation of China (No. 11202020 and 91230203) and President Foundation of Chinese Academy of Engineering Physics (No. 201501043) are gratefully acknowledged.

#### COMPUTATIONALLY-EFFICIENT SIMULATION OF INDUSTRIAL BUBBLE COLUMN BIOREACTORS: INVESTIGATION OF SCALE AND MIXING EFFECTS ON YIELD

David Fletcher, University of Sydney, Australia; Dale McClure, University of Sydney, Australia; John Kavagnah, University of Sydney, Australia; Geoff Barton, University of Sydney, Australia

Over the last four years we have undertaken a collaborative research project aimed at resolving long-standing industrial challenges with bubble column bioreactors. During this time we have developed a computationally-efficient CFD model using ANSYS CFD software and performed validation at benchtop, pilot and industrial scales. This validation includes comparison of experimentally determined holdup, oxygen transfer and mixing times with results from transient, 3D Euler-Euler multiphase flow simulations. This paper will provide a description of the equations, closure relations, the modelling approach and the numerical challenges. Typical model validation results will then be presented and discussed. We will introduce a variety of different techniques to illustrate how the impact of mixing and dosing of feed can influence the growth of micro-organisms – the real area of interest of industries using this type of reactor. Finally, we will highlight the areas where input from the broader multiphase community would assist in the further development of such models.

#### MEASURING LIQUID FLOW IN STRUCTURED PACKINGS USING GAMMA AND X-RAY TOMOGRAPHY

Rainer Hoffmann, Linde AG, Germany

Detailed knowledge of the liquid distribution in structured packings is an essential requirement for developing high-performance columns. Two different tomographic technologies are used to resolve the liquid distribution on different length scales. To simulate the behavior of cryogenic liquids model fluids like silicone oil or methyl-pentane are used.

For macroscopic measurements of the whole column cross-section (up to Ø 750 mm) a gamma tomograph has been built. The tomograph is qualified to operate in ATEX areas and can resolve liquid flow phenomena like wall flow, inlet flow distribution, crossover between packing blocks etc.

For microscopic measurements of flow details x-ray tomography is used. Using a voxel size of 22 µm the influence of the packing's geometry parameters on the liquid flow can be analyzed. The resulting data is also used for verification of corresponding two-phase flow CFD simulations.

#### NUMERICAL SIMULATION OF AN AIR JET IN CROSS WATER FLOW WITH APPLICATION TO AIR-LAYER DRAG REDUCTION

Hojun Moon, Pohang University of Science and Technology (POSTECH), Republic of Korea; Donghyun You, Pohang University of Science and Technology (POSTECH), Republic of Korea

Recently, lubrication systems that inject air through an upstream hole or a slit to form an air layer in the downstream are considered for reduction of skin-friction drag over a ship or an underwater vehicle. Through experimental parametric studies, it has been reported that two different patterns of downstream air-covered layers namely, the delta shape and the lambda shape are formed depending on the amount of air injected through an upstream hole. In spite of the experimental observation, the underlying mechanism for determining the shape of the air layer and the effect of the shape of the air layer on the skin-friction drag are not well understood. In the present study, a mass-conserving volume-of-fluid method is coupled with an incompressible flow solver to accurately predict the interaction between an air jet and cross water flow. Both the Reynolds-averaged Navier-Stokes (RANS) and the large-eddy simulation (LES) methods are considered to account for turbulence effects. Simulation results predicted by using RANS and LES are compared with experimental data in terms of the angle and shape of the formed air layer. The present study reveals how the air-layer shape is correlated with the amount of the injected air and the relative speed of the injected air to the cross water flow.

\* Supported by National Research Foundation of Korea Grant No. NRF-2014R1A2A1A11049599.

### NUMERICAL SIMULATION OF A HIGHLY-VISCOUS MICROJET GENERATED BY AN IMPULSIVE FORCE

*Hajime Onuki, Tokyo University of Agriculture and Technology, Japan; Oi Yuto, Tokyo University of Agriculture and Technology, Japan; Yoshiyuki Tagawa, Tokyo University of Agriculture and Technology, Japan*

In this study, we numerically investigate the pressure field in a highly-viscous microjet generator. A generation system for a viscous microjet is desired in various devices, such as ink-jet printers and needle-free injection systems. Recently we have proposed a new system for generating an extremely highly-viscous microjet (up to 10,000 cSt) and presented a physical model using pressure impulse approach. We have measured the jet velocity and compared experimental results with the physical model. It has been found that the velocity in experiments and the model agree. However, the pressure field in the liquid has not been clarified yet, although it is crucial for elucidating the mechanism of the generation of viscous microjets. In this study, we reproduce the microjet using Level-Set method and simulate the pressure field under condition of various viscosities and liquid levels. In the system for generating viscous jets, a wettable-microtube is partially submerged into a liquid in a container. The liquid level inside the tube is deeper than that outside of the tube. The impulsive force is then applied to the bottom of the container, followed by the microjet generation inside the tube.

\* Support from JSPS KAKENHI Grant Number 26709007 is gratefully acknowledged.

### THREE DIMENSIONAL MULTI-PHASE EULERIAN MODEL FOR NON-UNIFORM SEDIMENT-LADEN FLOW

*Kazuyuki Ota, Central Research Institute of Electric Power Industry, Japan; Takahiro Sato, Central Research Institute of Electric Power Industry, Japan; Hajime Nakagawa, Disaster Prevention Research Institute, Kyoto University, Japan*

This study presents a three dimensional multi-phase Eulerian model for non-uniform sediment-laden flow. The model uses momentum equations and continuity equations for water and several particle size classes. As regards the particle phase, interphase forces are considered with both the water phase and the remaining particle phases by using the drag coefficient. The inter-particle collision among the same particle sizes is considered as the additional viscous effect of the particle phase. The Volume of Fluid method is used to capture the air-water interface. The  $k-\epsilon$  type closure with a buoyancy modification term is used to account for the turbulence as a mixture fluid. The model was first applied to several experiments of non-uniform suspended sediment flow. The model simulated the sediment concentration of each particle class fairly well. The model was subsequently applied to large-scale debris flow experiments. In each experiment, water-saturated sediment discharged from behind a gate, descended a 95-m steep flume, and formed a deposit on a nearly horizontal runoff surface. The model sufficiently reproduced the velocity and shape of the debris flow. This work showed the accuracy of the three dimensional multi-phase Eulerian model for wide range of non-uniform sediment-laden flow.

### NUMERICAL MODELLING OF MULTIPHASE FLOW PHENOMENA COUPLED WITH REACTION KINETICS IN CHEMICAL LOOPING COMBUSTION PROCESS OF A FUEL REACTOR

*Luming Chen, the University of Nottingham, Ningbo China, China; Xiaogang Yang, the University of Nottingham, Ningbo China, China; Xia Li, University of Nottingham, UK; Collin Snape, University of Nottingham, UK*

CFD numerical modelling of multiphase flows coupled with heterogeneous chemical reaction kinetics in chemical looping combustion (CLC) in a fuel reactor has been conducted. A correlation parameter used to relate the time-dependent fluidised bubbles to the local eddies was proposed by correlating the local velocity fluctuation with the pressure fluctuation. The relations between the concentration of gaseous reactants or products and local eddies were also sought. It was revealed from the simulations that the apparent surface area of fluidised bubbles occurred in the reactants in the fuel reactor is closely correlated with variation of the fuel concentration. The parameter correlating the apparent surface areas of fluidised bubbles due to fluidised bubble occurrence with the local fuel concentration may be used in the CLC as an indicator for monitoring the reduction rate as the local pressure and velocity distributions are strongly associated with the fluidised bubbles formed in the fuel reactor.

### ANALYSIS OF INTERMITTENT FLOW PATTERNS OF WATER-STEAM FLOW IN SOLAR THERMAL POWER PLANTS WITH DIRECT STEAM GENERATION

*Alexander Hoffmann, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Tobias Hirsch, German Aerospace Center (DLR), Germany; Eckhard Schleicher, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Robert Pitz-Paal, German Aerospace Center (DLR), Germany*

Solar thermal power plants that use parabolic troughs for direct steam generation depict a commercially available technology for a renewable electricity production. The once-through operation mode is the most cost-effective approach for this type of power plant and, consequently, is mainly considered in the current research. This operation mode preheats and evaporates water and superheats steam within one continuous pipe. Hence, all mass qualities are passed along the pipe and various flow patterns are present. It is presumed that intermittent flow pattern influence the position of the mixture-vapor transition point at the end of evaporation and lead to undesired thermal stresses in the receiver pipe. To get an insight in the two-phase flow a numerical analysis is performed with ATHLET. This code provides a 1D mass, momentum and energy balance for both phases water and steam, respectively. The applicability of ATHLET on intermittent flows is discussed and confirmed. In the result an intermittent flow is numerically determined at specific void fraction ranges in the design of the DISS test facility at Plataforma Solar de Almería, Spain. The occurrence of these unsteady flow pattern diminishes at pressure  $P > 60$  bar. It is shown that the oscillating nature of the two-phase flow effects the position of the mixture-vapor transition point. For validation purposes an experiment with a wire-mesh sensor is designed. The obtained results contribute substantially to the understanding of two-phase flow behavior in direct steam generation processes.

### THE MODELING AND EXPERIMENTS OF AN ADVANCED OZONATION SYSTEM WITH RADIAL DIFFUSER

*Karol Swiderski, ETH Zurich, Switzerland; Bruno Heiniger, Degremont Technologies AG, Switzerland; Luca Ramoino, Degremont Technologies, Switzerland; Walter Uttinger, Degremont Technologies, Switzerland; Djamel Lakehal, ASCOMP AG, Switzerland; Chidambaram Narayanan, ASCOMP AG, Switzerland*

For many physical processes the representation of different phases simply by a volume fraction is not sufficient - especially problems involving phase change or mass transfer of dispersed phases. In the present work the Direct Quadrature Method of Moments (DQMOM) has been implemented into the commercial CFD code TransAT. DQMOM has recently become a very attractive approach for solving population balance equation thanks to its capability of representing the most interesting properties of the population, such as mean diameter, void fraction, number of particles, etc. TransAT is a finite volume software, with structured multiblock grids focused on multiphase flow modelling: including two-phase interface tracking, Lagrangian particle tracking and multiphase mixtures with an algebraic slip model that has been generalized for an arbitrary number of phases (N-phase ASM). DQMOM technique was coupled with the turbulent N-phase ASM model in order to extend the model to handle varied dispersed phase populations such that each class has its own velocity field. Air-water injection from a radial diffuser is studied here numerically and experimentally. The comparison between experiments and simulations is made for bubble size evolution and void fraction distribution in a vessel.

\* Support from project TRIUMPH (Treating Urban Micropollutants and Pharmaceuticals in Wastewaters) being a part of ACQUEAU – the Eureka Cluster for Water is gratefully acknowledged.

### TURBULENCE AND MOMENTUM INTERFACIAL TRANSFER IN AN AXISYMMETRIC BUBBLY JET FLOW

*Aroua Aouadi, National Engineering School of Tunis, Tunisia; Ghazi Bellakhal, National Engineering School of Tunis, Tunisia; Jamel Chahed, National Engineering School of Tunis, Tunisia*

This paper presents a numerical study of axisymmetric bubbly jet flow by using an eulerian-eulerian two-fluid model. The model presented in Chahed et al (2003), focuses on two aspects of the interaction between phases: first, the turbulent correlations associated to the added mass force are taken into account in the expression of the force exerted by the liquid on the bubbles. Second, the Reynolds stress tensor of the continuous phase is decomposed into a turbulent part and a pseudo-turbulent part; each part is predetermined by a transport equation. The model is applied to axisymmetric bubbly jet flow and the numerical results are confronted with the experiences of Sun et al (1985). One of the remarkable results of these experiments is to show that the bubbles are animated by a negative relative velocity near the central zones of jet. Consideration of the turbulent contributions of the added mass force in the momentum transfer allows to explain and to reproduce the negative relative velocity observed in the high sheared zone of the vertical jet. The analysis of the momentum interfacial transfer makes it possible to specify the role of the turbulent contributions and of the drift velocity on the vertical relative velocity.

## THE EFFECT OF THE CHANNEL INCLINATION ON FLOW PATTERN CHARACTERISTICS DURING GAS-LIQUID FLOW IN A SMALL RECTANGULAR CHANNEL

Francisco Loyola, University of Magallanes, Chile; Gherhardt Ribatski, University of São Paulo, Brazil

The present paper concerns an experimental investigation of the effect of channel inclination on the two-phase flow characteristics for air-water flow in a rectangular channel with cross-sectional area of  $6.0 \times 6.5 \text{ mm}^2$ . The experiments were run for inclinations channel from varying from  $+90^\circ$  to  $-90^\circ$ . The effects of turning the test section along its longitudinal axis on the flow patterns characteristics were also investigated. Experimental data were obtained for mass velocities from 90 to 760  $\text{kg/m}^2\text{s}$ , corresponding to gas and liquid superficial velocities from 0.03 to 19.4  $\text{m/s}$  and from 0.1 to 0.76  $\text{m/s}$ , respectively. Flow patterns maps were developed based on the following approaches: (i) analyses of two-phase flow images obtained from a high-speed video camera; and (ii) analyses through the k-means clustering algorithm of the pressure drop and optical signals concerning the two-phase flow. The bubbly, intermittent and annular flow patterns were characterized. From the analyses to the data, it was found that the flow pattern transitions are significantly affected by the channel inclination and rotation. The intensity of stratification effects are affected by rotating the channels along its longitudinal axis.

## VISUALIZATION AND HEAT TRANSFER OF HIGH PRESSURE CARBON DIOXIDE IN A SMALL HORIZONTAL TUBE

Haicai Lyu, Xi'an Jiaotong University, China; Qincheng Bi, Xi'an Jiaotong University, China; Hang Li, Xi'an Jiaotong University, China; Yajun Guo, Xi'an University of Architecture and Technology, China

Visualization and heat transfer experiment were performed to investigate heat transfer behaviors and two-phase flow pattern of carbon dioxide in a small horizontal tube. The experimental dataset was obtained in a horizontal smooth tube with 0.65m long and 2mm ID. The mass flux is in the range of 400 to 800  $\text{kg/m}^2\text{s}$ , and the pressure is 6MPa to 8.5MPa, which includes subcritical and supercritical pressure of  $\text{CO}_2$ . The forced convection and two-phase heat transfer coefficients were obtained to investigate the influences of the fluid bulk temperature, mass flux and pressure, respectively. The results indicated that the heat transfer coefficient was affected by the fluid buoyancy in the horizontal tube under the supercritical pressure. To get the flow pattern, a transparent quartz glass tube was selected. A high definition recorder of sony HXR-N70C was used to seize the flow image of high pressure carbon dioxide. Bubble, slug, annular flow was identified at different mass quality. By analysing the connection between flow pattern and mass quality, a flow regime map was drew. Based on the flow pattern, different correlations of heat transfer coefficient were compared to improve accuracy of the predicted correlation under two-phase flow region.

## ON THE CLASSIFICATION OF DROPLET BREAKUP IN AN EXTERIOR STREAM

Nicolas Rimbert, University of Lorraine, France; Miloud Hadj-Achour, University of Lorraine, France

Secondary fragmentation is the mechanism by which an isolated droplet fragments in a constant exterior fluid stream. Experiments (especially shock tube experiments) have led to a classification of the different fragmentation modes thanks to two non dimensional numbers, the Weber number (ratio of kinetic to surface energy) and the Ohnesorge number or Viscosity group. By comparing the characteristic deformation time of a droplet in an exterior stream to the characteristic time of Rayleigh-Taylor or Kelvin-Helmholtz instabilities as well as the characteristic deformation length and the corresponding instabilities wavelengths, we give a simple explanation to the observed classification. The droplet deformation model is a new semi-analytical model based on the assumption of ellipsoidal deformation of the droplet and axisymmetric potential flow outside the droplet. It is however slightly adapted in order to give the correct droplet oscillation frequencies for the smallest Weber numbers.

\*This work was done under the research program on nuclear safety and radioprotection (RSNR) and received funding from French government managed by the National Research Agency (ANR) under Future Investments Program (PIA), research grant n° ANR-10-RSNR-01. The authors would like to also acknowledge support from EU COST MP 1305 "Flowing Matter".

## EFFICIENT COMPRESSED AIR ENERGY STORAGE THROUGH SPRAY-COOLING

Chao Qin, University of Virginia, USA; Eric Loth, University of Virginia, USA

Compressed Air Energy Storage is particularly attractive for off-shore wind turbines if high efficiency can be achieved. Such efficiency is possible if the compression portion is nearly isothermal by employing water spray during compression to promote heat transfer. The present study examines this concept in an axisymmetric domain to investigate compression using a spray discharge within the cylinder at various mass loadings. The spray is based on a single pressure-swirl nozzle directed along the centerline and operating at the maximum liquid mass flux possible while retaining a mean droplet diameter of no more than 30 microns. The simulations uncovered flow characteristics such as vortex formation for the air-flow near the cylinder head and strong spatial variations in droplet size and concentration. Despite these effects, the overall two-dimensional efficiency was similar to that of one-dimensional predictions. The results also indicated that a single pressure-swirl nozzle injection resulted in an injected mass loading of 1.6 and yielded efficiency as high as 93% for a first-stage compression cycle. However, a second-stage compression cycle (with an intake pressure of 10 bar) using this same single nozzle resulted in reduced overall work efficiency indicating a multi-nozzle configuration should be considered for second-stage compression.

## FLOW BOILING HEAT TRANSFER AND TWO-PHASE FLOW CHARACTERISTICS IN 4.0 MM TUBE AT MICROGRAVITY CONDITIONS

Cesar Manuel Valencia Castillo, Universidad Autónoma de San Luis Potosí, Mexico; Luca Gugliemetti, University of Rome "La Sapienza", Italy; Leonardo Bragelli, Guglielmo Marconi University, Italy; Luca Saraceno, ENEA, Italy; Giuseppe Zummo, ENEA, Italy

Results of flow boiling heat transfer and two-phase flow of refrigerant flowing in a 4.0 mm tube at micro-gravity conditions are presented and discussed. The experiments at low gravity were performed during the parabolic flight campaign of October-November 2013. Test were performed in a tube of a Pyrex tube (4.0 mm in diameter), with a transparent metal layer deposited on the outer surface. The metal layer made of ITO allowed to heat the test section. Working fluid was FC-72, widely used in parabolic flights. The observed flow patterns at low gravity varied from bubbly flow up to the intermittent flow and annular flow. Flow pattern visualisation and vapour bubble measurements were performed with a high-speed digital video camera and images were analysed with an image processing software. Vapour bubble parameters, length, width, nose velocity, and frequency, were measured and presented with their relations with heat transfers and flow patterns at zero gravity. At microgravity bubbles were found larger than those at normal gravity with lower velocity (due to the absence of buoyancy), almost linear trajectories, and absence of deformation in comparison to those observed at earth conditions. Larger bubble diameters are controlled by two mechanisms: detachment diameter and coalescence rate.

## BINARY DROPLET INTERACTIONS AND COALESCENCE – SUPERPOSITION OF HYDRODYNAMICS AND IONIC EFFECTS

Felix Gebauer, TU Kaiserslautern, Germany; Jorn Villvock, TU Berlin, Germany; Matthias Kraume, TU Berlin, Germany; Hans-Jorg Bart, University of Kaiserslautern, Germany

In spite of extensive studies on coalescence events in liquid-liquid dispersions, these are still not fully understood. The droplet size distribution (DSD) arising from the competition of coalescence and breakage phenomena is an essential part for the process design and optimization. There is still no predictive model available to describe the DSD with a sufficient accuracy. Coalescence terms which consider ionic effects as well as hydrodynamic effects use fitting parameters to adapt the model for the experimental setup. This results in a large variety of the developed models.

Due to the complexity in a liquid/liquid extraction apparatus a decoupling approach was pursued. Performing basic single droplet interactions in a developed single droplet test cell allows the separation of different effects (e.g. ions, mass transfer, droplet size, impact conditions, etc.) influencing the coalescence. The standardized test cell is fully automated and equipped with a high resolution camera system (time:  $>30\text{k fps}$ ; space: macro lens). A quantification occurs with the determination of the coalescence time as well as the coalescence probability. The presence of the most ions ( $\text{Cl}^-$ ,  $\text{NO}_3^-$ ,  $\text{SO}_4^-$  and  $\text{OH}^-$ ) results in a negative effect on the coalescence behavior. The observed superposition by hydrodynamic effects (kinetic energy, oscillation) have a tremendous influence on the coalescence.

\*This work was supported by the German Research Foundation (DFG).

#### CONTACTLESS COALESCENCE TECHNIQUE OF ACOUSTICALLY LEVITATED DROPLETS USING ULTRASONIC TRANSDUCER ARRAY

Ayumu Watanabe, University of Tsukuba, Japan; Soma Watahiki, University of Tsukuba, Japan; Motonori Niwa, University of Tsukuba, Japan; Atsushi Goda, University of Tsukuba, Japan; Tetsuya Kanagawa, University of Tsukuba, Japan; Akiko Kaneko, University of Tsukuba, Japan; Yutaka Abe, University of Tsukuba, Japan

Contactless handling of the droplet can prevent from the container walls, and has been expected to the application for drug discovery, analytical chemistry, and so on. The acoustic levitation method is one of the levitation techniques, and can levitate a droplet by the acoustic radiation pressure, where the sample is trapped at the pressure node of acoustic standing wave formed between an emitter and a reflector. The purpose of this study is the development of technique to transport and handling millimeter-sized pair of droplets in air, and coalesce it without contact. We realized contactless droplets coalescence, i.e., a pair of droplets was levitated in two localized standing waves formed by an ultrasonic transducer array, and two droplets coalesce by closing the standing waves. Then, we observed the coalescence behavior of the droplets via a high-speed video camera. The collision droplets represented the different behaviors depending on the following parameters: the change of acoustic radiation pressure acting on the droplets, physical properties and diameter of the droplets, and a collision speed of the droplets. Finally, we categorized the behaviors of collision droplets into levitation, falling, and breaking up.

#### STRONGLY COUPLED PARTICLE-LADEN FLOWS IN VERTICAL CHANNELS

Rodney Fox, Iowa State University, USA; Jesse Capecelatro, University of Illinois at Urbana-Champaign, USA; Olivier Desjardins, Cornell University, USA  
Eulerian-Lagrangian (EL) simulations of strongly coupled (high-mass-loading) particle-laden flows in vertical channels are performed with the purpose of (i) exploring the fundamental physics and (ii) modeling of wall-bounded multiphase turbulence. The unclosed terms in the exact Reynolds-averaged equations for high-mass-loading suspensions are evaluated from the EL data. The particle velocity fluctuations are partitioned into spatially correlated and uncorrelated components, corresponding to the particle-phase turbulent kinetic energy (TKE) and granular temperature, respectively. Unlike in low-mass-loading turbulent channel flows, fluid-phase TKE is mainly produced by momentum coupling with the particles (through the fluid velocity seen by the particles), which is transferred to the particle phase through drag exchange. Clusters are observed to fall at the channel walls, leading to locally reduced drag and large values of the particle-phase TKE. Nevertheless, very little momentum transport occurs in the wall-normal direction in either phase due to the strong anisotropy caused by falling clusters. It is shown that the anisotropy of the particle-phase Reynolds stresses determines the particle-phase volume fraction profile, and therefore it must be correctly captured when developing multiphase turbulence models. A two-phase Reynolds-stress model (RSM), based on the Launder, Reece and Rodi isotropization of production (LRR-IP) RSM, is presented. The EL simulations are used to validate the LRR-IP RSM, which is shown to predict correctly the wall-normal distributions of the two-phase turbulence statistics.

#### PREFERENTIAL ACCUMULATION AND ENHANCED RELATIVE VELOCITY OF INERTIAL DROPLETS DUE TO INTERACTIONS WITH HOMOGENEOUS ISOTROPIC TURBULENCE

Colin Bateson, University of Washington, USA; Alberto Aliseda, University of Washington, USA

We present results from wind tunnel experiments that measure the evolution of small inertial ( $\approx 5\text{--}100\text{ }\mu\text{m}$ ) water droplets in homogeneous, isotropic, grid turbulence. High-speed imaging of the light scattered by the droplets from a thin laser sheet are acquired and analyzed via a Particle Tracking algorithm. Droplet trajectories are resolved with high spatial ( $\approx 10\text{ }\mu\text{m}$ ) and temporal resolution ( $\approx 200\text{ }\mu\text{s}$ ), yielding high quality statistics of velocity and acceleration over  $O(10^7)$  droplets. To shed light over the turbulence collision kernel for inertial particles, we analyze the preferential concentration and enhanced relative velocities resulting from their inertial interactions with the underlying turbulence. The particle relative velocity, measured from tracks along a streamwise plane, is conditionally analyzed with respect to the distance from the nearest particle. We focus on the non-normality of the statistics for the particle-particle separation velocity: a negative bias in the separation velocity of particles for short separations indicates a tendency of particles to collide more frequently than a random agitation by turbulence would predict. We use Voronoi tessellations to track clusters, measuring for the first time their lifetime statistics, and calculate droplet settling velocity statistics inside and outside clusters, validating the hypothesis that clusters enhance droplet settling.

#### DIRECT NUMERICAL SIMULATION OF THE FORMATION OF SEDIMENT PATTERNS UNDER THE ACTION OF AN OSCILLATING FLOW

Marco Mazzuoli, University of Genoa, Italy; Aman Kidanemariam, Karlsruhe Institute of Technology (KIT), Germany; Paolo Blondeaux, University of Genoa, Italy; Giovanna Vittori, University of Genoa, Italy; Markus Uhlmann, Karlsruhe Institute of Technology (IfH), Germany

The early stages of the formation of bedforms in an oscillatory boundary layer are investigated by means of Direct Numerical Simulations. Sediment grains are approximated by spherical inertial particles with relative density  $s=2.65$ . The flow is fully resolved about each particle with an immersed boundary technique while particle collisions are accounted for with a discrete element method. The investigation consisted of two parts. At first, six DNSs were performed for such values of the flow and particle Reynolds numbers that no separation occurred behind the spheres. The particles were initially aligned along the main-flow direction and deposited either on a smooth wall or over a layer of resting well-packed spheres randomly arranged on the bottom. In all the DNSs, particles lose their alignment in few oscillating cycles, migrate in the flow-orthogonal direction and form transversal chains under the action of an upgrowing non-periodic secondary flow. The process matches the experimental evidences describing this phenomenon. In the second part, the formation of rolling-grain ripples was simulated with an analogous approach using  $O(2 \times 10^5)$  particles arranged in 13-15 layers. DNSs reproduced two of the experiments performed by Blondeaux et al., 1988. A description is presently given.

#### INVESTIGATION OF SUBGRID SCALE EFFECTS TO PARTICLE PAIR DYNAMICS WITH COHERENT VORTEX EXTRACTION

Yan Xiong, HuaZhong University Of Science and Technology, China; Jing Li, Huazhong University of Science and Technology, China; Zhaohui Liu, Xi'an Jiaotong University, China; Chuguang Zheng, Huazhong University of Science and Technology, China

The subgrid-scale motions relevant to the particles pair dynamics represent a great challenge to the LES of turbulent particle-laden flows. In order to address this challenge, we examine the capability of FDNS (filtered DNS) to predict the collision-related statistics such as the radial distribution function  $g(r)$ , the radial relative velocity  $w_r$  and so as to the geometric collision rate for a wide range of particle Stokes numbers. In this paper, we introduce a wavelet filter to obtain the coherent vortices with a purely gaussian random flow field out of any coherent structures left behind. Compared to classical Tophat or spectral filters which just cut off the high wavenumber components regardless of any effects of the flow structures in SGS motions, we can retain the most vortex-based structures even in the SGS field with the help of high compression rate of wavelet. In this way we can justify whether the so-called SGS coherent structures dominate these effects regarding to the particle-pair dynamics. The results show that the errors result from filtering process reduce obviously with this structural-based filter and that exhibit a promising tool for LES modeling of particle laden flow.

#### COMPARISON OF VOID FRACTION MEASUREMENTS USING DIFFERENT TECHNIQUES IN DOWNWARD TWO-PHASE FLOW

Masahiro Kawaji, City College of New York, USA; Dinesh Kalaga, City College of New York, USA; Damon Turney, City College of New York, USA; Freddy Hernandez, City College of New York, USA; Simon Kleinbart, City College of New York, USA; Jorge Pulido, LanzaTech, USA

Local, area and volume average void fraction measurements have been conducted in downward gas-liquid two-phase flow through vertical pipes of different diameters. The measurement techniques used include gamma densitometry, electric resistance tomography (ERT), wire mesh sensor, optical void probe, borescope coupled to a high-speed video camera, and differential pressure transducer. Each technique has its own advantages but the measurements made using different techniques must be consistent with all others. Thus, the consistency among different measurement techniques is examined in this work by first comparing the local void fraction measurements made using the optical void probe and borescope coupled to a high-speed video camera. It is found that the direction of the optical void probe with respect to the flow has a large influence on the measured local void fraction, especially when the downward two-phase flow contains both very small and relatively large bubbles. Next, the ERT and wire mesh sensor data are compared to check their agreement on two-dimensional void fraction distributions as well as the area-average void fraction. Finally, chordal void fraction data obtained with a gamma densitometer are converted to radial void fraction profiles using a body of surface revolution method in order to compare with radial void fraction profiles obtained by traversing the void probe and borescope coupled to a high-speed video camera.

#### LOW GAS LOADING FLOWS IN DOWNWARD INCLINED PIPES

Stein Tore Johansen, SINTEF, Norway; Sjur Mo, SINTEF Materials and Chemistry, Norway; Ernst A. Meese, SINTEF Materials and Chemistry, Norway

In downward inclined two-phase flow gas may start to accumulate to a significant level as the liquid flowrate falls below a critical rate. The physics of such flows has been studied by simulations using a Quasi 3-dimensional (Q3D) flow model. This model has been presented in previous ICMF papers. In particular, the air water experiments from Pothof in low gas loading downward inclined flows have been analyzed. In the experiments it was found a significant scaling with pipe diameter that could not be explained by the Froude number. Simulations have been carried out a selection of experimental points. The results, including diameter scaling will be presented and discussed. The simulations have revealed that the accurate treatment and observations of the downstream conditions are critical in such flow cases. These issues will be discussed.

In industrial applications 1D flow models are often used. By multidimensional analyses of the flow the observed phenomena can be interpreted at a very detailed level. This understanding can be fundamental in order to develop better 1D flow model closures for such flows.

#### AN EXPERIMENTAL INVESTIGATION OF INTERFACE SHAPE IN SEPARATED FLOW REGIMES INSIDE THE INCLINED TUBE.

Taehwan Ahn, Pusan National University, Republic of Korea; Byongjo Yun, Pusan National University, Republic of Korea; Jaejun Jeong, Pusan National University, Republic of Korea

Interface shape in separated flow regimes such as a stratified flow and annular flow is one of major parameters for predicting a pressure drop and void fraction in two-fluid model. In this work, an experimental investigation has been carried out to measure the parameters on the interface characteristics of separated flows in a 40 mm diameter circular tube with inclination angle ranging from  $-30^\circ$  to  $30^\circ$ . An atmospheric air-water was used at superficial velocities of  $j_G=1.0\sim 50$  m/s for gas, and  $j_L=0.01\sim 1.0$  m/s for liquid. Time-averaged film thickness distribution and wetted wall fraction were measured under steady-state conditions by conductance sensors installed on a circumferentially rotating measurement section. Furthermore, measurement methods were verified by comparison with image processing results of the cross-sectional visualization under a curved interface conditions. From the wetted wall fraction data obtained, a new correlation was developed to cover a much wider range of flow conditions and angle of inclination than those of previous models. In addition, the void fraction obtained from the film thickness distribution data was used to evaluate the momentum balance applying the new interface shape model, and the development direction for a better phenomenological model was suggested.

#### AN EXPERIMENTAL INVESTIGATION INTO THE CLUSTERING OF SPHERICAL BUBBLES RISING ALONG AN INCLINED FLAT PLATE

Toshiyuki Ogasawara, Osaka Prefecture University, Japan; Shota Shirai, Osaka Prefecture University, Japan; Hiroyuki Takahira, Osaka Prefecture University, Japan

The motion of mono-dispersed spherical bubbles rising along an inclined flat plate is investigated experimentally. An inclination angle of the flat plate is controlled to change the bubble Reynolds number in the range from 100 to 200. As experimental conditions, the inclination angle of the plate, the number density of bubbles and also the boundary condition on the bubble surface (free-slip and no-slip conditions) are varied, and their effects on the motion of the bubbles are analyzed.  $MgSO_4$  solution and Triton X-100 solution are used to achieve free-slip and no-slip bubble surfaces, respectively. In both cases, bubble coalescence is almost inhibited. When the number density of bubbles becomes higher and the rising velocity along an inclined flat becomes slower, the bubbles tend to line up horizontally and such arrangements pile up to the bubble cluster. The radial pair distribution function and the conditional average of the relative velocity of two bubbles are calculated to evaluate the spatial distribution and bubble-bubble interaction quantitatively. The results shows that configurations of two close bubble configuration frequently appear, the horizontally-oriented one being stable compared to the vertically-oriented one. A qualitative tendency of the clustering motion is confirmed when the no-slip condition is imposed on the bubble surface, however, quantitative differences occur due to the difference in the wake structure between these two conditions.

\* Support from JSPS KAKENHI Grant Number 15K17976 is gratefully acknowledged.

#### STUDIES OF DOWNWARD GAS-LIQUID ANNULAR FLOW

Sethu Anupriya, IIT Madras, India; Sreenivas Jayanti, IIT Madras, India

Vertically downward gas-liquid annular flow may occur in vertical U-tube heat exchangers and is thus of practical interest. It also presents an interesting contrast with upward annular flow as the influence of gravity is reversed. While this may have a negligible effect on the gas phase flow, the effect on the liquid film may be considerable. The presence of a faster-moving liquid film may also influence the dynamic processes of entrainment and deposition. However, downward gas-liquid annular flow has not been investigated widely. In the present study, we report on pressure drop measurements of vertical upward and downward air-water annular flows conducted in tubes of 0.025, 0.038 and 0.050 m inner diameter at atmospheric pressure and room temperature conditions. Water and air flow rates have been varied over the Reynolds number range of 0 to 8500 and 25000 to 250000, respectively. Existing models of air-water annular flow have been found to predict the upward flow data well. Application to the downward flow shows strong overprediction of the pressure drop data at low water Reynolds numbers and underprediction at high water Reynolds numbers. Further studies are being conducted to develop a modified model for gas-liquid downward annular flow.

#### MODELLING OF TWO SPHERICAL OSCILLATING BUBBLES IN AN ELASTIC TUBE

Hiroyuki Takahira, Osaka Prefecture University, Japan; Tatsuya Kitahara, Osaka Prefecture University, Japan; Toshiyuki Ogasawara, Osaka Prefecture University, Japan

Bubble oscillations in an elastic tube are important for medical applications using nano/microbubbles such as ultrasound contrast agents and high intensity focused ultrasound. The present study deals with the interactions between two bubbles in an elastic tube to understand the bubble dynamics in a blood vessel. The boundary integral equations for two spherical gas bubbles in the tube are derived: The tube wall is modeled with a spring-mass system and each bubble is represented by a point source for the bubble volume oscillation and a dipole for the translational motion. The natural frequencies of the bubble-elastic tube system are derived by linearizing the boundary integral equations. In the present work, the influences of the characteristics of the tube on two interacting bubbles are mainly discussed by using eigenvalue analysis. Two bubble resonance frequency modes are obtained from the eigenvalue analysis. It is shown that the lower resonance frequency of the bubble corresponds to the mode in which both bubbles oscillate with the same phase; the tube wall also oscillates with the same phase. The higher corresponds to the mode in which one bubble oscillates with inverse phase to the other; the tube wall symmetrically oscillates with the same phase to the bubble with respect to the tube center. The effects of the tube length and the initial bubble location on the resonance frequencies are also investigated. We also discuss the nonlinear bubble oscillations in the tube.

\* Supports from JSPS KAKENHI Grant Number 15K13876 are gratefully acknowledged.

#### INTERACTION OF TWO BUBBLES RISING IN A THIN-GAP CELL: EXPERIMENTS AND MODEL FOR THE VERTICAL ENTRAINMENT

Veronique Roig, INP Toulouse, France; Audrey Filella, INP Toulouse, France; Patricia Ern, INP Toulouse, France

We report experiments on the hydrodynamical interaction of oscillating bubbles rising in line at high Reynolds number in a liquid otherwise at rest that is confined in a vertical thin gap cell. From a precise description of bubble motions and measurements of velocity perturbations induced in the liquid by HF PIV, a detailed characterization of the vertical entrainment of the trailing bubble in the wake of the leading bubble is provided. Experiments performed for a large range of bubble sizes, reveal that entrainment is sensitive to the relative size of the bubbles. A dynamical model of entrainment that takes into account the axial evolution of the wake of the leading bubble is proposed to reproduce the acceleration observed for the trailing bubble. This model also includes the axial evolution of the potential flow preceding the trailing bubble that explains the final acceleration of the leading bubble occurring during interaction when the trailing one is clearly larger.

## ADHESION AND SLIDING DYNAMICS OF AIR BUBBLES ON SUPERAEROPHOBIC SURFACES

Ridvan Ozbay, Stevens Institute of Technology, USA; Ali Kibar, Kocaeli University, Turkey; Chang-Hwan Choi, Stevens Institute of Technology, USA

In this study, experimental analyses of the adhesion force of captive air bubbles on superaerophobic surfaces are reported compared to that on aerophobic surfaces. A bare silicon substrate and a micropillared silicon substrate were used as the aerophobic and superaerophobic surfaces, respectively. The lateral adhesion (i.e., friction or depinning) force of air bubbles on the surfaces was analyzed on the basis of sliding at inclination. Theoretically, the adhesion force of an air bubble to an inclined surface depends on contact width and contact angle hysteresis, and it is balanced with other external forces including effective buoyancy force and pressure forces. The results indicate that the sliding angle, contact angle hysteresis, adhesion force are much lower for the micropillared superaerophobic surface than for the aerophobic surface, which is mainly due to the entrapped water layer into the micropillars. It is also observed that the formation of a water layer between the bubble and the superaerophobic surface is more stable than that between the bubble and the aerophobic surface. The adhesion force slightly decreases with an increase in bubble volume for both aerophobic and superaerophobic surfaces. Although the contact width increases with bubble volume, the contact angle hysteresis decreases more dramatically so that the adhesion force is effectively reduced with an increase in bubble volume. This is attributed to the effect of buoyancy force, which becomes more significant with an increase in bubble volume (i.e., larger Bond number) and gets significantly affected by the surface morphology.

## FLOW STRUCTURES ANALYSIS DOWNSTREAM GAS SPARGERS OF DIFFERENT ORIFICE PATTERN IN BUBBLE COLUMNS

Felix Moeller, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Andre Bieberle, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Frank Barthel, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Markus Schubert, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Uwe Hampel, TUDresden, Germany; Manfred Weber, INEOS Phenol GmbH, Germany; Markus Weber, INEOS Phenol GmbH, Germany

Bubble column reactors are widely employed for gas-liquid reactions due to their superior mass and heat transfer accompanied by being geometrically simple and free from any moving part. The gas sparger is a crucial device, which directly determines the process performance. The bubbles released from the orifices of the sparger start rising up and buoyancy and drag forces, in turn, depending on bubble size, control the bubble traveling velocity, and eventually the overall hydrodynamic behavior, such as mixing and gas holdup. Hence, to get an insight into the impact of orifice patterns and holes' diameter of a gas sparger on the axially evolving gas-liquid dispersion, an ultrafast X-ray tomography study is performed in a 2 m height bubble column of 100 mm diameter. Two different perforated plate-type spargers are used, namely a single orifice sparger with a 2.9 mm center hole and a plate with 13 orifices arranged in a triangular pitch with a hole diameter of 0.8 mm, respectively. Experiments were performed in a non-coalescing system for superficial gas velocities up to 2.5 cm/s. From the X-ray measurements, the evolving gas structures, bubble size distributions and cross-sectional gas holdup distributions are extracted along the column height and the benefit of a fine initial distribution is evaluated.

\*Support from the ERC for funding the XFLOW-ERC Starting Grant (Grant Agreement No. 307360) is gratefully acknowledged.

## INVESTIGATION OF SINGLE BUBBLES AND BUBBLE CHAINS IN LIQUID METAL UNDER THE INFLUENCE OF MAGNETIC FIELDS

Erik Strumpf, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Natalia Shevchenko, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Olga Roshchupkina, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Sven Eckert, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany

Two-phase flows play an important role in industrial applications, such as the continuous casting of steel. Here, inert gas is injected into the beginning of the submerged entry nozzle to avoid nozzle clogging as well as to catch impurities within the melt and to transport them to the free surface, away from the solidification front. Furthermore electromagnetic brakes are used to dampen the highly turbulent flow and to reduce high velocities in the liquid metal. Although a vast number of simulations and experiments of bubbly flows in water exist, investigations in liquid metal are still lacking. However, liquid metal experiments are unavoidable for a correct modelling of such special cases, like the influence of magnetic fields on the flow or the two-phase flow behavior. In this study the ascents of single bubbles and of bubble chains in a liquid metal are investigated. For this purpose a benchmark experiment is set up, a cuboid vessel of the dimensions 144 x 144 x 12 mm<sup>3</sup>, which is filled with eutectic alloy GaInSn. A transversal magnetic field up to 1.1 T is imposed to this vessel. Ultrasound Doppler Velocimetry (UDV) is used to map the flow in the continuous phase of bubble chain regimes as well as the ascent velocity of single bubbles. X-ray radiography is applied to obtain detailed information within the disperse phase of bubble chains, such as bubble diameter, shape, trajectory etc. The combined results of both measurement techniques presented in this paper give a detailed overview of flow properties.

## EXPERIMENTAL INVESTIGATION OF HEAT TRANSPORT IN BUBBLY FLOW

Biljana Gvozdic, University of Twente, The Netherlands; Elise Almeras, University of Twente, The Netherlands; Varghese Mathai, University of Twente, The Netherlands; Chao Sun, University of Twente, The Netherlands; Detlef Lohse, University of Twente, The Netherlands

Bubbly flows coupled with heat transport are relevant for many applications in industry. Previous studies have shown that injection of bubbles into a carrier fluid effectively enhances the convective heat transfer in forced and natural convection systems. Studies on the local convective heat flux are crucial to disentangle the role of the injected bubbles and the turbulent fluctuations induced by the bubbles. The exact mechanism behind this phenomenon is still unclear since previous heat transport measurements in bubbly flows are limited to time-averaged global quantities. In this study we perform measurements of local heat flux fluctuations along with global fluxes in a homogeneous bubbly flow. The experiments are conducted in a rectangular bubble column heated from one side. Laser Doppler Anemometry and thermistors are used for simultaneous measurements of liquid velocity and temperature fluctuations at nearly same point in space. We study the statistical properties of the flow, PDFs and power spectrum of the fluctuations of temperature, velocity and local heat flux. We present the results on heat transport enhancement due to bubble injection (bubble diameter ~ 3 mm) for different gas volume fractions (0.03% - 5%).

## ON THE FORMATION OF MESO-SCALE STRUCTURES IN DISPERSED GAS-LIQUID FLOWS

Henrik Strom, Chalmers University of Technology, Sweden; Klas Jareteg, Chalmers University of Technology, Sweden; Srdjan Sasic, Chalmers University of Technology, Sweden; Christophe Demaziere, Chalmers University of Technology, Sweden

Meso-scale structures in gas-liquid flows can be viewed as non-uniformities in the volume fraction fields whose characteristic sizes range from several dispersed phase diameters to the size of the bounding geometry. The formation of such structures is due to clustering of the dispersed phase and takes place as the uniformly bubbling state loses stability at some critical bubble volume fraction. A two-fluid model can be used to predict the formation of meso-scale structures, but the analysis is intricate due to the mathematical difficulties associated with these models. The appearance of instabilities in numerical solutions to the two-fluid model is therefore often suppressed by the use of excessively dissipative numerical schemes or by regularization of the physical model. In this work, we use a conventional two-fluid model, based on the one-pressure formulation, to investigate the stability of a uniformly bubbling suspension in a periodic domain. We show how the manifestation of meso-scale structures is affected by the physical parameters of the problem (e.g. bubble size and loading), the model formulation (e.g. modeling of the momentum exchange), and also on the numerical algorithm employed (e.g. discretization schemes). Our results illustrate a viable route in the derivation of a numerical framework that can resolve physical meso-scale structures whilst suppressing unphysical instabilities in dispersed gas-liquid flows. \*This research was conducted with funding from the Swedish Research Council (Vetenskapsrådet) as a part of the DREAM4SAFER framework grant (contract number C0467701).

## BOILING FRONT FORMATION IN FLASH DEPRESSURIZATION

Henda Djeridi, LEGI -Université de Grenoble, France; Jean-Philippe Matas, Université Claude Bernard Lyon and CNRS, France; Pierre Boivin, M2P2, France; Alain Cartellier, Université Grenoble Alpes and CNRS, France

We study the flash vaporization of FC-72. In a previous experiment a jet of FC-72 was injected into a depressurized reservoir and we followed the pressure rise and measured drop sizes. Results surprisingly showed that provided the walls were thermalized below T<sub>sat</sub>, the vaporization rate and resulting drop sizes were independent of the reservoir pressure, and hence of the superheat (Desnoux et al CRM 2013). In the present configuration the liquid is at rest in a glass tube (diameter 2.4cm): when the glass tube is connected to the depressurized reservoir a boiling front forms. The velocity of this boiling front is measured for several reservoir pressures: we find that front velocity is independent of superheat, in agreement with the 1st experiment. If the initial temperature of the liquid is increased (the tube is immersed in a thermalized bath) the velocity of the boiling front increases. These results are consistent with the results of a numerical simulation (code Leonard). Finally, drop sizes and velocities of entrained drops are measured for various superheats (PDI), and confirm the results of the first series of experiments.

We will discuss these results and offer, via a simple model, an explanation for this saturation of the vaporization rate.

### COMPARATIVE ASSESSMENT OF FLOW BOILING HEAT TRANSFER CORRELATIONS FOR HORIZONTAL MINI AND MICRO CHANNELS WITH CO<sub>2</sub> AS WORKING FLUID

Jigar Gamit, Sardar Vallabhbhai National Institute of Technology, India; Salim Channiwala, Sardar Vallabhbhai National Institute of Technology, India

Carbon dioxide has been reintroduced as a potential working fluid for heat pumps, subcritical cascade refrigeration and as a secondary fluid (volatile brine) in industrial refrigeration due to its thermo-physical properties and its impact on the environment (zero ODP and GWP of 1). Effort has been made to develop a better correlation for flow boiling heat transfer with CO<sub>2</sub> as working fluid. As an initial step, comparative study has been carried out with 3425 experimental data collected from 12 research papers pertaining to CO<sub>2</sub> as a working fluid. 24 flow boiling heat transfer correlations were reviewed and compared with the collected data. It was found that no correlation has satisfactory accuracy, even the ones developed for CO<sub>2</sub>. The best one has a root mean square deviation of 67.88%.

### A MECHANISTIC IR CALIBRATION TECHNIQUE FOR BOILING HEAT TRANSFER INVESTIGATIONS

Andrew Richenderfer, Massachusetts Institute of Technology, USA; Matteo Bucci, Massachusetts Institute of Technology, USA; Thomas Mckrell, Massachusetts Institute of Technology, USA; Jacopo Buongiorno, Massachusetts Institute of Technology, USA

A new surface temperature calibration technique for Infrared Thermography has been developed for heat transfer diagnostics. The calibration technique uses a coupled conduction-radiation analysis to calculate the complete temperature distribution within the heated substrate and the surface heat flux distribution. This calibration technique improves upon previous empirical and analytical calibration techniques two-fold. First, the signal received by the infrared camera is quantified and analyzed using the properties of the camera to calculate the photon flux. Second, this calibration technique accounts for variations in the optical properties of the heated substrate over the radiation spectrum. Previous techniques used either an empirical calibration, or a simplified, spectrally-averaged analysis.

To demonstrate the technique, the analysis has been applied to a transient boiling study. An infrared camera is used to capture the boiling surface at 2500 fps with an integration time of 0.4 ms. The resolution of the camera is 116  $\mu\text{m}$  x 116  $\mu\text{m}$ . The pixel size also corresponds to the mesh size in the conduction analysis so there is a 1-to-1 match for the heater boundary condition. To validate the radiation calculation a temperature controlled blackbody was used with a known emissivity. Validation of the complete model was done using controlled single-phase transient conduction tests. The model yielded temperatures within 0.1°C of the directly measured temperature, whereas typical empirical calibration curves have accuracies closer to 2 °C.

### STRUCTURE OF A LASER-INDUCED UNDERWATER SHOCK WAVE

Yoshiyuki Tagawa, Tokyo University of Agriculture and Technology, Japan; Shota Yamamoto, Tokyo University of Agriculture and Technology, Japan; Keisuke Hayasaka, Tokyo University of Agriculture and Technology, Japan; Masaharu Kameda, Tokyo University of Agriculture and Technology, Japan

A laser-induced underwater shock wave is used in various applications such as medical devices. Its structure has been often regarded as spherically symmetric. In this study we measure the shock using a combined system, which obtains nanosecond-order image sequences of plasma formation, shock-wave expansion, and temporal evolution of shock pressure simultaneously. Remarkably, we find the non-spherically-symmetric distribution of peak pressure for a wide range of experimental parameters. The maximum pressure is more than four times larger than the minimum pressure. This result clearly differs from the spherical-shock model. However, in contrast to the result for peak pressure, distribution of pressure impulse that is the integration for the pressure with respect to the elapsed time is spherically symmetric. Based on these measurements, we suggest a multiple-structure model instead of the spherical-shock model: The laser-induced shock wave is not a single spherical shock but a collection of spherical shock waves. These spherical shock waves are emitted from a collection of plasmas formed in a fine region illuminated by a laser pulse. The experimental results reveal that the number of plasmas relates strongly to the pressure distribution.

\* Support from JSPS KAKENHI Grant Number 26709007 is gratefully acknowledged.

### A LOOK ON RELEVANT NONDIMENSIONAL NUMBERS FOR FLOW BOILING IN MICROCHANNELS AND MINICHANNELS

Mohamed Awad, Mansoura University, Egypt; Zan Wu, Lund University, Sweden; Bengt Sundén, Lund University, Sweden

In this study, a look on relevant nondimensional numbers for flow boiling in microchannels and minichannels is presented. These nondimensional numbers include Bond number, Froude number, Weber number, Capillary number, Reynolds number, Boiling number, Fang number, Kandlikar first number, Kandlikar second number, Kandlikar third number, and Garimella number (convective confinement number). The literature data plotted in this study were extracted from 25 datasets. Only the stable data points located within the elongated bubbly flow and the annular flow were collected. Flow regime transitions for flow boiling were calculated based on Ong and Thome (2011) and Revellin and Thome (2007). The collected database includes 2573 data points for 15 different working fluids. The water data points from Sumith et al. (2003) (i.e., the only dataset for water) with a diameter of 1.45 mm, show very different behaviors, e.g., with low FrLO and WeLO values, and high Fa, K1 and K3 values. The different behaviors of water might be due to its unique properties, such as a high level of surface tension.

### EXPERIMENTAL AND NUMERICAL INVESTIGATION OF FLASH ATOMIZATION WITH CRYOGENIC FLUIDS

Meng Luo, Technische Universität München, Germany; Thomas Ramcke, Universität der Bundeswehr München, Germany; Michael Pfitzner, Universität der Bundeswehr München, Germany; Oskar Haidn, Technische Universität München, Germany

Flashing happens when the fluid deviates from its saturation conditions triggering a huge nucleation, bubble growth and massive atomization and vaporization. In this paper, the experimental and numerical study is conducted in terms of exploring the cryogenic fluids behavior (LN<sub>2</sub> or/and Lox) in low pressure environment with injection through a cylindrical nozzle. The high-speed shadowgraph technique was employed for the morphological study of the flashing spray pattern and also the thermal behavior of the spray was obtained by an intrusive measurement technique with a rack of several thermocouples located in the spray centerline. Experimental results obtained for different initial pressure and temperature (or superheat level) will be compared to the numerical results. As for the numerical part, the Euler-Lagrange approach is employed. Considering the flashing boiling effect, a superheated flash droplet evaporation model has been implemented into a CFD-code in order to take account of the resulting source terms for the Eulerian and Lagrangian phase, respectively. This model is based on an internal heat transfer coefficient modeling the superheat provided by the droplet. The range of applicability of the modeling approach for the conditions tested will be the results of a comparison of experimental and numerical data.

\* Support from grant MUNICH AEROSPACE and China Scholarship Council (CSC) is gratefully acknowledged.

### CAVITY SHEDDING MECHANISMS ON A NACA0015 HYDROFOIL

Harish Ganesh, University of Michigan, USA; Juliana Wu, University of Michigan, USA; Steven L. Ceccio, University of Michigan, USA

Recent investigations of partial cavitation have shown that the transition from stable to shedding cavities can be related to the presence of both propagating bubbly shocks and re-entrant liquid jets originating in the cavity closure region. In the present study, formation of sheet cavitation and its transition to periodically shedding cavities is studied on a NACA0015 hydrofoil in a recirculating water tunnel at different attack angles. Using high-speed videos, time resolved X-ray densitometry, and unsteady pressure measurements, the instantaneous void fraction flow fields are obtained to identify the principal mechanism responsible for transition from stable to shedding cavities over a range of attack angles and cavitation numbers. A map of the kind of cavitation dynamics observed and the corresponding mechanism in the attack angle cavitation number parameter space is then presented. The role of attack angle, by the virtue of its relationship to the pressure gradient at cavity enclosure, in the observed mechanism is then explored.

\* This work was funded by U.S. Office of Naval Research, grant number N00014-14-1-0292, under program manager Dr. Ki-han Kim.



## JATROPHA BIODIESEL PRODUCTION USING HYDRODYNAMIC CAVITATION

Prateek Dwivedi, IIT Roorkee, India; Sumana Ghosh, Indian Institute of Technology Roorkee, India

Cavitation is the process of formation, subsequent growth and collapse of bubbles in fluid, releasing large magnitude of energy. Both acoustic and hydrodynamic cavitation have been found to be efficient in producing the desired chemical changes in processing applications for example biomedical waste water treatment etc. The chemical effects of cavitation have been extensively investigated during recent years. Several studies have been reported on chemical effect of acoustic cavitation. The major challenge in this field is to identify and control the parameters those influence cavitation because cavitation bubbles mutually interact, often in an uncontrolled way.

The present work deals with an experimental investigation which find out the influence of different parameters (liquid flowrate, temperature etc.) on the pressure signal generated by the collapse of bubbles in horizontal tube fitted with sudden contraction. Aim is to design an experimental setup to induce chemical reactions by of controlling these variables separately to achieve a optimum conversion for chemical reactions. The final aim is to carry out the transesterification and establish yield of reaction between alcohol and vegetable oil to produce biodiesel using hydrodynamic cavitation in the designed experimental set up. The experimental setup is tested with Weissler reaction and showed excellent conversion.

## CHARACTERIZATION OF FLOC MORPHOLOGY IN A TURBULENT TAYLOR-COUETTE REACTOR

Lea Guerin, Chemical Engineering Laboratory, France; Carole Coufort Saudejaud, Chemical Engineering Laboratory, France; Alain Line, University of Toulouse, France; Christine Frances, Chemical Engineering Laboratory, France

Many industrial processes fields are dealing with solid-liquid suspensions. In these fields, the size distribution and the morphology of the aggregates can strongly affect the dispersed solid phase properties and modify its behavior.

This work is about the acquisition of an experimental data base concerning the changes of morphological parameters of aggregates during flocculation runs in a Taylor-Couette reactor under turbulent conditions. Aggregation of spherical latex particles was promoted by salt or polymer addition. In order to highlight the impact of hydrodynamics on physical phenomena such as aggregation, breakage or restructuring, hydrodynamic sequenced experiments with alternately low and high shear conditions have been performed.

On line laser diffraction analysis was done to obtain the volume size distribution of the aggregates across time. Morphological analysis was also conducted using a device coupling microscopy and image analysis giving access to multiple size and shape parameters.

Although size and shape are obviously correlated, this work points out that their dependency to hydrodynamic can be different depending on operating conditions. The morphological properties of aggregates are also affected by the history of the suspension.

## NUMERICAL STUDY OF AGGREGATION DYNAMICS WITH A LATTICE BOLTZMANN METHOD COUPLED WITH DISCRETE ELEMENTS THROUGH FLOW PENALIZATION

Jeff Ngoma, Université de Lorraine, France; Jean-Sebastien Kroll-Rabotin, Université de Lorraine, France

The lack of knowledge on aggregation processes in liquid metal reactors is a major hindrance to industrial needs for continuous improvement of inclusion cleanliness in high performance alloys. This study details a method for numerical investigations at a mesoscopic scale, where inclusion geometries and aggregate morphologies are fully resolved and where models are available for contacts and distance interactions between inclusions. The flow dynamics are computed with an explicit lattice Boltzmann method (LBM) since the time scales are constrained by the interactions between solids anyway. Solid-solid and fluid-solid interactions are captured by coupling a discrete element method (DEM) with a penalization method in a similar way as immersed boundary methods (IBM) or force coupling methods (FCM). The whole method aims at remaining computationally cheap while remaining relatively effective to represent particles of any shape and to capture their hydrodynamic interactions.

Preliminary applications to particle interactions are presented to illustrate the purpose of the method, that is deriving specific aggregation models. In the longer run, the aim is for such models to serve as aggregation kernels in population balances, which are suitable for reactor scale simulations.

## EXPERIMENTAL INVESTIGATION OF THE LORENTZ-FORCE-DRIVEN CONVECTION AROUND A GROWING HYDROGEN BUBBLE

Dominik Baczymski, Universitaet der Bundeswehr München, Germany; Franziska Kambach, IFW Dresden, Germany; Xuegeng Yang, Technische Universität Dresden, Germany; Gerd Mutschke, Technische Universität Dresden, Germany; Margitta Uhlemann, IFW Dresden, Germany; Kerstin Eckert, Technische Universität Dresden, Germany; Christian Cierpka, Universität der Bundeswehr München, Germany

The production of hydrogen through water electrolysis shows great potential for energy storage systems due to its high energy density. The efficiency of this process is, however, limited by the hydrogen gas bubbles evolving at the electrode and can be enhanced by an accelerated bubble detachment. The application of magnetic fields is a particularly promising method which might reduce the bubble size at detachment. The superposition of the magnetic field with the inherent electric field gives rise to Lorentz forces and results in a strong rotating shear flow around the micron-sized bubble. To investigate the effect of the imposed flow on the periodic growth of a single hydrogen bubble and mass transfer toward the electrode, experiments were conducted at a microelectrode under a magnetic field. The three-dimensional flow around the evolving bubble was measured by different particle imaging and tracking techniques, while shadowgraphy and measurements of the electric current served to characterize the bubble growth. The results are supported by numerical simulations and show an extensive acceleration of the fluid around the bubble and a complex unsteady secondary flow which interacts with the wake of the previous bubble and might be significant for the performance of the process.

## JETTING REGIME IN CO-FLOWING CAPILLARY JETS IN CONFINED GEOMETRY

Emilia Nowak, University of Birmingham, UK; Lyes Kahouadji, Imperial College London, UK; Omar K. Matar, Imperial College London, UK; Mark Simmons, University of Birmingham, UK

Microfluidic devices occur in various fields ranging from inkjet printing, DNA chips, lab-on-a-chip technology, micro-propulsion to droplet-based microfluidics where the droplet is formed in either a dripping or jetting mode as a function of We and Ca number. Here, we examine the jetting regime of co-flowing immiscible liquids in laboratory-fabricated confined geometry via high-speed imaging, shadowgraphy and PIV that allows interface topology and flow field tracking. Typical convective as well as absolute instabilities are observed as a function of the liquid properties, flow rates and the capillaries' diameter ratio. For the high viscosity ratio of the inner to outer phase, jet recoil of the inner phase is observed. We also present comparisons with direct numerical simulations using a new solver involving massively-parallel simulations of fully three-dimensional multiphase flows. These are based on high-fidelity hybrid front-tracking/level set algorithm for Lagrangian tracking of arbitrarily deformable phase interfaces including breakup and coalescence, and a precise treatment of surface tension forces, interface advection and mass conservation.

\* Support from the Engineering & Physical Sciences Research Council, UK, through the MEMPHIS program grant is gratefully acknowledged.

## DIRECT NUMERICAL SIMULATIONS OF FLOW AROUND A NON-SPHERICAL PARTICLE

Sathish Pacha Sanjeevi, Eindhoven University of Technology, The Netherlands; Johan Padding, Eindhoven University of Technology, The Netherlands; J. A. M (hans) Kuipers, Eindhoven University of Technology, The Netherlands

Many industrial processes involve fluidization of solid particles in a fluid medium. Computer simulations can be used to predict the hydrodynamics of such processes. Traditionally, it is assumed that the particles are spherical because this largely simplifies the simulation procedure. However, in practice the particles are often non-spherical, which increases the importance of lift forces and hydrodynamic torque around the different axes of the particles. This study aims at analyzing a single highly elongated particle in fluid flow. A highly parallel lattice Boltzmann solver is used to perform direct numerical flow simulations. The results are used to develop closures for drag, lift and torque as a function of Reynolds number and flow incident angle. These closures are useful in much coarser discrete particle method simulations. Extending the single particle simulations, complex three-dimensional multi-particle simulations will be performed to analyze the influence of solids volume fraction on the drag, lift and torque coefficients.

#### A COUPLED EULERIAN-LAGRANGIAN METHOD FOR SPRAY ATOMIZATION ON UNSTRUCTURED MESHES

Fabien Evrard, Imperial College London, UK; Fabian Denner, Imperial College London, UK; Ricardo Serfaty, Petrobras, CENPES, Cidade Universitaria, Brazil; Berend Van Wachem, Imperial College London, UK

Understanding and modelling the behaviour of liquid sprays is key for a wide range of processes. With the recent increase in available parallel-computing resources, direct numerical simulations with interface capturing or interface tracking methods have been able to reveal the complex physics of the primary atomization process, but remain prohibitively expensive. We present a Volume-of-Fluid/Lagrangian particle-tracking coupling approach for the modelling of spray atomization on unstructured meshes. The flow solver has a collocated-arrangement for the variables and is fully coupled. Candidate fluid structures which are small compared to the characteristic length of the liquid jet are removed and replaced by Lagrangian particles evolving within the underlying flow. By doing so, the computational demand is reduced, enabling simulations at a larger scale. We also introduce an algorithm for the detection of filament structures and a breakup model for their replacement by Lagrangian point-particles. The full framework is validated using capillary jet and droplet collision test cases. Finally, an example of an atomizing spray is given.

\* Support from Petrobras is gratefully acknowledged.

#### SIMULATION OF MICROSCOPIC FLUID PARTICLE MOTION ON SOLID SURFACE USING A DIFFUSE-INTERFACE MODEL

Naoki Takada, Nat'l Inst. of Adv. Industrial Science and Technology (AIST), Japan; Junichi Matsumoto, Nat'l Inst. of Adv. Industrial Science and Technology (AIST), Japan; Sohei Matsumoto, Nat'l Inst. of Adv. Industrial Science and Technology (AIST), Japan; Kazuma Kurihara, Nat'l Inst. of Adv. Industrial Science and Technology (AIST), Japan; Ryohei Hokari, Nat'l Inst. of Adv. Industrial Science and Technology (AIST), Japan

Simulations of fluid-particle motion on solid surface with homogeneous or heterogeneous wettability are conducted by using a diffuse-interface tracking method for better understanding microscopic fluid flow phenomena in various engineering fields. The method employs a phase-field approach for modeling fluid-fluid interfacial dynamics based on the free-energy theory, where the interface is regarded as finite volumetric zone between different phases. A lattice-Boltzmann model is employed to second-order accuracy in both space and time for solving a set of Navier-Stokes equations and a conservation-modified Allen-Cahn advection equation. The major findings are as follows: (1) initial circular shape and volume of fluid are well conserved in two-dimensional linear translation; (2) the method predicts well the capillary force effect on departure motion of droplet on solid surface in stagnant liquid under gravity in agreement with semi-empirical predictions; (3) the shape and motion of droplet on solid surface textured with linear grooves is predicted qualitatively well in comparison with available data. The above results prove that the method will be useful for predicting the multiphase fluid motions in MEMS devices, on surface-fabricated substrates and in micro-fabrication processes.

#### PRESSURE DROP THROUGH ORIFICES FOR SINGLE- AND TWO-PHASE VERTICALLY UPWARDS FLOW

Abdelwahid Azzi, University of Sciences and Technology Houari Boumediene, Algeria; Ammar Zeghloul, University of Sciences and Technology Houari Boumediene, Algeria; Faiza Saidj, University of Sciences and Technology Houari Boumediene, Algeria; Abdelkader Messilem, University of Sciences and Technology Houari Boumediene, Algeria; Barry James Azzopardi, University of Nottingham, UK

Orifices are used for metering purposes in various modern engineering fields such as, nuclear, petroleum industry and chemical engineering processes.

The pressure drop through orifice is a key parameter for determining the flowrate and its calculation with accuracy requires a reliable prediction models. In this work, measurements of upward single- and two-phase gas-liquid flow pressure drops through orifices are reported.

For single-phase flow, the pressure drop increases by decreasing the open area ratio or the orifice thickness. It increases with the superficial velocity of the phase. The correlations of Idel'chik and Akhter et al. correlate well the orifice pressure drop coefficient.

For two-phase flow, a decrease in the opening area ratio or in the orifice thickness results in an increasing of the pressure drop. For a fixed liquid superficial velocity, the pressure drop increases with the gas superficial velocity; however for low open area ratio, a decreasing in pressure drop, from a certain superficial gas velocity, follows this increasing.

An assessment of the correlations for predicting two-phase flow pressure drop through orifice has been achieved. It appears that relationships of Morris and Simpson et al. are the most reliable ones.

#### MULTI-MATERIAL MODELLING FOR COMPRESSIBLE FLOWS WITH THE CONSERVATIVE SHARP-INTERFACE METHOD

Shucheng Pan, Technische Universitaet Munchen, Germany; Xiangyu Hu, Technische Universitaet Munchen, Germany; Nikolaus Adams, Technische Universitaet Munchen, Germany

The numerical simulation of compressible multi-material flows has extensive applications in engineering and physics. One challenge for these simulations is the prediction of the dynamics associated with multiple material interfaces. Compared to Lagrangian methods, one main advantage of Eulerian methods is their ability to capture interface topology changes. However, for problems involving three or more materials, most of these methods have difficulties on handling interface singularities, where three or more materials meet. In this work, we present a multi-material method for simulating compressible flows. A multi-region level-set technique, which is able to represent an arbitrary number of materials by a single unsigned level-set function combined with an additional phase indicator, is used to represent the multiple material interfaces. In order to use high-order finite-difference schemes, such as WENO schemes, near singularities, a simple, local and fast phase-reconstruction operator is developed. Furthermore, a fast and accurate method is proposed for reinitializing the multi-region level-set field. In addition, the conservative sharp-interface method, has been extended for flows with three or more materials. A new and fast interface-interaction model is proposed to cope with the complex interaction between multiple materials. Validation results for a range of challenging test cases show that the present method is numerically robust, accurate and computationally significantly more efficient than previously proposed methods.

#### EXPERIMENTAL STUDY ON SLUG FLOW DEVELOPMENT IN A RISER AND VERTICAL TO HORIZONTAL 90° BEND

Rajab Omar, The University of Nottingham, UK; Barry James Azzopardi, University of Nottingham, UK; Buddhikha Hewakandamby, University of Nottingham, UK

The influence of pipe bends on the distribution of the phases has not been studied widely in multiphase flows. This paper demonstrates evolution of the axial slug flow around a vertical to horizontal 90° bend and downstream. Air-Silicone oil (viscosity 5 cP, density 930 kg/m<sup>3</sup>) experiments were conducted a system consists of a 68 mm ID riser and 90° bend and a 9.2 m horizontal pipe. The behaviour of slug characteristics was analysed at three different locations (15, 30 and 45D) upstream and also two locations (69, 127D) downstream of the bend using ECT and WMS simultaneously. The flow around the bend was observed using a high speed camera. The study shows that the bend alters the flow by separating the phases resulting stratified flow downstream of the bend. The formation of the slugs in the horizontal section downstream is influenced by the hydrodynamics and is independent of the flow patterns upstream of the bend for a wide range of flow rates.

#### SOLID VOLUME FRACTION FLUCTUATIONS IN A LIQUID FLUIDIZED BED

Maedeh Marefatallah, University of Alberta, Canada; Sean Sanders, University of Alberta, Canada

In this study, the dynamic behavior of concentrated solid-liquid flow is investigated using a liquid fluidized bed. High speed Electrical Impedance Tomography (EIT) is used to measure the local solid concentration and the concentration fluctuations at two different axial positions over a range of fluidization conditions. The unique aspect of this technique is that it provides local concentration fluctuations even for opaque concentrated mixtures. Only a few experimental studies have reported solid-phase concentration fluctuations and those were limited to averaged values over the cross-section (e.g. Zenit and Hunt, IJMF, vol. 26, 2000).

Particles with different diameters and densities were tested to determine the effects of Stokes number on the nature and magnitude of the concentration fluctuations and on the radial distribution of solids during fluidization. The combination of these results with the previous experimental data available for glass particles studied under similar conditions enables us to provide more accurate models to predict the magnitude of local concentration fluctuations not only as a function of the solid fraction (Buyevich and Kapbasov, ChESci, vol 49, 1994) but also the Stokes number. These models can also be used to correlate the solid concentration fluctuations to the fluctuating kinetic energy of the particles (Gevrin et al., AIChE J., vol.56, 2010), which in turn can be used to predict the solid phase pressure and the solid phase stress tensor terms in the models using kinetic energy approach (e.g. Gidaspow, 1934).

## CHARACTERIZATION OF DYNAMICS OF THREE-PHASE SLURRY FLOW USING GAS VOLUME FRACTION FLUCTUATION MEASUREMENTS

Parul Tyagi, Indian Institute of Technology Delhi, India; Vivek Buwa, Indian Institute of Technology Delhi, India

Experimental characterization of three-phase slurry flow, especially under dense flow conditions that are relevant to industrial applications, has been a challenging problem. In the present work, we report measurements of instantaneous and time-averaged gas volume fraction and its distribution and bubble size distribution in a slurry bubble column using "in-house" developed dual-tip voidage probes. Gas volume fraction fluctuations were measured at superficial gas velocities in the range of 5-30 cm/s and for solid loading in the range of 5-40 vol. %. With increase in solid loading, overall and local gas volume fraction was found to decrease, accompanied by increase in bubble size. Measured local volume fraction time-series and the corresponding frequency analysis exhibited distinctly different dynamic characteristics: frequency of local fluctuations caused by bubble swarms was found to increase with increase of superficial gas velocity and the frequency was found to decrease with increase in solid loading (indicating formation of large bubbles/slugs). Dynamic characteristics inferred through gas volume fluctuations were analyzed using measurements of bubble size distribution at different solid loading. In addition to the new experimental data, the present work is expected to improve the understanding of dynamics of slurry flows that will aid the development and verification of computational models for their numerical simulations.

## AGGLOMERATION AND TRANSPORT OF A POPULATION OF OXIDE INCLUSIONS IN AN ALUMINUM INDUCTION FURNACE

Jean-Pierre Bellot, University of Lorraine, France; Akshay Bansal, University of Lorraine, France; Pierre Chapelle, University of Lorraine, France; Emmanuel Waz, Constellium, France; Yves Delannoy, Grenoble-INP, France; Pierre Lebrun, Constellium, France

Electromagnetic forces are widely used in liquid metal processing, in particular in the aluminum cast house, enabling melting of solid and stirring of the melt. In an induction furnace, the operating conditions result in an intense turbulent fluid flow within the crucible, raising a specific question with regard to inclusion dynamics. Indeed the behavior of the inclusion population is an issue of great concern for the metallurgists because of the strong effect of the particle size distribution on the mechanical properties of the Al alloys. A two-stage numerical model was developed based on the ANSYS Fluent software with several separate User Defined Functions. In the first stage, a 2D axially symmetric numerical model describing the coupled magnetohydrodynamic and free surface phenomena taking place in an induction furnace was set up. The calculation of the free surface shape is based on the Volume Of Fluid method and the model predictions were compared with measurements of the free surface shape obtained using a structured light projection technique. In the second stage, a Population Balance Model was adopted to simulate both transport of inclusions within the melt and interactions between inclusions (namely aggregation mechanisms). Simulations reveal a strong effect of the electromagnetic drift velocity on the inclusion trajectories.

## NUMERICAL AND EXPERIMENTAL STUDY OF TURBULENT FLOW IN A REAL-SCALE MODEL OF A CONTINUOUS CASTING MOLD

Hyun-Jin Cho, POSCO Technical Research Laboratories, Republic of Korea; Sang-Woo Han, POSCO Technical Research Laboratories, Republic of Korea; Youn-Bae Kang, POSTECH, Republic of Korea

Computational modeling of turbulent fluid flow has an important role to understand the structure of a flow and design the stabilized condition in the continuous casting mold. The flow patterns in a water model of a slab mold were investigated experimentally and numerically by means of several types of turbulence models and spatial discretization methods. The numerical simulations were made with a commercial CFD code Fluent. The experimental study was carried out by using a real-scale water model in order to measure the flow fields using Particle Image Velocimetry (PIV). The results show the k- $\epsilon$  model among two-equation turbulence models was validated with experimental results. It was also observed that the discretization in space for momentum, kinetic energy, dissipation rate has a major influence on the main flow.

## NUMERICAL OPTIMIZATION OF THE PHASE INTERFACE IN HIGH-PRESSURE DIE CASTING

Markus Frings, RWTH Aachen University, Germany; Marek Behr, RWTH Aachen University, Germany; Stefanie Elgeti, RWTH Aachen University, Germany

The high-pressure die casting process is the most common casting process for aluminum components. In this casting process, molten aluminum at high temperature is pushed into a cavity under high pressure. The formation of the phase interface between the molten metal and the air has a large impact on the overall cast quality. The most influencing factor of this interface shape is the so-called shot curve. The shot curve describes the movement of the plunger, which pushes the metal towards the cavity. Due to the high pressure and temperature, it is not convenient to design the shot curve experimentally. Especially, if it comes to industrial application, experiments are almost impossible. Therefore, in the presented work numerical optimization is used to find an appropriate shot curve. Hence, different objective functions are provided. Each of these objectives quantifies the flow behavior w.r.t. a specific process goal. The choice of the objectives is motivated by the investigation of the process and the results of the simulations. It is shown that the interface shape can be controlled by a small set of design variables. Furthermore, the approach can be easily embedded into existing industrial environments.

\* This work is supported by the German Research Foundation under the Cluster of Excellence "Integrative production technology for high-wage countries. Computing resources are provided by the AICES graduate school and the Jülich Aachen Research Alliance (JARA).

## SPREADING BEHAVIORS ON MOLTEN-METAL FALLING-FLOW AT ELEVATED TEMPERATURES IN NUCLEAR SEVERE ACCIDENT CONDITIONS

Michitsugu Mori, Hokkaido University, Japan; Tatsuki Matsumoto, Hokkaido University, Japan; Takahito Ogura, Hokkaido University, Japan; Shuichiro Miwa, Hokkaido University, Japan

This study aims at accurately observing spreading behaviors of molten-core falling-flow, which reproduces the leak flow into the containment vessel pedestals from the reactor pressure vessel in severe accidents. In Fukushima nuclear disaster, molten core would leak out and relocated on the pedestal floor of the containment vessel, where the deposition of the molten core solidified to be debris should be predicted to make decommissioning advance at Fukushima nuclear power plants. Although the experiments for core catcher designs found in earlier studies focused on observing the molten core of continuous deposition on a floor, we could find few experimental reports to observe the behaviors of molten core that spreads after falling and impinging on the floor of the containment. The present experiments allow to observe the behaviors of the molten metal falling flow simulating the deposition of the molten core on the floor, in which the data are accumulating to predict the spreading behaviors of the molten core to be debris. The experiments have also performed the debris formation in the bottom water layer by falling the molten metal into the water changing its depth. Furthermore, we have analyzed the retrospective experiment of molten core spreading flow of VULCANO VE-U7 experiment and the present experiments using CFD code. The bottom crusting of molten metal mainly involves the flow in stopping, and the contact temperature at the interface could be a primary factor to decide length of spreading flow of molten metal.

## STUDY ON THE REMOVAL OF LOW LOADING HEAVIER PHASE BY LIGHTER PHASE IN MULTIPHASE PIPE

Aminu Koguna, Cranfield University, UK; Liyun Lao, Cranfield University, UK; Hoi Yeung, Cranfield University, UK

Production fluids from the reservoir are transported in pipelines from the well heads to the platform and from the platform to process facilities. At low flow velocity water, sand or liquids like condensate could settle at the bottom of pipelines that could lead to grave implications for flow assurance. During shutdown the settled heavy liquid (e.g. water), could result in corrosion in pipelines, while following restart stages the settled water could form water plugs that could damage equipment, while settled sand could also form a blockage that needs to be purged. Observations of low water cut in oil and water flows and low sand concentrations in water and sand flows were conducted in four and two inch diameter pipelines. Conductive film thickness probes were used to ascertain structural velocities. Comparisons are made between two cases in order to gain better understanding of the dispersal process of low loading heavier phase in multiphase flows.

## HYDRODYNAMICS OF GAS ENTRAPMENT IN MICROCAVITIES

Miguel Gomez-Lopez, University of Kentucky, USA; Soroosh Torabi, University of Kentucky, USA; Christine Trinkle, University of Kentucky, USA; Jose Grana-Otero, University of Kentucky, USA

Boiling typically requires a superheated liquid in contact with a surface with discrete microscopic cavities. Bubbles grow on these microscopic defects, with sizes ranging from tens to hundredths of microns, and ultimately detach, leaving in the cavity a pocket of vapor that serves as initial nucleus for another bubble growth-detachment event. The dynamics of this vapor entrapment, still poorly understood, is a ubiquitous key aspect in boiling phenomena. Gravity effects being negligible due to the small sizes, the hydrodynamics of the vapor entrapment is mainly controlled by surface tension effects, which can be balanced either by inertia or by viscous effects, depending on the effective Reynolds number based on the cavity size. We have studied experimentally, by means of high-speed, high-resolution imaging, in a simplified flow configuration, the dynamics of vapor entrapment in micro-machined defects of controlled geometry on a wall of a Hele-Shaw cell, for Reynolds numbers ranging from small to moderately larger than unity. In particular, we have obtained the volume of entrapped gas, determined, aside from the geometry of the cavity, by the Reynolds number and by the liquid-substrate contact angle.

\* Support from NASA through award number NNX13AB12A is gratefully acknowledged.

## DYNAMICS OF VERTICAL FALLING FILMS – AN EXPERIMENTAL AND NUMERICAL STUDY

Anders Akesjö, Chalmers University of Technology, Sweden; Mathias Gourdon, Chalmers University of Technology, Sweden; Lennart Vamling, Chalmers University of Technology, Sweden; Srdjan Sasic, Chalmers University of Technology, Sweden

Vertical falling liquid films are found in a large number of process systems, e.g. in food and pulp & paper industry. The hydrodynamics of such systems are complex due to action of gravity and the existence of a free surface of the fluid, resulting in a variety of complex flow patterns, each with its own characteristic wave dynamics. The latter is of great importance to investigate, since profound understanding of the hydrodynamics of films has yet to be achieved.

In this study, we simulate falling films using Volume of Fluid (VoF) method. We explore wave formation and propagation for fluids with different viscosities and at different Reynolds numbers. One of the aims is to study the transition from sinusoidal to wavy-laminar flow regimes.

To validate the results of our simulations, we have performed optical laser triangulation measurements of the film thickness at several vertical positions for two different subcooled fluids and at various Reynolds numbers. The results conclude that both the Reynolds number and the Kapitza number are key parameters determining the wave characteristics of the flow. The results also show that the perturbation applied in the boundary conditions does not have a major effect on the outcome of the simulations.

## NONLINEAR FORCED WAVES IN A VERTICAL RIVULET FLOW

Sergey Alekseenko, Institute of Thermophysics SB RAS, Russian Federation; Sergey Aktershev, Kutateladze Institute of Thermophysics, Russian Federation; Dmitriy Markovich, Kutateladze Institute of Thermophysics, SB RAS, Russian Federation; Sergey Kharlamov, Kutateladze Institute of Thermophysics, Russian Federation; Aleksey Bobylev, Kutateladze Institute of Thermophysics, Russian Federation

Authors present the results of numerical simulation of three-dimensional waves on the surface of the rivulet flowing down a vertical plate. The Kapitza-Shkadov approach is used to describe the wave flow of the rivulet. Characteristics of linear and nonlinear regular waves in the rivulet were obtained in numerical calculations depending on the forcing frequency at different Reynolds numbers and contact wetting angles. The results of simulation are compared with the authors' previous experimental data. The comparison shows that the applied model adequately describes the shape of the wave surface of a rivulet, though gives underestimated values of the wave propagation velocity and wavelength.

## EULER-EULER MODELING OF HYDRODYNAMICS AND MASS-TRANSFER IN BUBBLY FLOWS

Roland Rzehak, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Sebastian Kriebitzsch, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Eckhard Krepper, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany

CFD simulations of dispersed bubbly flow on the scale of technical equipment are feasible within the Eulerian two-fluid framework of interpenetrating continua. However, accurate numerical predictions rely on suitable closure models. To achieve predictive capability all details of the closure models have to be fixed in advance without reference to any measured data.

Concerning the fluid dynamics of bubbly flows a baseline model has recently been proposed to this end and is shown to work for a range of different applications in a unified manner. This provides a reliable background which is well suited to add more complex physics. The approach will be explained and some illustrative examples shown.

Concerning mass transfer in bubbly flows only few studies have been performed to date and these use a variety of different closures for the mass transfer coefficient in rather similar situations. A review of the data and theoretical support for the different closure models reveals some gaps in the current understanding that need to be filled in order to obtain a robust description covering relevant conditions. A preliminary attempt at such a baseline closure for mass transfer is made here which may guide further research. Due to the remaining uncertainties in the modeling a first study on model validation is presented which makes use of experimentally determined mass transfer coefficients. The need for and requirements on suitable data for this purpose are emphasized.

## A SUBGRID MODEL FOR LARGE-EDDY SIMULATION OF TURBULENT BUBBLY FLOWS

Fathia Chaib Ina, National Engineering School of Tunis, Tunisia; Ghazi Bellakhal, National Engineering School of Tunis, Tunisia; Jamel Chahed, National Engineering School of Tunis, Tunisia

A subgrid-scale model for large-eddy simulation of turbulent bubbly flows is developed on the basis of the second order turbulence closure in bubbly flows developed by Chahed et al. (2003). The reduction of the second order turbulence modeling leads to a turbulent viscosity formulation for bubbly flow that points out two antagonist interfacial effects on the turbulence structure in bubbly flows: the bubbles agitation induces in one hand an enhancement of the turbulent viscosity and on the other hand a modification of the characteristic scale of the eddies stretching which can reduce the shear stress. On the basis of this physical analysis a subgrid scale (SGS) model for bubbly flows is proposed. In comparison with single phase flow, the SGS applied to bubbly flows appears with a multiplication factor that may increase or decrease the subgrid-scale eddy viscosity. This SGS model has been implemented in an Eulerian two-fluid large eddy simulation software and has been applied to the simulation of homogeneous turbulent bubbly flows with uniform velocities and with uniform shear. The results are in good agreement with experimental data of Lance et al. 1991. In particular, the numerical simulations allow reproducing the modification of the energy spectrum slope in homogeneous bubbly turbulence.

## DNS ASSISTED MODELING OF BUBBLY FLOWS

Gretar Tryggvason, The University of Notre Dame, USA; Jiakai Lu, University of Notre Dame, USA; Ming Ma, University of Notre Dame, USA

As direct numerical simulations (DNS) of multi-fluid and multi-phase systems have become relatively routine, finding new ways to extract knowledge, and reduced order models, from the enormous amount of data that the simulations produce, is emerging as a major challenge. DNS often yield new insights and sometimes such insight results in simplified models. However, the process usually relies on the experience of the investigator. Finding ways to extract information fast and reliably in as automatic way as possible would make DNS results much more useful. We review how DNS of bubbly flows in laminar vertical channels have resulted in new insights and models, and then explain how we are using statistical learning tools to extract closure relationships from the data, for a simple set of model equations for the average flow, resulting in more general predictive equations for the average flow. Simulations of more complex flows, such as the transient motion of polydisperse bubbly turbulent flow and flows undergoing topological transitions—where bubbles coalesce and break up—are presented and the challenges in conducting DNS and extracting closure relationships for such flows discussed. We conclude with a short analysis of the prospects of using a similar approach for more complex multiphysics problems.

\* Support from NSF grant CBET-1335913 and the Consortium for Advanced Simulation of Light Water Reactors, under U.S. DoE Contract No. DE-AC05-00OR22725, is gratefully acknowledged.

## STUDY OF AN INTERFACE SHARPENING METHOD FOR LARGE INTERFACE TRACKING WITHIN A TWO-FLUID MODEL

Solene Fleau, EDF R&D, France; Stephane Mimouni, EDF R&D, France; Stephane Vincent, Université Paris-Est, France

Bubbly flows occurring in nuclear power plants represent a crucial safety issue. However, the large range of bubble diameters and shapes experimentally observed in these complex flows have required the development of new computational approaches. Thus, in this article, a multifield method adapted to a two-fluid model is presented. The small spherical bubbles are followed with a dispersed approach whereas the large deformable ones are located with interface tracking methods. Nevertheless, a special treatment is required to locate precisely these interfaces and to evaluate accurately the local quantities, such as the normal vector and the curvature since the two-fluid model induces a numerical interface smearing. This paper is devoted to the implementation of an interface sharpening equation, which ensures mass conservation and does not affect the physical results. Therefore, the integration of this equation in the numerical scheme of the code is detailed. Criteria to limit the creation of spurious velocities and to recognize the diffused interfaces are also defined. Finally, different validation test cases are proposed including deformable large bubbles and free surfaces.

## TWO-PHASE HEAT TRANSFER AND FLUID FLOW WITHIN PLATE HEAT EXCHANGERS

Jungho Kim, University of Maryland, USA; Valentin Solotych, University of Maryland, USA; Lee Donghyeon, POSTECH, Republic of Korea; Raffaele Amalfi, École Polytechnique Fédérale de Lausanne, Switzerland; John Thome, École Polytechnique Fédérale de Lausanne, Switzerland

Infrared (IR) thermography was used to measure the local heat transfer coefficients with two plate heat exchanger geometries. The patterns were machined into polycarbonate and IR transparent calcium fluoride plates, both of which were electrically heated using flexible film heaters. The test fluid was HFE7100 at mass fluxes between 25 and 100 kg m<sup>-2</sup>s<sup>-1</sup>, and qualities from 0 to 0.9. The apparatus and data reduction technique were validated by comparing the single-phase heat transfer and pressure drop data against the prediction methods from the literature. Adiabatic flow visualizations were used to link the flow patterns with the observed heat transfer. The local heat transfer coefficient and the frictional pressure gradient increased with mass flux and vapor quality. The local frictional pressure gradient and heat transfer coefficient were compared with available correlations. The comparison indicated the need for new prediction methods for predicting the local thermal-hydraulic performance over a wide range of operating conditions.

## HEAT TRANSFER IN FLAT PLATE PULSATING HEAT PIPES WITH ASYMMETRIC CHANNEL DESIGNS

Dong Soo Jang, Republic of Korea University, Republic of Korea; Joo Seong Lee, Republic of Korea University, Republic of Korea; Dongwoo Kim, Republic of Korea University, Republic of Korea; Hanseok Mun, Republic of Korea University, Republic of Korea; Yongchan Kim, Republic of Korea University, Republic of Korea

This study investigates the heat transfer characteristics of flat plate pulsating heat pipes with asymmetric channels. There were three types of pulsating heat pipes with different channel depths of 0.4, 0.6, and 0.8 mm. Each consists of four different asymmetric ratio channel designs. The asymmetric ratios of channel cross sectional areas were from 1 to 4. The hydraulic diameters of channels were about from 0.5 to 1.5 mm. FC-72 was used as the working fluid due to its low viscosity and surface tension fluid characteristics. The filling ratio of the working fluid was 50%. The oscillating motion of the working fluid was observed in a visualization device, and its flow patterns were investigated. The effect of channel asymmetric ratio on the thermal performance was analyzed under various heat inputs. The results showed that the performance characteristics of the pulsating heat pipes with asymmetric channels varied with respect to working conditions. The asymmetric ratio and heat input showed significant influence on the performance of the pulsating heat pipes.

## EXPERIMENTAL INVESTIGATION OF TAYLOR BUBBLE DISSOLUTION IN MILLI-CHANNELS

Mohammadreza Haghnegahdar, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Stephan Boden, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Uwe Hampel, TUDresden, Germany

In the work presented in this paper, the dissolution rate of a single Taylor bubble of carbon dioxide into water was compared within a square and circular channel of 6 mm hydraulic diameter. The bubbles were held stationary in the down-flowing liquid and observed with high resolution X-ray radiography and tomography. The acquired X-ray images of the bubbles were analyzed with respect to volume, surface area and length of the bubble and were utilized to obtain the liquid side mass transfer coefficient by measuring the changes in the size of the bubble at constant pressure. The X-ray method was chosen since it is not dependent on the refractive index; therefore it is the most accurate method in comparison with other conventional optical techniques. The results for the long term dissolution of single CO<sub>2</sub> bubbles show that the dissolution curves for bubbles with different initial size follow the same trend and have relatively constant slope. In addition, the comparison of the results for the square and circular channels showed that, there are some distinguishable differences between the trend of the liquid side mass transfer coefficient, *k<sub>L</sub>*, in two curves. For square channel, *k<sub>L</sub>* increases with decrease of bubble sizes, while no obvious trend could be detected for the circular channel.

## MASS TRANSFER FROM SINGLE CARBON DIOXIDE BUBBLES IN WATER-ALCOHOL SOLUTIONS IN A VERTICAL PIPE

Yohei Hori, Kobe University, Japan; Jiro Aoki, Kobe University, Japan; Kosuke Hayashi, Kobe University, Japan; Shigeo Hosokawa, Kobe University, Japan; Akio Tomiyama, Kobe University, Japan

Mass transfer from single carbon dioxide bubbles rising through fully-contaminated water in a vertical pipe of 12.5 mm diameter is measured to investigate effects of surfactant properties. A wide range of bubble diameter, *d*, is tested to cover various bubble shapes such as ellipsoidal and Taylor bubbles. Five alcohols having different carbon chain lengths are used for surfactants, i.e., 1-pentanol, 1-heptanol, 1-octanol, 2-octanol and 1-decanol. Note that their adsorption-desorption properties are also different from each other. Experiments are performed at fully-contaminated conditions, for which the mass transfer rate, *k<sub>L</sub>*, does not change at all even by further increase of surfactant concentration. The *k<sub>L</sub>* of contaminated ellipsoidal bubbles are smaller than those of clean ellipsoidal bubbles. The reduction of *k<sub>L</sub>* becomes larger as the carbon chain length of alcohols increases. In other words, the surfactant properties affect mass transfer from fully-contaminated ellipsoidal bubbles. The *k<sub>L</sub>* of contaminated Taylor bubbles increase with *d* and take almost the same value for all the surfactants at large *d*. The Sherwood numbers of ellipsoidal bubbles in water-alcohol solutions are well correlated by introducing the ratio of the adsorption rate constant to the desorption rate constant.

## AEROSOL SCAVENGING BY DROPS: DERIVING A COLLECTION KERNEL FROM MICROPHYSICAL MODELLING

Gael Cherrier, INRS, France; Emmanuel Belut, INRS, France; Anne Taniere, Université de Lorraine - LEMTA, France; Nicolas Rimbert, University of Lorraine, France; Fabien Gerardin, INRS, France

Scavenging aerosol particles by droplets is an efficient way of removing solid particles from gases. Despite its interest for both industrial applications and occupational health, predicting realistic removal rates by means of a simple collection kernel is still a challenge (Belut (2014)). A realistic collection kernel must simultaneously take into account the aerosol dynamics and the liquid-gas interactions around a moving drop. Besides flow pattern, the capture of particles is also affected by heat transfer, phase change, electric forces, Brownian motion, and inertial effects that must be taken into account.

Examining the collection kernel of Wang et al. (1978) (low inertia particles undergoing Brownian, phoretic and electrostatic scavenging) or the collection kernel of Beard et al. (1971) (inertial capture only) shows the lack of universality of scavenging kernels but also the lack of numerical or experimental reference data allowing a more general scavenging kernel to be derived.

We propose to extend available reference data using numerical experiments. CFD simulations of particle-laden flows around and inside a drop at moderate Reynolds number are achieved, taking into account interfacial effects. The aerosol is modeled using Lagrangian stochastic tracking with large time steps. A global scavenging kernel is then derived from the collection efficiencies obtained in numerical simulations.

## ANISOTROPIC TURBULENT CLUSTERING OF CLOUD DROPLETS SETTLING IN ISOTROPIC TURBULENCE

Keigo Matsuda, Japan Agency for Marine-Earth Science and Technology, Japan; Ryo Onishi, Earth Simulator Center, Japan; Keiko Takahashi, Japan Agency for Marine-Earth Science and Technology, Japan

Recent studies have reported that microscale turbulent clustering of cloud droplets can be a cause of significant increase of the radar reflectivity factor. However, the influence of gravitational settling of cloud droplets is not clearly understood yet. The gravitational settling can be a cause of the anisotropic structure of turbulent clustering, which can be a cause of a radar beam direction dependency of the radar reflectivity factor enhancement due to turbulent clustering. Thus, this study aims to investigate the influence of gravitational droplet settling on the anisotropic structure of turbulent clustering by using a direct numerical simulation (DNS) of particle-laden isotropic turbulence, where the cloud droplets are tracked by the Lagrangian method. The influence of turbulent clustering on the radar reflectivity factor is evaluated by using the spectral density function of droplet number density fluctuation. The results show that the spectral density in vertical direction tends to be smaller than that in horizontal direction under conditions in cumulus clouds. This result indicates that the enhancement of radar reflectivity factor observed in horizontal direction is larger than that in vertical direction.

## INVESTIGATION OF IMPINGING DROPLET SPREADINGS ON HOT/COLD SURFACES WITH TR-PIV

Nejdet Erkan, The University of Tokyo, Japan; Koji Okamoto, The University of Tokyo, Japan

In this study we investigated dynamical differences between two marginally distinct regimes of water droplets and solid surface interactions, i.e., droplet impacts onto a very hot surface (300 °C) in film boiling regime (Leidenfrost regime) and impacts on a dry surface at room temperatures (cold surface) at low Weber numbers scaling around 4 and 10.

We visualized the impact process with high-speed shadowgraphy technique to investigate transient expansion of liquid lamella. Simultaneously, for the same droplet we performed spreading velocity measurements inside the liquid lamella with Particle Image Velocimetry (PIV). Full field 2D velocity vector maps on the horizontal cross-sectional plane were resolved. Utilizing vector maps, radial velocity profiles were obtained.

The results demonstrated that Leidenfrost droplets have lower spreading magnitudes compared to the impacts onto cold surface. Radial velocity profiles for both cases exhibit simultaneous linear and non-linear modes. The nonlinearity is caused by the vortical flows formed at outer regions of the spreading liquid lamella. We demonstrated that even at low impact velocities the linear parts of the profiles obey a quasi-one-dimensional theory proposed in the literature.

## FEEDBACK EFFECT IN GAS-SOLID DISPERSE FLOW THROUGH A VERTICAL CHANNEL

Yoichi Mito, Kitami Institute of Technology, Japan

Following the study on the correlation between particle dispersion and gas turbulence in a vertical channel, presented by Mito at ICMF 2013, the effect of inertia and gravity of solid particles on the feedback forces and the resultant mass transfer in gas-solid disperse flow through a vertical channel, where the gas is flowing turbulently, 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 through an infinitely-wide channel, of which the frictional Reynolds number is 150. The gas flow is driven downward with the constant pressure gradient. The feedback forces are calculated using a point force method. Small volume fractions, ranging from  $1 \times 10^{-4}$  to  $2 \times 10^{-3}$ , three inertial time constants and four Froude numbers, including infinity, are considered. The gas-phase momentum balance, that is integrated over the domain, is used in discussion. Drag decreases with increasing volume fraction and by the effect of gravity, due to increases in slip. Wall friction and turbophoresis decrease with increasing particle inertia, due to decreases in gas turbulence.

## A NEW LIQUID PHASE MODEL FOR DROPLET HEATING AND EVAPORATION BASED ON HILL'S VORTEX

Zhifu Zhou, Xi'an Jiaotong University, China; Weiyu Li, Xi'an Jiaotong University, China; Bin Chen, Xi'an Jiaotong University, China; Guo-Xiang Wang, University of Akron, USA

Droplet evaporation is of fundamental importance in extensive engineering applications. This study aims to build a comprehensive liquid phase model for droplet heating and evaporation, with the capacity of predicting the internal temperature distribution and evaporation rate reasonably. Meanwhile, its computational simplicity is comparable with the uniform temperature model (lump model), suitable for implementation into computational dynamics (CFD) codes. This model is based on the assumption that the droplet internal temperature follows the third-order polynomial distribution. Convective heat transfer on droplet surface and Hill's spherical vortex within droplet are taken into account to solve the coefficient of polynomial temperature distribution. The model prediction is compared with the experimental data from literature for both temperature and evaporation at large temperature difference between droplet and ambient gas. It's found that the predictive results by new model agrees reasonably with experimental data, and the new model has much better performance than other liquid phase models, i.e. lump model and liquid conduction model. It is of large potential to improve the simulation outcomes of CFD for sprays.

\* Support from grant (51406151, 51336006) is gratefully acknowledged.

## LAGRANGIAN SUBGRID MODELLING FOR THE RELATIVE SEPARATION OF TRACERS IN TURBULENCE

Federico Toschi, Eindhoven University of Technology, The Netherlands; Alessandra Sabina Lanotte, Italy; Irene M. Mazzitelli, University of Rome Tor Vergata, Italy

Turbulence has an important influence on the transport of small particulate matter like dust in the atmosphere, fuel droplets in combustion chambers or even small biological entities in marine environments. The relative separation of pairs and of small number of particles is strongly influenced by the multi-scale space- and time-correlations of turbulent velocity fluctuations. While in applications it is often necessary to model the smallest turbulent scales, this may have an important impact on particles' dynamics. We present a recently proposed subgrid Lagrangian model capable of reproducing the effect of unresolved eddies on single particle dynamics (absolute dispersion) as well as on the relative separation of tracers. The model is simple, computationally efficient and is capable to reproduce Richardson separation between pairs in a cloud of an arbitrary number of tracers.

## MACROSCOPIC CHARACTERISTICS OF HEAVY PARTICLE RESUSPENSION OBTAINED FROM DIRECT NUMERICAL SIMULATIONS OF PRESSURE DRIVEN CHANNEL FLOW

Stergios Yantsios, Aristotle University of Thessaloniki, Greece; Panagiotis Saliakellis, Aristotle University of Thessaloniki, Greece

Three-dimensional direct numerical simulations are presented of the flow of fluid containing heavy spherical particles in a channel. The simulations resolve the motion of individual particles and the details of the flow at the particle level. They are based on the finite element method and the Distributed Lagrange Multiplier/Fictitious Domain Method. The problem is characterized by four parameters, namely a Reynolds and an Archimedes number, a density ratio and a dimensionless particle size. The last three are fixed and the effects of the Reynolds number are studied over a range covering the onset of particle resuspension and the transition of flow to significant unsteadiness. Encouraging estimates for the onset of resuspension and the particle flux as a function of the excess Shields number are found. The results are analyzed in terms of the vertical distribution of horizontally averaged macroscopic quantities, obtained by applying integral momentum balances. On the basis of such balances, particle stresses and interfacial forces between the fluid and particle phases are estimated and compared to existing models, developed mainly for Stokes flow conditions. The qualitative agreement is encouraging but points to the need for taking into account several effects, notwithstanding those of finite particle Reynolds numbers. Provided that such information becomes available the set of equations derived from the suspension and particle phase momentum equations could describe the average equilibrium distribution and the flow of both phases.

# MIXTURE SIMULATIONS TO INVESTIGATE THE EFFECT OF SUSPENDED SEDIMENT ON VORTEX GENERATION OVER A RIPPLED SAND BED

A. Penko, Joseph Calantoni, US Naval Research Laboratory, USA

Three-dimensional vortex structures dominate the hydrodynamics of near-bed oscillatory flow over sand ripples and play a significant role in ripple migration rates and sediment transport. Consequently, suspended sediment in the water column causes turbulence modulation and affects the amount of turbulent kinetic energy. A three-dimensional fluid-sediment mixture model (SedMix3D) is used to examine the small-scale turbulent dynamics over sand ripples. The model simulates the fluid-sediment mixture as a continuum with sub-models for the bulk parameters of the mixture (diffusion, hindered settling velocity, and mixture rheology) in order to simulate the turbulent, three-dimensional, boundary layer flow over a dynamic sediment bed. Two simulations were examined: a fluid-sediment mixture with a dynamic rippled bed and a pure fluid (no suspended sediment) over a fixed rippled bed. The vortex formations, ejections, and the total turbulent kinetic energy were compared in the two simulations. The simulations illustrate a significant amount of variance in the turbulent structures in the cross-flow direction and in time. Additionally, the effects of the three-dimensionality of the vortices on sediment fluxes and transport rates are also examined.

# THE EFFECT OF SHEAR STRESS FORMULATION AND OF THE SHAPE FACTOR ON SLUG CAPTURING

Arianna Bonzanini, Università degli Studi di Brescia, Italy; Marco Ferrari, Università degli Studi di Brescia, Italy; Pietro Poesio, Università degli Studi di Brescia, Italy

In earlier works, we showed that a one-dimensional, hyperbolic, transient five equations two-fluid model is suitable to predict the formation, growth, and subsequent development of slugs in horizontal and near-horizontal flow. Statistical characteristics (slug velocity, length, and frequency) can be numerically predicted with results in good agreement with experimental and available data. As expected, in this model some approximated and simplified assumptions are adopted to close the problem. In our paper, we focus on two different aspects: the impact of the shear stress formulation on the results and the possibility to account for 2D effects by including the shape of the velocity profile by a proper shape factor.

Shear stresses are modeled using the modified two-fluid model (MTF), which improves the standard closure relations starting from the ones based on the two-plate model. The results obtained using the standard model and the MTF are compared for different cases. Then, the effect on the numerical results of the shape factor, the parameter that takes into account the shape of the velocity profile, is investigated. The shape factor is computed at each time step and at each computational cell using the two plate approximation. The effect of shape factor is discussed with several examples.

# TURBULENCE CONTRIBUTION IN MOMENTUM INTERFACIAL TRANSFER IN AN UPWARD LIQUID-LIQUID PIPE FLOW

Mariam Rezig, National Engineering School of Tunis, Tunisia; Ghazi Bellakhal, National Engineering School of Tunis, Tunisia; Jamel Chahed, National Engineering School of Tunis, Tunisia

In this study we rely on eulerian-eulerian two fluid model presented in Chahed et al (2003) to assess the roles of turbulence and interfacial forces in the phase distribution in an upward liquid-liquid pipe flow. The numerical results are compared to the experimental data of Lucas et al (2009) which concern the analysis of the phase distribution in a liquid-liquid upward flow of dispersed oil droplets (size 6 mm) in water in vertical cylindrical pipe with 80 mm diameter and 2.5 m length. The experimental data concern the radial profiles of the velocity and of local oil volume fraction distribution for different mean oil fraction values from 0.056 to 0.205. The comparison of the numerical results with the experimental data shows a good concordance. The numerical simulations carried out in the framework of a comprehensive analysis show the pertinence of the improvements proposed for the closures for turbulence and for interfacial transfer and emphasize the roles of the turbulent contribution of the added masse force and the drift velocity in the phase distribution phenomena. Furthermore the numerical analysis makes it possible to determine the role of the lift and wall forces in the near wall distribution of the dispersed phase.

# MEASUREMENT OF LIQUID DROPLETS IN ANNULAR-MIST FLOW USING SHADOWGRAPH TECHNIQUE

Samwel Andayi, Tokyo University of Marine Science and Technology, Japan; Yuchen Wang, Tokyo University of Marine Science and Technology, Japan; Tatsuya Hazuku, Tokyo University of Marine Science and Technology, Japan; Tomoji Takamasa, Tokyo University of Marine Science and Technology, Japan

For the study of annular two-phase flow it has been necessary to develop a measuring technique to obtain detailed liquid droplets information such as diameter and velocity as well as their spatial distributions with high resolution and no disturbance of the flow. In this study, simultaneous measurements of diameter and velocity of liquid droplets in annular two-phase flow by an image-processing based on shadowgraph is examined.

A viewing section for taking liquid droplet image in a test pipe was located over a liquid film extraction unit made of porous material. After removing the liquid film through the extraction unit, only droplets flowed up the viewing section. Shadowgraph images of droplets in the viewing section were taken by a video camera synchronized with a pulsed-diode laser light source. The diameters and velocities of the liquid droplets were measured by an image-processing of the captured images.

A preliminary test to check the accuracy of the present measuring technique was conducted using multi-solid particles with nominal diameters of 0.3, 0.5, 0.7 and 1.0 mm. A Fairly good agreement between the measured and nominal diameters was obtained. The relative deviations of the measured diameters to nominal diameters of 0.3, 0.5, 0.7 and 1.0 mm were 5.87, 2.52, 2.83 and 2.06 %, respectively.

# THE EFFECT OF PHYSICAL PROPERTIES ON CHURN FLOW IN A LARGER DIAMETER RISER

Garaev Damir, University of Nottingham, UK; Josep Escrig Escrig, University of Nottingham, UK; Barry James Azzopardi, University of Nottingham, UK; Buddhikha Hewakandamby, University of Nottingham, UK

Churn flow is a major flow pattern which occurs in the vertical risers employed in the offshore oil and gas industry. However, this is one flow pattern about which the least is known. Indeed, there have been publications in the past which claimed it did not exist. To examine the existence of the churn flow phenomenon, large diameter vertical pipe experiments have been carried out at the University of Nottingham multiphase flow lab. In this paper, we present for two gas-liquid pairs taken from experiments on a 127 mm diameter, 10 m tall riser. The fluids used are air-water and air-silicone oil. The latter has a surface tension of 0.02 N/m, a density of 900 and a viscosity~5 times that of water. Measurements have been made with a Wire Mesh Sensor base on conductance or capacitance for the water and oil respectively. Data gathered were analysed to determine the flow regimes and occurrences of structures. The characteristics of the flow for the two viscosities were compared to draw conclusions on the impact of viscosities. The transitional flows are explained in is paper.

# NUMERICAL SIMULATIONS OF A RISING DROP OR BUBBLE WITH SHAPE OSCILLATIONS IN THE PRESENCE OF SURFACTANTS

Antoine Piedfert, LGC-CNRS / University of Toulouse, France; Frederic Risso, IMFT- CNRS/Université of Toulouse, France; Olivier Masbernat, Tlemoine, France; Benjamin Lalanne, LGC - University of Toulouse, France

Understanding the effects of surfactants on interface dynamics is crucial for processes involving bubbles or droplets. We have developed a numerical tool able to compute multiphase flows with insoluble surfactants adsorbed at interfaces and investigated their effects on the dynamics of a rising drop or bubble. The numerical tool describes deformable interfaces using the level-set method on a fixed Cartesian grid. Surface tension is non-uniform and depends on the local concentration of surfactants. Its value is extended over a finite volume on both sides of the interface and computed using a transport equation. Surface tension gradients induce Marangoni effect, numerically treated as a jump condition on tangential stress using the ghost-fluid method.

Once validated, the code is used to investigate the case of a rising drop with adsorbed surfactants. The simulations show that surfactants are advected downwards, which generates steep concentration gradients. At steady state, a balance is reached between downward convective transport of surfactants and upward Marangoni flux. Surfactants change drastically shape oscillations, inducing higher damping rates in agreement with theoretical predictions of Lu and Apfel (JFM 222, 1991) and experiments of Abi Chebel et al. (JFM 702, 2012).

## LIFT FORCE ACTING ON SINGLE BUBBLES IN LINEAR SHEAR FLOWS

Shohei Aoyama, Kobe University, Japan; Iztok Zun, University of Ljubljana, Slovenia; Kosuke Hayashi, Kobe University, Japan; Shigeo Hosokawa, Kobe University, Japan; Akio Tomiyama, Kobe University, Japan

The lift coefficients,  $CL$ , of single bubbles in linear shear flows are measured to investigate effects of the bubble shape, liquid velocity gradient and fluid properties on  $CL$ . The gas and liquid phases are air and glycerol-water solutions, respectively. The Morton number,  $M$ , ranges from  $\log M = -6.6$  to  $-3.9$ . The test section is a vertical square channel and linear shear flows are formed by rotating a vertically-aligned stainless belt. The trajectories and shapes of bubbles are obtained by using two high-speed video cameras. The drag and lift coefficients are calculated by substituting them into the equation of bubble motion. Experimental data show that the lift coefficients of spherical bubbles at low bubble Reynolds numbers,  $Re$ , depend on the dimensionless shear rate  $Sr$  and  $Re$ . The  $CL$  model proposed by Legendre and Magnaudet (J. Fluid Mech., 1998) agrees with the data of the spherical bubbles. Although the effects of the Morton number on  $CL$  have been neglected in previous studies, the present data show clear dependency on the Morton number. The reversal of the sign of  $CL$  of ellipsoidal bubbles is confirmed to take place at  $Eo/E^{1.9} \sim 9$ , where  $Eo$  is the Eötvös number and  $E$  the aspect ratio.

## A PARALLEL FRONT-TRACKING ALGORITHM FOR THE SIMULATION OF RISING BUBBLES

Aristeu Silveira Neto, Federal University of Uberlândia, Brazil; Millena Martins Villar, Federal University of Uberlândia, Brazil; Alexandre Roma, University of Sao Paulo, Brazil; Ricardo Serfaty, Petrobras, CENPES, Cidade Universitaria, Brazil; Berend Van Wachem, Imperial College London, UK; Marcio Pivello, Federal University of Uberlândia, Brazil

This work presents a computational methodology for the parallel simulation of three-dimensional, two-phase flows, based on adaptive strategies for space discretization. The method is based on the Front-Tracking Method and the discretization of the Eulerian domain employs a Structured Adaptive Mesh Refinement strategy along with an implicit-explicit pressure correction scheme. Modelling of the Lagrangian interface was carried out with MFTL (Multiflow Triangulated Library), which provides the tools for handling the three-dimensional interface in a parallel environment. Among these tools, a fully parallel remeshing module was implemented in order to preserve the shape and volume of the interface during the simulation. Results from shear flow and spurious currents tests show that the method can accurately track complex shape geometries, as well as represent the presence of the interface while yielding low levels of artificial velocities due to spurious currents.

## EFFECT OF DEFORMABILITY ON BUBBLE MIGRATION IN TURBULENT GAS-LIQUID FLOWS

Alexander Vichansky, CD-adapco, UK

It is well-known that gas bubbles rising in a vertical pipe may peak either near walls or in the centre of the pipe. Size of the bubbles has been long identified as the parameter governing transition from the wall-peaking to centre-peaking. More recently it was shown that deformability, rather than size is responsible for the transition. In the present research we propose a model for wobbling-induced migration of the bubbles. Our starting point is instantaneous NS equation for two immiscible liquids. The interface between the liquids is captured by an indicator function. Averaging of the momentum balance together with the indicator function yields NS equations for two inter-penetrating continua and an equation for inter-phase momentum transfer. Deformability of the bubbles enters the force balance via correlation between liquid-phase stress and position/orientation of the interface, i.e., higher deformability implies higher correlation – and therefore – stronger wobbling-induced force. The force can be estimated energy balance in a homogeneous isotropic turbulence. The force is implemented in a two-phase CFD code.

## THE INTERACTION OF A HIGH-RE MODERATE-WE BUBBLE WITH AN INCLINED WALL

Christophe Barbosa, Universidad Nacional Autonoma de Mexico, Mexico; Dominique Legendre, Institut de Mécanique des Fluides de Toulouse, France; Roberto Zenit, Universidad Nacional Autonoma de Mexico, Mexico

In spite of the recent progress on the understanding of the dynamics of bubbly flows, the effect of boundaries is still not well studied. Yet, the interaction between the bubble phase and containing walls plays a significant role in practical applications. In this study, we analyze the steady motion of single high Reynolds, moderate Weber number bubbles interacting with an inclined wall. We conduct experiments considering with different liquids; bubbles are released in a tank to reach their terminal properties to then interact with an inclined wall. We observe the interaction for different wall for inclinations ranging from 5 (nearly horizontal) to 85 degrees (nearly vertical). We also conduct numerical simulations considering a finite volume/front-capturing method. Two distinct behaviors are observed. For all cases, the bubbles initially collide repeatedly with the wall. For small wall inclinations, after a brief period when the collision amplitude progressively decreases, the bubble continues to ascend sliding at a constant speed on the wall. Beyond a certain critical angle, the bubble continues to ascend but bounces repeatedly against the wall with constant amplitude. We analyze the two behaviors and argue about the physical origin for the transition, considering both drag and lift forces on the bubble.

## MEASUREMENT OF BUBBLE DISPERSION IN A TURBULENT BOUNDARY LAYER

Celine Gabillet, Institut de Recherche de l'Ecole Navale, France; Alexandre Bon, Institut de Recherche de l'Ecole Navale, France; Jean Yves Billard, Institut de Recherche de l'Ecole Navale, France; Sébastien Cazin, Institut de Mécanique des Fluides de Toulouse, France; Catherine Colin, INP Toulouse, France

In the context of drag reduction by bubble injection applied to ships hulls, bubbles' migration away from the wall is detrimental (Sanders et al., 2006). Despite numerical attempts to predict the bubble turbulent dispersion in the near wall region (Mattson and Mahesh, 2011, Ferrante and Elghobashi, 2004, Lu et al., 2005), experimental data are required. This paper presents the results of 3D bubble dispersion and trajectories measured in a turbulent boundary layer, spatially evolving under a flat plate. The Reynolds number  $Re_\tau$  was varied from 780 to 3000. Bubbles, injected at the wall, were between 30 to 250 wall units in diameter ( $db^+$ ). The Weber and Froude numbers (based on the friction velocity and bubble size) were in the range 0.02-0.27 and 0.3-1 respectively. A specific image processing, using visualizations of bubbles and reflected bubbles at the wall, was developed to track the bubbles in the 3D space. Bubble trajectories evidence alternative impact at the wall. The mean streamwise velocity, rms vertical and spanwise velocity components increase with respect to  $db^+$ . Opposite trend is observed for the rms streamwise velocity component. Scaling laws are investigated for two different external velocities and streamwise locations.

\* Support from grant ANR-12-ASTR-0017 F-DRAIHN and from French National Research Agency under grant ANR-12-ASTR-0017 F-DRAIHN is gratefully acknowledged.

## FLOW PAST A BUBBLE-GENERATING HYDROFOIL FOR SHIP DRAG REDUCTION

Ichiro Kumagai, Meisei University, Japan; Takahiro Tsukahara, Meisei University, Japan; Yuichi Murai, Hokkaido University, Japan

Development of ship drag reduction techniques is one of the urgent issues for ship builders to save costs, energy and environment. In our research group, a power-saving device for air bubble generation using a hydrofoil was invented; the device could effectively introduce small air bubbles around the ship hull and succeeded in reducing the net power consumption up to 15% in the full-scale tests (Kumagai et al., Ocean Engineering, 2015). Recently, in order to improve the performance of the bubble generation, a novel hydrofoil was invented. This hydrofoil has an air hole on its top side, which generates small air bubbles when the hydrofoil moves forward. In this presentation, we'll describe the device principle based on the simple fluid dynamic theory and show the recent results of towing tank experiments: flow behavior around the bubble-generating hydrofoil, threshold of bubble generation, void fraction, and the local skin friction under the ship hull. The void imaging data obtained by a high-speed video camera revealed that a large amount of air bubbles was introduced when the hydrofoil was installed at the normal position. However the local drag reduction rate for the inverted hydrofoil was better than that for the non-inverted hydrofoil because the inverted setting can effectively transport air bubbles to the ship bottom.



## INVESTIGATION OF THE BUBBLE SIZE DISTRIBUTION AROUND AN UNDERWATER VEHICLE

Sara Vahaji, RMIT University, Australia; Li Chen, Defence Science and Technology Organisation, Australia; Sherman Cheung, RMIT University, Australia; Jiyuan Tu, RMIT University, Australia

The interaction of the bubbles around underwater vehicles is of high interest for marine industries, especially in determining the appropriate mass and momentum transfer between two phases. In this regards, the prediction and evolution of the bubble size distribution around underwater vehicles are important considerations. In order to adequately capture the distribution and to account for its effect on the local hydrodynamics, a numerical assessment has been performed to investigate the bubbly two-phase flow around the DARPA Suboff submarine. The ensemble-averaged mass and momentum transport equations for continuous and dispersed phases are modeled within the two-fluid modeling framework. These equations are coupled with population balance equations (PBE) to aptly account for the coalescence and break-up of the bubbles. The influence of different gas injection rates and locations on the bubble size distribution is investigated. The results are validated against the experimental data, published by David Taylor Research Center, for single phase flow around the same geometry where reasonable agreement is found. Also, the same numerical approach is applied to a simpler geometry – bubbly flow over a flat plate. The results show promising capabilities of the model for the prediction of bubble size distribution.

\*Support from DSTO (Defense Science and Technology Organization) is gratefully acknowledged.

## STRATIFICATION OF INTERMITTENT AND ANNULAR R245-FA FLOW BOILING IN HORIZONTAL ORIENTATION

Thibaut Layssac, Centre d'énergétique et de thermique de Lyon CETHIL, France; Remi Revellin, INSA de Lyon, France; Stephane Lips, CETHIL, France

An optical method is developed for round mini-channels to estimate liquid film thicknesses and then characterize stratification of R-245fa horizontal flow boiling. For each condition of mass velocity, temperature and vapor quality, the flow in the 2.95 mm inner diameter channel is recorded with a speed of acquisition of 2000 frames/second during a total period of 2.7 s. The grayscale pictures are treated with an algorithm which determines liquid-vapor interface positions for annular and intermittent flows. The tests are thus performed for a large range of flow conditions. The saturation temperatures tested are 60, 70, 80, 90, 100°C and the mass velocities are 100, 200 and 300 kg.m<sup>-2</sup>.s<sup>-1</sup>. These parameters and vapor quality influence the flow inertia, buoyancy effects and superficial tension. The respective effects of these parameters are then discussed by the consideration of Froude and Bond numbers. It appears eccentricity decreases with an increasing Froude number, which corresponds to a higher effect of inertia above buoyancy, resulting of an increasing vapor quality or mass velocity. Eccentricity increases with Bond number, corresponding to a higher effect of buoyancy above superficial tension, appearing with an increasing temperature.

## A MULTISCALE APPROACH TOWARDS IN-TUBE CONVECTIVE BOILING FLOW SIMULATION

Miad Yazdani, United Technologies Research Center, USA; Abbas Alahyari, United Technologies Research Center, USA; Thomas Radcliff, United Technologies Research Center, USA; Marios Soteriou, United Technologies Research Center, USA

Convective boiling flow constitutes immensely separated scales of physical phenomena, making their numerical simulation prohibitive with current computational capabilities. The approach presented in this work bridges this scaling gap so as to provide a predictive model for thermal and hydrodynamic characteristics of internal convective boiling flow. The simulation approach consists of a high-fidelity model near the nucleating wall and a "medium-fidelity" LES-based model for the core. The high-fidelity model is based on our prior work where the macro-scales of flow dynamics are directly simulated and the nucleate boiling characteristics are simulated through combination of deterministic Computational Fluid Dynamics, stochastic Monte-Carlo, and asymptotic analytical formulation. The medium-fidelity model uses a WALE subgrid model internally to each phase and treats the region near the liquid-vapor interface using a modified version of the subgrid model proposed by Herrmann. The high and medium fidelity models communicate through an overlapping buffer region where the effect of nucleation is introduced into the large-scale convective flow. The composite model is capable of predicting flow regimes and their transition depending on operating conditions. In addition, the model can predict heat transfer characteristics under various flow regimes, albeit rather qualitatively in the vicinity of dry-out where film resolution requirements prohibit a quantitative simulation.

## VARIATIONS IN GEOMETRIC VARIABLES OF WATER DROPLET ON LOW-SURFACE-ENERGY SOLIDS

Yukihiro Yonemoto, Kumamoto University, Japan; Tomoaki Kunugi, Kyoto University, Japan

A droplet wetting behavior is very important for spray cooling and heat exchanger that involves drop-wise evaporation. In such applications, an accurate estimation of heat removal is essential. The evaporation process of the droplet is related to the wettability between the solid and liquid. But, the wetting behavior during the volume reduction process is not simple. When a droplet volume decreases, the droplet exhibits characteristic tendency such as hysteresis and size dependency of its contact angle. In the present study, we performed experimental and theoretical studies by focusing on these two characteristic behaviors. In the experiment, natural evaporation of micro and millimeter-sized water droplets on some low-surface-energy solids is observed. Then, the variations of geometrical parameters such as the droplet height, contact area radius and the contact angle are discussed using our new wettability model that was developed by considering the gravitational potential of the droplet, adhesion energy in the horizontal and vertical directions at the contact line. From the result, it is found that the size dependence of the contact angle can be predicted without the line tension concept.

## PREDICTION OF SOLID-LIQUID EQUILIBRIUM IN PARAFFINIC SYSTEMS WITH AN IMPROVED ACTIVITY COEFFICIENT METHOD FOR THE NON-IDEALITY OF SOLID PHASE

Juheng Yang, China University of Petroleum, Beijing, China; Wei Wang, China University of Petroleum Beijing, China; Pan Song, China University of Petroleum, Beijing, China; Qianli Ma, China University of Petroleum, Beijing, China; Qing Quan, China University of Petroleum, Beijing, China; Jing Gong, China University of Petroleum Beijing, China

A thermodynamic model for the prediction of wax precipitation with an improved activity coefficient method to describe the non-ideality of solid phase is established. For liquid phase, regular solution model and Flory free-volume equation are adopted to consider the two contributions of activity coefficient: enthalpy contribution, the energetic interactions between the components; entropy contribution, the differences in size and shape between the molecules respectively. For solid phase, a method to account for the two parts of the non-ideality is proposed. For the description of residual part (enthalpy contribution), an improved regular solution model is developed, based on the combination of regular solution theory and local composition theory; while Wilson equation with the consideration of an empirical correction factor  $\alpha_{ls}$  is used for combinatorial part (entropy contribution). The improved thermodynamic model is compared with experiment data from quaternary system and multi-paraffins systems. Results show that present model has a great capability for wax precipitation prediction of paraffin mixtures.

\* Support from grant the Key National Science and Technology Specific Project is gratefully acknowledged.

## THE EFFECTS OF NOSE SHAPES AND VISCOSITY ON THE CAVITY EVOLUTION AROUND THE UNDERWATER PROJECTILE

Kohei Okuno, Keio University, Japan; Akiko Matsuo, Keio University, Japan

The numerical simulation of the cavity evolution around the underwater projectile is performed using the computational fluid dynamics code that consists of the solid/gas two phases analysis of two-dimensional calculation method. When the cavitating bubble generated around the projectile covers the whole surface of projectile, it is expected to reduce the frictional resistance significantly. In this study, the effect of the fluid viscosity is focused, and the viscosity model is introduced to consider the dependence of the viscosity on temperature and pressure of fluid. It is found that the viscosity affects the fluid separation at the projectile nose where the most of cavity is generated, so that the viscosity has an important role on the cavity evolution. Furthermore, we conduct the calculation with the different nose shapes of projectile, and evaluate its influence on the cavity evolution. The cavitation radius and drag coefficient are calculated and compared with the experimental results. It is observed that the blunter the nose shape is, the larger the cavitation radius is. This trend is agreed with the experimental results. The reason is considered that the fluid flow around the blunter nose is bent more obliquely and the cavity spreads along the fluid flow.

#### LIQUID-FOUNTAIN INSTABILITY AND CAVITATION INSIDE BEADS-FOUNTAIN ASSOCIATED WITH ULTRASONIC ATOMIZATION

Kenta Myosen, DOSHISHA University, Japan; Yui Ito, DOSHISHA University, Japan; Yasushige Mori, DOSHISHA University, Japan; Katsumi Tsuchiya, DOSHISHA University, Japan

Ultrasonic atomization (USA) has been interpreted to be caused by parametric decay instability of capillary waves—occurring along a conical “liquid column/fountain”—and/or cavitation bubbles, if present, in symbiosis with capillary instabilities. This study attempts to unveil any possible relationship between USA-induced mist generation and liquid-column dynamics. The liquid-fountain dynamics and the co-occurring sonochemical luminescence are captured via high-speed and high-sensitivity imaging, respectively. Two types of droplets have been identified—depending on their energetic level of bursting—along with possible differences in generation mechanism. The bursting phenomena occur when the liquid column breaks in the specified region; the growth rate and breakup frequency are found to depend on the intensity of the ultrasound within the liquid fountain. While little direct (visual) evidence is available for the existence of cavitating bubbles within the liquid fountain, some distribution of cavitation has been detected. When the condition is properly adjusted, the fountain will take the form of a corrugated jet, or a chain of “beads” of submillimeter diameter in contact, steadily formed under low input power. The generation of mist, more specifically a swarm of liquid droplets of microscale or less, can be enhanced by a limited number—often singular events—of abruptly discharged droplet(s) of inception velocity in the range as high as 10–25 m/s; such a bursting droplet is found to be associated with a cavitation bubble generated within a bead constituting the fountain.

#### EXPERIMENTAL STUDY OF A CAVITATING BACKWARD FACING STEP FLOW: VAPOR PHASE INVESTIGATION USING X-RAY ATTENUATION MEASUREMENTS AND WALL PRESSURE MEASUREMENTS

Guillaume Maurice, LEGI, Grenoble University, France; Henda Djeridi, LEGI - Université de Grenoble, France; Stephane Barre, LEGI, Grenoble University, France

The purpose of the present experimental study is to get a better understanding of the dynamics of the vapour phase in a cavitating backward facing step flow and provide a refined data base for void ratio transport equation used for modelling cavitating turbulent flows. The backward facing step flow provides us a well-known test case to compare vortex dynamics, and a realistic industrial configuration such as backflow in turbo machinery. To highlight the vapour phase dynamics such as large vortex structures, free shear layer instability, reattachment wall interaction and reverse flow, the flow is diagnosed by X-ray attenuation techniques and by analysing the wall pressure fluctuations correlated with high speed visualizations and void fraction measurements. The two-phase structures were analyzed at different cavitation levels corresponding to 1% to 50% of void ratio range inside the shear layer, recirculation area and reattachment zone. The mean and fluctuating topology have been performed leading to three specific area in the flow such as vaporization, transport and condensation areas, while the high order statistical moments provided extreme events associated with high void ratio level and pressure wave propagation.

#### PARTICLE AND AGGLOMERATE DISPERSION IN GAS FLOW BY COLLISIONS AND FLOW INDUCED STRESSES

Tobias Wollborn, IWT - Foundation Institute of Materials Science, Germany; Claas Knoop, IWT - Foundation Institute of Materials Science, Germany; Udo Fritsching, Foundation Institute of Material Science, Univ. Bremen, Germany

Agglomeration and dispersion of solid particles play an important role in many industrial processes. An ultrasound agitated gaseous fluid is used to control the agglomeration process by dispersion. The dispersion is performed in direct and indirect ways. In the direct way agglomerate dispersion occurs by the induced stress of the agitated fluid on the primary particles of the agglomerate. In the indirect way agglomerates are accelerated in the agitated fluid. Breakage is caused by agglomerate collisions. For a better understanding of the breakage process two different simulation methods are used: CFD simulation to analyze the induced stress of the agitated fluid on the primary particles and DEM simulation to investigate the breakage behavior of the agglomerates. The induced stress of the agitated fluid is investigated by simulating a standing ultrasound wave and measuring the drag force of the agitated fluid on the primary particle surface. Stress conditions inside the agglomerate are evaluated by comparing the different force vectors of the single particles. The influence of the agitated fluid is investigated by varying frequency and amplitude of the ultrasound wave. Breakage behavior of the agglomerates in the indirect dispersion is carried out by simulating agglomerate collisions. Agglomerate properties are set by using the Hertz-model for the elastic-plastic behavior of the primary particles and a Stiff-bond-model for the bondings between the particles. The project is embedded into the SPP 1486 PiKo “Partikeln in Kontakt” which is funded by the German Research Foundation (DFG).

#### INFLUENCE OF SORPTION EFFECTS ON THE EFFICIENCY OF MULTIPHASE CONVEYANCE

Stefan Wagner, University of Kassel, Germany; Andrea Luke, University of Kassel, Germany

Multiphase conveying systems are increasingly used to improve the efficiency in the process industry and in oil and gas industry, for example, the gasses has not be flared anymore by conveying of crude oil over long distances. Multiphase pumps are installed to overcome the pressure losses during the conveyance. The multiphase flow entering into the pump is decreased by gap flows. Degassing phenomena within the first chamber is caused by the depression and the rotational and shear forces between the screws and the saturated liquid. The degassing effects is enhanced by cavities trapped with gas in the solid surfaces evoked by abrasion and corrosion. The gas is solved again into the liquid during the pressure build-up. To investigate these sorption and desorption effects, a model of the conveying chamber of the multiphase pump is designed. The screws are represented by rotating discs with similar surface microstructure. The pressure gradients as function of rotation speed, thermo-physical properties of the saturated fluid and the microstructure is analyzed theoretically and experimentally. The examined fluid system is paraffinic white oil saturated with air and carbon-dioxide, resp.

\* Supported by Federal Ministry of Economics and Technology on the basis of a decision by the German Bundestag

#### DROPLET-PARTICLE COLLISIONS IN HOMOGENEOUS AND ISOTROPIC TURBULENCE

Georgios Charalampous, Imperial College London, UK; Yannis Hardalupas, Imperial College London, UK

Collisions of droplets with solid particles in a turbulent carrier phase occur in many environments including spray dryers. Multiple outcomes are possible from such collisions depending on the relative droplet-particle size and velocity, the angle of collision and the carrier phase turbulence. Such information is important for understanding as well as for developing collision kernels for CFD simulations. Presently droplet-particle collisions have only been considered experimentally in quiescent environments while the contribution of the carrier phase turbulence has not been quantified. Here, an experimental investigation is conducted where the collision of a droplet with a solid spherical particle is examined in quiescent and turbulent gas environments. Mono-sized liquid droplets with diameters between 170-280 microns and stationary targets in the form of spherical particles with diameters between 500-2000 microns are considered with relative droplet particle velocities between 6-11m/s. The surrounding homogeneous and isotropic turbulence is generated by a purpose-built ‘box of turbulence’ facility, which generates variable levels of turbulence with zero mean velocity. The current investigations were performed up to turbulent Reynolds numbers of 200. The results quantify the influence of the gas phase turbulence on (a) the trajectories of the liquid droplets and the droplet statistical dispersion characteristics; (b) the outcomes of droplet-particle collisions for different relative droplet-particle sizes and velocities and the corresponding probabilities of collisions and outcomes.

#### THE PARTICLE COLLISION FREQUENCY IN PARTICLE-LADEN TURBULENT CHANNEL FLOW

Hans Kuerten, Eindhoven University of Technology, The Netherlands; Bert Vreman, AKZO Nobel, The Netherlands

Direct numerical simulation of particle-laden turbulent channel flow at relatively high Reynolds number ( $Re_\tau=950$ , based on the friction velocity and half the channel height) with a large number of inertial particles is applied to determine the particle concentration field and the particle collision frequency. Even for values of the overall particle volume fraction lower than 0.01%, particle collisions have a significant effect on the wall-normal particle concentration profile due to turbophoresis and preferential concentration, which together lead to large local particle concentration, if no four-way coupling is applied. The theoretical expression for the collision frequency contains the radial distribution function of the particles at contact. Determination of this quantity from the DNS results is not trivial, since the radial distribution function shows singular behavior in the central region of the channel for values of the particle distance tending to the particle diameter. This is caused by the occurrence of repeated collisions of particle pairs in short time intervals. The repeated collisions occur when two particles collide with low relative velocity and the fluid velocity field pushes them together again after a collision.

# THE INFLUENCE OF STOKES NUMBER ON PARTICLE CLUSTERING AT THE EXIT OF A TURBULENT JET ISSUING FROM A LONG PIPE

Timothy Lau, Centre for Energy Technology, Australia; Graham Nathan, Centre for Energy Technology, Australia

The naturally occurring phenomenon of particle clustering is of keen interest due to its significant impact on many two-phase flows, most notably, in concentrated solar thermal and solid fuel combustion systems. This study aims to investigate the influence of particle Stokes number on the clustering of particles in a turbulent shear flow. The experiments consisted of simultaneous measurements of particle velocity and concentration at the exit of a turbulent jet issuing from a long, round pipe using particle image velocimetry and planar nephelometry. Utilising mono-disperse solid particles of diameter  $d_p=10, 20$  and  $40\mu\text{m}$ , the Stokes number based on the pipe diameter,  $Sk_0$ , was varied across two orders of magnitude between  $0.3 \leq Sk_0 \leq 22.4$ . The resultant Reynolds number was in the range  $10,000 \leq Re \leq 40,000$ . The degree of clustering was measured utilizing the widely-used box counting method, as well as an in-house developed cluster determination scheme. The results show that particle clustering in the turbulent jet is significantly influenced by the particle Stokes number for  $Sk_0 \leq 5.6$ . It was found that the degree of clustering increases significantly as the Stokes number is decreased. Furthermore, it was found that as the Stokes number is increased, the width of the clusters increases.

\* Support from the Australian Research Council (Grant No. DP120102961) and the Australian Renewable Energy Agency (Grant No. USO034) is gratefully acknowledged.

# AN ACCURATE METHOD TO TAKE INTO ACCOUNT LUBRICATION FORCES IN NUMERICAL SIMULATION OF SUSPENSIONS.

Aline Lefebvre Lepot, CNRS, Ecole Polytechnique, universit  Paris-Saclay, France; Guillaume Chivot, Universit  Paris-Sud, France; Mathieu Girardin, Ecole Polytechnique, CNRS, Universit  Paris Saclay, France; Loic Gouarin, CNRS, France; Benoit Merlet, Universit  de Lille 1, France

We are interested in numerical simulation of three-dimensional zero Reynolds number macroscopic suspensions of rigid particles suspended in a Newtonian fluid. In such simulations, the well-known difficulty is to take into account the singular lubrication forces exerted between close particles. This problem is usually addressed using the well-established Stokesian Dynamics correction. It is based on an explicit expansion of the total lubrication forces exerted on pair of particles and on a pairwise additivity hypothesis. In this contribution, we use a new method to compute accurate solutions without introducing any model or hypothesis. It is based on an explicit expansion of the velocity and pressure lubrication fields. Doing so, short range interactions are propagated to the whole flow, including many-body lubrication effects. Moreover, the velocity and pressure fields are corrected, which allows us to compute accurately macroscopic quantities such as grid (pore) pressure, and to study the contribution of many body interactions to the suspension non-Newtonian properties. Another interesting feature of the method is that, since no tabulation is needed, it can be extended without any additional work to polydisperse suspensions.

\* This work has been supported by the Labex LMH (government program: ANR-11-IDEX-003-02 and ANR-11-LABX-0056-LMH) and the Labex PALM (LabEx PALM- ANR-10-LABX-0039)

# NUMERICAL SIMULATIONS OF VISCOUS AND COMPRESSIBLE TWO-PHASE FLOWS USING HIGH-ORDER SCHEMES

Marion Capuano, Ecole Centrale de Lyon, France; Christophe Bogey, Ecole Centrale de Lyon, France; Peter Spelt, Laboratoire de M canique des Fluides et d'Acoustique, France

A computational method for compressible viscous two-phase flows is presented. The simulations are performed by solving the compressible Navier-Stokes equations combined with two convection equations governing the evolution of the interface between the two fluids. The equations are discretized using high-order explicit centered finite differences. A shock capturing methodology based on a shock sensor and a spatial filtering is also implemented in order to capture various discontinuities, such as shock waves, expansion waves, contact discontinuities and interfaces. For time advancement a six-step Runge-Kutta algorithm is used. The numerical methods are all conservative and provide low dissipation and dispersion errors. One-dimensional test cases have been computed in order to assess the accuracy of the algorithm. Results of three two-dimensional cases are also considered. They are compared with experimental and analytical solutions. The first case concerns the Richtmyer-Meshkov instability generated on a post-shocked interface between air and SF6 gas. The second one is for a helium bubble hit by a shock wave traveling through air. Finally, numerical results are compared against the solution of a 2D Rayleigh-Plesset solution of an oscillating air bubble in water. The results obtained for the test cases show that the method is reliable and that it captures sharp discontinuities, whilst accounting for viscous effects.

# THE SPARSE CARDINAL SINE DECOMPOSITION APPLIED TO STOKES INTEGRAL EQUATIONS

Francois Alouges, Ecole Polytechnique, CNRS, universit  Paris-Saclay, France; Matthieu Aussal, Ecole Polytechnique, France; Aline Lefebvre Lepot, CNRS, Ecole Polytechnique, universit  Paris-Saclay, France; Franck Pigeonneau, UMR 125 CNRS/Saint-Gobain, France; Antoine Sellier, Ecole Polytechnique, CNRS, universit  Paris-Saclay, France

We are interested in numerical simulation of fully three-dimensional suspensions or emulsions involving rigid particles or bubbles suspended in a Newtonian liquid in the creeping flow regime. In such simulations, taking into account a high number of particles in order to predict macroscopic properties is a tremendously-involved problem. It can be efficiently tackled using Boundary Element Methods which are well-established and very accurate but lead to fully-populated linear systems and therefore to very expensive computations, both in terms of memory and CPU-time. Fast algorithms such as the Fast Multipole Method (FMM) have been developed to circumvent these drawbacks and now allow one to deal with problems involving tens of thousands of unknowns. In this contribution, we present a new fast algorithm called Sine Cardinal Sparse Decomposition (SCSD). Originally developed for wave equations (e.g. Helmholtz equation) where it has been found to outperform the FMM, this method is based on a suitable decomposition in the Fourier space of the underlying operators and on a subsequent careful approximation of the kernels. We here show and illustrate how the method is extended to the Stokes solver. We also compare the new method with existing FMM implementations.

# NON-CONSERVATIVE PRESSURE-BASED COMPRESSIBLE FORMULATION FOR MULTIPHASE FLOWS WITH HEAT AND MASS TRANSFER

Mathieu Labois, ASCOMP AG, Switzerland; Chidambaram Narayanan, ASCOMP AG, Switzerland

A pressure-based compressible multiphase flow solver has been developed based on non-conservative discretization of the mixture continuity equation. The formulation is an extension of the single phase incompressible pressure-correction approach, such that it can be applied to compressible single phase flows, two-phase flows using interface resolving methods (level set or volume of fluid) and general n-phase mixture flows using ensemble-averaged methods. The formulation is currently presented with the single pressure and single temperature assumption, but extension to multiple temperatures is straightforward. A robust treatment of phase change allows the method to model conditions with rapid phase change such as expansion through nozzles and valves. The method has been validated thoroughly using canonical single phase problems such as the shock tube, tank filling and sudden valve closure problems. Multiphase flow validation has been carried out for sound propagation in mixtures using the ensemble-averaged model and pressure wave transmission and reflection across an air-water interface, using the level set interface tracking method. The method has been used to study sound propagation in saturated steam-water systems under thermodynamic nonequilibrium, where the expected drastic reduction in the speed of sound is reproduced. Finally the method is applied to the problem of high-speed steam jet condensation in a subcooled pool of water

# COMPARISON OF THREE INJECTOR GEOMETRIES IN GAS LIFT PERFORMANCE IN A LARGE DIAMETER PIPE

Abubakr Ibrahim, The University of Nottingham, UK; Buddhikha Hewakandamby, University of Nottingham, UK; Barry James Azzopardi, University of Nottingham, UK

As easily exploitable oil resources are fast depleting, uneconomical deposits have become commercially viable. These resources have high viscous oils making the production procedures challenging. Gas-lift is one of such production techniques that has been applied over several decades. However, the understanding of the mechanisms at play is not complete due to lack of information, especially in large diameter pipes. In an attempt to fill this gap, an experimental study was conducted on gas injection method's influence on gas-lift performance in a 127mm pipe in a natural recirculation loop. Three injector geometries were used and the void fraction data was captured at four axial positions using 5cP silicone oil. Resulting flow regimes were identified, effect of the injection method on flow development and efficiency of gas-lift was investigated at two different submergence ratios. A coupled slug flow phenomena within churn flow was observed with mini slugs of frequency increasing with increasing gas superficial velocity and a constant frequency big slug waves that constitute of a train of micro slugs with higher average liquid hold-up.

Published gas-lift models were compared to the experiments for predicting the lift performance curve. The paper will report the findings in detail.

## COMPARISON AMONG EXPERIMENTAL TECHNIQUES FOR PARTICLE SIZE MEASUREMENTS IN FLOWS

Giampaolo Romano, University La Sapienza, Roma, Italy; Francisco Alves Pereira, CNR - INSEAN, Italy; Martijn Van Rijsbergen, MARIN, The Netherlands; Fabio Di Felice, CNR-INSEAN, Italy

The measurement of variables of interest in multiphase flows is rather complex due to the overlapping of particles-droplets-bubbles in the test region. The problem is even much more complicated for measurements using non-intrusive optical techniques, in which sometimes tracer particles must be added to the fluid phase.

Specifically, even in turbulent flows, the most relevant problem is not strictly to measure accurately the velocity field of each phase, rather to determine with small errors the size of single particles (droplets, bubbles). The correct size determination is also highly required in phase discrimination procedures. Therefore, it is mandatory to carefully evaluate errors in size measurement techniques. To this end, the present investigation aims to compare results obtained on measuring particle size in a dedicated apparatus, i.e. a completely transparent tank filled with water (composition and temperature are controlled and changed), in which cathode and anode wires generated oxygen and hydrogen bubbles (the electric variables as also the wire diameter and separation are also controlled and changed). At least three different techniques are used and compared for different parameter configurations: Shadow Imaging, Interferometric Particle Size and Particle Doppler Sizing.

\* Support from HTA Community of Practice on Nuclei Measurements is gratefully acknowledged.

## THIN FILM THICKNESS MEASUREMENTS USING ULTRASONIC PULSE ECHO TECHNIQUE

Yousuf Alaoui, University of Nottingham, UK; Nicholas Watson, University of Nottingham, UK; Barry James Azzopardi, University of Nottingham, UK; Buddhikha Hewakandamby, University of Nottingham, UK; Georgios Dimitrakis, The University of Nottingham, UK; Abbas Hasan, University of Nottingham, UK; Alexander Kalashnikov, University of Nottingham, UK

There is still strong interest in the characteristics of liquid films of both conducting and non-conductive liquids. These arise from the electric power generation and oil/gas production industries. These requirements make the ultrasonic technique very attractive as it does not depend on the electrical characteristics of the liquid. This paper presents the development of an ultrasonic technique using a 5MHz transducer for measuring wavy thin film thickness of <6 mm in a vertical pipe. Initial benchtop experiments were performed and different signal processing techniques were developed. For film thicknesses <0.5 mm a frequency domain technique was developed and for film thickness >0.5 mm a time of flight method was utilized. These methods were capable of measuring the wavy film thickness at inclination angles, of the reflecting interface, of <4° and validated using a theoretical volume measurement and optical techniques. The developed ultrasonic technique was then tested on a falling film rig with a pipe diameter of 127mm and different flow rate between 14.8 and 40litres/min. The results from the ultrasonic measurements showed good agreement with a Multi Pin Film Sensor (MPFS) and highlight the potential of the technique.

## A COMBINED EXPERIMENTAL/COMPUTATIONAL STUDY OF THE HYDRODYNAMIC CHARACTERISTICS OF HARMONICALLY EXCITED THIN-FILM FLOWS

Alexandros Charogiannis, Imperial College London, UK; Fabian Denner, Imperial College London, UK; Berend Van Wachem, Imperial College London, UK; Serafim Kalliadasis, Imperial College London, UK; Christos N. Markides, Imperial College London, UK

We present a simultaneous application of planar laser-induced fluorescence imaging (PLIF) and particle tracking velocimetry (PTV), complemented by direct numerical simulations (DNS) and aimed at the detailed characterization of the hydrodynamic characteristics of harmonically excited, thin-film flows. The experimental campaign comprises three aqueous-glycerol solutions spanning the range  $Re \approx 10 - 320$ . PLIF was employed in order to recover spatiotemporally resolved film-height data, and PTV in order to generate 2-D velocity-vector maps of the flow-field underneath the wavy interface. In that way, velocity and flow rate data could be evaluated at any location along the film topology. Mean flow rates were decomposed into steady terms (product of the mean film-height and bulk-velocity), and the covariance of the film-height and bulk-velocity fluctuations (unsteady terms). Steady terms were found to vary linearly with the flow  $Re$ , while unsteady terms scaled linearly with the film-height variance. The instantaneous and local flow-rate varied linearly with the instantaneous and local film height, with both experimental and numerical data obeying a simple analytical relationship, which allows for spatiotemporally resolved flow-rate data to be obtained without explicit knowledge of the flow-field underneath the wavy interface.

\* Support from EPSRC grants EP/J006041/1, EP/K008595/1 and EP/M021556/1 is gratefully acknowledged.

## MULTI-CAMERA X-RAY TOMOGRAPHY OF MULTIPHASE FLOWS

Olaf Skjaeraasen, Institute for Energy Technology, Norway; Heidi Denis, University of Oslo, Norway; Morten Langsholt, Institute for Energy Technology, Norway

We present X-ray measurements of 2-phase gas-liquid and 3-phase gas-liquid-liquid flow systems in a near-horizontal 98 mm (inner diameter) pipe. The processed data include time-evolving holdup profiles obtained via single-camera imaging from various azimuthal angles, as well as cross-sectional phase distributions obtained via X-ray tomography. By combining images obtained from up to six different viewing angles, both flow structures and statistical flow properties can be obtained, shedding new light on; e.g., gas and droplet entrainment processes in slug flow.

The X-ray system which was used has a cathode voltage of 60 keV and an imaging rate of up to 300 frames per second. It involves 1-6 cameras (each with its own X-ray point source) for monitoring 2-phase or 3-phase systems. For statistically steady flows, a very high signal-to-noise level can be reached by averaging over a large ensemble of images.

The paper includes a discussion of the principal and achieved accuracies of X-ray tomography with 2-6 projection angles. It is demonstrated that even with a modest number of projections, detailed physical insight can be obtained. Results are presented for slug flow and semi-annular flow involving purified SF<sub>6</sub> gas, Exxol D60 (viscosity 1.5 mPa s) or Primol 352 (viscosity 150 mPa s) oil, and purified water, with various gas and liquid flow rates at moderate pressures.

## NUMERICAL AND EXPERIMENTAL INVESTIGATION OF LIQUID FILM FLOWS ON PACKINGS IN ABSORBERS FOR POST-COMBUSTION CO<sub>2</sub> CAPTURE

Yoshiyuki Iso, IHI Corporation, Japan; Ryosuke Ikeda, IHI Corporation, Japan; Jian Huang, IHI Corporation, Japan; Mariko Saga, IHI Corporation, Japan; Shiko Nakamura, IHI Corporation, Japan; Kenji Takano, IHI Corporation, Japan

It is important to develop Carbon Capture and Storage (CCS) technology for coal fired power plants in order to reduce greenhouse gas emissions. Post-Combustion CO<sub>2</sub> Capture (PCC) is one of the economical and efficient technologies for reducing CO<sub>2</sub> emissions. Gas-liquid interfacial flows, such as the flue gas and liquid solvent, are applied in CO<sub>2</sub> absorbers for PCC. Efficient control of liquid flows by using packings can increase the gas-liquid interfacial area and mass transfer rate.

We investigate that the effects of surface texture treatments on the interfacial flow using a Computational Fluid Dynamics and an experiment. The results show that the surface texture treatments can help to prevent liquid channeling and can increase the wetted area. Furthermore, we develop our advanced design of surface texture treatments and packing geometry. The absorption column tests using the simulated solvent (NaOH) as well as the actual amine solvent are carried out to measure the performance of absorption rate and gas pressure drop. Our advanced packing has approximately twice the absorption performance comparing to a conventional structured packing. The gas pressure drop of our advanced packing is decreased approximately by half comparing to a conventional one at the same absorption amount.

## REDUCED ORDER MODELLING OF LIQUID-GAS CHANNEL FLOWS

Gianluca Lavallo, The University of Edinburgh, UK; Mathieu Lucquiaud, The University of Edinburgh, UK; Prashant Valluri, The University of Edinburgh, UK

Interfacial instabilities of counter-current liquid-gas flows have a crucial importance in several industrial prototypes, such as the absorption units for carbon-capture applications.

The direct numerical simulation (DNS) of those flows requires high computational cost and parallel computation. We thus present an alternative approach consisting of a reduced order model for the liquid layer coupled to Navier-Stokes equations in the gas phase.

Using this strategy, we study counter-current two-layer channel flows with moderate and high density ratios, and focus on the flooding phenomenon, which is still an unsolved problem in chemical applications. Speed and growth rate of linear waves match with the Orr-Sommerfeld theory and our Level-Set DNS, and non-linear wave profiles agree with DNS, as pressure and velocity fields also do. Finally, our model is tested with complex gas velocity profiles of cross-flow absorbers, and with simple mass diffusion models across the interface.

\* Support from EPSRC grant No. EP/M001482/1 is gratefully acknowledged.

## GROWTH OF A LOCALIZED PERTURBATION IN A COUNTERCURRENT TWO-PHASE FLOW

Gennaro Coppola, *Università di Napoli Federico II, Italy*; Francesco Capuano, *Centro Italiano Ricerche Aerospaziali, Italy*; Luigi De Luca, *Università di Napoli Federico II, Italy*

The work is concerned with the nonlinear evolution of a localized perturbation at the interface between two immiscible fluids of different density. In contrast to the more standard periodic disturbance, which evolves into the familiar train of Kelvin-Helmholtz linear waves, an isolated bump on the interface, treated as a vortex sheet discontinuity surface, leads to the emergence of a single travelling wave whose main characteristics are investigated.

A physical model is developed to predict the essential features of the single wave within the early instants of its evolution. The model relies on the Bernoulli equation and small perturbation aerodynamic theory. The height of the wave is found to scale quadratically both with time and with the velocity difference between the two fluids. A simplified functional dependence with the density ratio is also proposed. The model predictions are verified by means of numerical simulations. The two-phase interface evolution is computed by a point-vortex method that accounts for density stratification, gravity and surface tension. The numerical results confirm the existence of a regime in which an algebraic growth of the disturbance prevails. As the wave grows and its height becomes comparable with its basis, a self-similar regime establishes and the growth rate switches from quadratic to linear time trend. The results arising from the model are corroborated also by Navier-Stokes simulations of merging sheared two-current flows.

## MASSIVE PARALLEL DIRECT NUMERICAL SIMULATIONS OF THREE-DIMENSIONAL JET FLOWS

Lyes Kahouadji, *Imperial College London, UK*; Omar K. Matar, *Imperial College London, UK*; Jalel Chergui, *LIMSI-CNRS, France*; Damir Juric, *LIMSI-CNRS, France*; Seungwon Shin, *Hongik University, Republic of Korea*

We present three-dimensional simulations of two types of jet flow atomization. The first consists of a liquid jet surrounded by a fast coaxial flow of air for which the liquid experiences primary axial waves, secondary azimuthal waves, ligament formation and finally breakup phenomena. The second jet flow consists of a global rotational motion for which the liquid shows in the beginning a clear preference for helical instabilities. We use 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 on the IBM BlueGene/Q machine with excellent scalability performance. Coriolis forces are taken into account and solved via an exact time-integration method that ensures numerical accuracy and stability. The parallelization is based on an algebraic domain decomposition where the velocity field is solved by a parallel GMRES method and the pressure by a parallel multigrid method and communication is handled by MPI message passing procedures. The method for the treatment of the fluid interfaces uses a hybrid Front Tracking/Level Set technique that defines the interface both by a discontinuous density field as well as by a local triangular Lagrangian mesh.

\*Support from the Engineering & Physical Sciences Research Council, UK, through the MEMPHIS program grant is gratefully acknowledged.

## DIRECT NUMERICAL SIMULATION OF PRIMARY BREAK-UP IN SWIRLING LIQUID JETS

Claudio Galbiati, *Università degli studi di Bergamo, Italy*; Simona Tonini, *Università degli studi di Bergamo, Italy*; Gianpietro Elvio Cossali, *Università degli studi di Bergamo, Italy*; Bernhard Weigand, *Universitaet Stuttgart, Germany*

The development and fragmentation of liquid jets exiting from a pressure swirl atomizer for aeronautical applications has been numerically predicted implementing a multi-phase flow model according to the VOF (Volume Of Fluid) methodology. Direct numerical simulations (DNS) have been performed using the in-house FS3D (Free Surface 3D) code. Four operating conditions, corresponding to typical test cases for an aircraft engine, have been investigated with the fuel injected under isothermal non reacting environment. The inlet boundaries of the numerical problem, corresponding to the characteristics of the annular liquid lamella at the nozzle exit and the internal air core, have been provided by previous internal nozzle flow calculations based on the LES (Large Eddy Simulation) methodology. The characteristics of the particles formed after the jet fragmentation are predicted, calculating the break-up length, the mean particle size, velocity and deformation. In the absence of experimental data describing the flow development and the jet break-up mechanisms occurring close to the nozzle exit, DNS simulations can provide detailed information on the above mentioned physical processes. The data extracted from the simulations are compared with simplified models and semi-empirical correlations from the available literature.

## A HIERARCHY OF COMPRESSIBLE TWO-FLUID MODELS WITH IDENTIFIED PHYSICAL RELAXATIONS AND RELATED NUMERICAL METHODS FOR THE SIMULATION OF SEPARATED AND DISPERSED PHASES

Florence Drui, *CEA, France*; Marc Massot, *CentraleSupélec, EM2C, France*; Samuel Kokh, *CEA, France*; Adam Larat, *EM2C, CentraleSupélec, France*

Compressible two-fluid models offer a potential solution for simulating separated two-phase flows configurations, such as sloshing in gas tanks. On the other hand, a specific family of such models has been developed to study acoustic waves propagation in multiphase systems of dispersed gas bubbles in water. On the way to connect the separated phase zone to the disperse phase zone of a same flow, we propose here a hierarchy of two-fluid models. Starting with Hamilton's variational principle and adding thermodynamically consistent dissipative structures, we built a new connected hierarchy, each level of which is mathematically well-posed. Every new relaxation small parameter is physically identified through acoustic linearization and analysis of the dispersion relations. Furthermore, numerical methods based on finite volume schemes are developed to preserve the properties of the models at the continuous level and to asymptotically handle the transition from each subsystem to another. Finally, simulations of simple academic configurations are performed, showing the ability of the models, the numerical methods and dynamically adaptive mesh techniques to solve for the transition from a separate- to a disperse-phase flow, with the potential for massively parallel simulations.

\* Support from EM2C laboratory, Maison de la Simulation, CEA and DGA are gratefully acknowledged.

## MULTISCALE SIMULATION OF LIQUID JET DISINTEGRATION AND PRIMARY ATOMIZATION USING EULERIAN-LAGRANGIAN COUPLING

Mahdi Saeedipour, *Johannes Kepler University, Austria*; Stefan Pirker, *Johannes Kepler University, Austria*; Simon Schneiderbauer, *Johannes Kepler University, Austria*

A multiscale approach in numerical simulation of liquid jet disintegration and primary atomization is investigated using an Eulerian-Lagrangian coupling. In this approach, a numerical simulation using volume of fluid method (VOF) is carried out to model the global (meso-scale) spreading of liquid metal jet. The formation of the micro-scale droplets which are usually smaller than the grid spacing in computational domain is determined by a surface energy-based sub-grid model. Where the disruptive forces (turbulence and surface pressure) of turbulent eddy near the surface of the jet overcome the consolidating surface tension, Lagrangian droplets are released with the local properties of corresponding eddies. The dynamics of generated droplets are modelled using Lagrangian particle tracking (LPT). A numerical coupling between Eulerian and Lagrangian frames are then established via mass, momentum and volume of fluid source terms. The presented methodology was tested for different liquid jets in Rayleigh, wind-induced and atomization regimes and validated against literature data. The droplet size distributions and Sauter mean diameter (SMD) values from numerical simulations are in good agreement with experimental correlations.

## LOCAL MASS TRANSFER PHENOMENA AT SINGLE TAYLOR BUBBLES IN A VERTICAL MINI CHANNEL

Sven Kastens, *Institute of Multiphase Flows, Germany*; Jiro Aoki, *Kobe University, Japan*; Chiara Pesci, *Technische Universität Darmstadt, Germany*; Holger Marschall, *Technische Universität Darmstadt, Germany*; Kosuke Hayashi, *Kobe University, Japan*; Michael Schlueter, *Hamburg University of Technology, Germany*; Dieter Bothe, *Technische Universität Darmstadt, Germany*; Akio Tomiyama, *Kobe University, Japan*

The local effects of surfactants on the interfacial mobility, hydrodynamic condition and mass transfer performance across the contaminated interface have not been fully understood. Therefore, laser-induced fluorescence measurements with high resolution in time and space as well as direct numerical simulations are carried out to investigate the local effects of surfactant onto mass transfer processes. Visualized concentration boundary layers at the interface and concentration fields in the wake of single Taylor bubbles rising through a vertical mini channel are compared with numerical predictions. A circular pipe of 6.0 mm diameter is used with a microscope that is equipped with a high-speed video camera. Deionized water with CO<sub>2</sub> and a special fluorescent dye to visualize the local dissolved CO<sub>2</sub> concentration is applied as well as Triton X-100 as surfactant.

## STEADY STATE ANALYSIS OF FLASHING DRIVEN NATURAL CIRCULATION LOOP: INFLUENCE OF HEATER AND COOLER ORIENTATION

Kush Kumar Dewangan, Indian Institute of Technology Kharagpur, India; Prasanta Kumar Das, Indian Institute of Technology Kharagpur, India

A steady state model has been developed to study the flashing driven natural circulation loop at low-pressure conditions. Heat transport capacity of natural circulation loop (NCL) is usually low and depends strongly on the length of the riser. As riser length increases the tendency of flashing in the adiabatic section of the riser increases as a result of an increase in the density difference between the riser and the downcomer. The position or orientation of the heater and the cooler can be manipulated to modify the driving force for the circulation in the loop. Thereby the hydrodynamic and heat transport characteristics of the loop can be controlled. One dimensional two-phase homogeneous equilibrium model (HEM) has considered to estimating the void fraction in the two-phase region. Circulation with flashing and without flashing in the adiabatic section of the riser has been compared for steady state condition. This study focuses on the influences of cooler and heater orientations on the performance of flashing driven natural circulation loops. Presented model clearly predicts that, at low operating pressure flashing in the unheated riser section significantly affects the performance of the NCL. The full paper reports the variation of void fraction, circulation rate, the limit of operation over a range of working parameters.

\*This work was supported by the Department of Mechanical Engineering, Indian Institute of Technology Kharagpur.

## MASS TRANSFER OF SOLUTE AND SOLVENT ACROSS A DEFORMING PERMEABLE MEMBRANE

Shintaro Takeuchi, Osaka University, Japan; Suguru Miyauchi, Tohoku University, Japan; Takeo Kajishima, Osaka University, Japan

A numerical method for mass transfer in a two-component solution (solute and solvent) across deforming permeable membrane is proposed, and the validity of the numerical result is assessed by comparing with analytical models. The method treats the discontinuities at the membrane by incorporating the jump values into a finite element formulation on a Cartesian coordinate system non-conforming with the membrane shape. The jump values are calculated implicitly by solving coupled weak forms of the conservation equations and the interfacial equations. For assessing the validity of the numerical result, the time development of the concentration distribution of the solute on the membrane moving at a constant speed is analytically predicted. Also, an analytical model for the motion of the permeable membrane in a periodically oscillating channel flow is derived under a sufficiently large permeable coefficient for the membrane. Through comparisons of the numerical results with the analytical predictions, the validity of the proposed numerical method is established. The applicability of the method is demonstrated by applying to a mass transfer problem of a two-component solution (solute and solvent) across a deforming membrane and the interaction between the fluid and membrane is discussed.

## SLIDING DROPS – MOLECULAR VERSUS CONTINUUM SIMULATIONS

Jos Derksen, University of Aberdeen, UK

We study the sliding motion of a liquid drop embedded in another liquid over a substrate as a result of a simple shear flow. This moving contact line problem is approached through molecular dynamics and also through Cahn-Hilliard-type phase field simulation. The latter is a continuum method with diffuse interfaces. It is based on a transport equation for an order parameter that describes the local liquid composition and contains the essential physics for phase separation and surface tension. Over the interface, the order parameter switches from minus one to plus one over the distance of typically five lattice spacings (the diffuse interface). The transport equations are being solved by means of a lattice-Boltzmann method. The molecular dynamics results are used to guide the choices for the (equilibrium or non-equilibrium) boundary conditions of the order parameter at the substrate. The two simulation approaches – MD and phase field – are compared in terms of the sliding speed of the drop over the substrate, the (apparent) advancing and receding contact angles, as well as the level of drop deformation.

## HYDRODYNAMICS AND MASS TRANSFER PROPERTIES OF A ROTATING BUBBLE COLUMN

Yuk Man Lau, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Robert Herrmann Heber, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Amir Motamed Dashliborun, Laval University, Canada; Markus Schubert, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany

Bubble columns are widely applied in the industrial world. Therefore it is very important to improve their efficiency and performance in terms of quality of products. Such process intensification can be achieved by improving the conditions for heat and mass transfer, for instance increasing the interfacial area through the use of internals or introducing pulsation to increase the contact time. In the present work, rotational movements are introduced to a laboratory scale bubble column. Experiments are carried out with different rotational movements, respectively, continuous, discrete or inverting oscillations. Continuous rotation is as the name implies constant rotation of the bubble column. Discrete rotation is where the bubble column rotates step-wise. Inverting oscillation changes the direction of the column rotation. Gas holdup, bubble-size distribution, Sauter mean diameter and apparent volumetric mass transfer coefficient ( $kLa$ ) are measured for oxygen bubble dispersions in degassed and nitrogen-desorbed water. The experimental methods used are oxygen probes and imaging wire mesh sensors. It is shown that rotation schemes modify the gas-liquid dispersion pattern and the process of mass transfer.

## THE SOLIDIFICATION OF WATER DROPLETS IMPINGING TO A COOLING HORIZONTAL PLATE

Ryota Kimura, Kyoto Institute of Technology, Japan; Yoshimichi Hagiwara, Kyoto Institute of Technology, Japan

The reduction of solidification of water droplets impinging to windows is important for safety driving of vehicles. In addition, the reduction of solidification due to mist deposited to inner walls is important for defrosting refrigerators. To clarify the governing factors for this solidification, we have carried out two-dimensional numerical simulation for a water droplet falling in the air and impinging to a cooling horizontal plate. We adopted a phase-field variable to identify air, water and ice. We discretized the governing equations of mass, momentum, energy and the phase-field variable by using a fine arrangement of grids. We solved these equations by using proper schemes and a computer with graphical processing units. The computational results show that the impinging droplet becomes rapidly an attached droplet with a predetermined contact angle. When the seed crystals of ice are allocated near the air/water interface, the ice is found to grow not towards the cooling plate surface but along the air/water interface. This is in agreement with our experimental results.

\*The authors acknowledge the financial support of the Japan Society for the Promotion of Science through the Grant-in-Aid for Scientific Research (A) (No. 15H02220)

## EFFECTS OF NANOPARTICLES ON THE DEPINNING FORCE OF A RECEDING DROPLET ON MICROPATTERNED SUPERHYDROPHOBIC SURFACES

Yuhua Jiang, Stevens Institute of Technology, USA; Wei Xu, Stevens Institute of Technology, USA; Kevin Conington, Stevens Institute of Technology, USA; Chang-Hwan Choi, Stevens Institute of Technology, USA

In real practices using droplets such as self-cleaning surfaces, microfluidics, ink-jet printing, self-assembly of particles, and pesticide deposition, most of them involve particle-laden droplets instead of pure liquid. However, most of the previous studies on the pinning dynamics of droplets were performed with the droplets of pure liquid on plain surfaces. In this study, we have studied the effects of nanoparticles such as their concentration and size on the depinning force of a droplet receding on micropatterned superhydrophobic surfaces with varying shapes and dimensions. The depinning force of a droplet on smooth surfaces increases with an increase in particle concentration and a decrease in particle size, because the greater concentration of particles and the smaller size of particles result in increases in droplet viscosity and surface roughness, respectively. On micropatterned superhydrophobic surfaces, the three-phase contact line of a droplet pins along the structures until a liquid-bridge between the bulk receding droplet and the satellite liquid left on the structures ruptures. Then, the depinning motion of a droplet is delayed or accelerated by the presence of particles, depending on the competition between the delay of droplet evolvments before the formation of liquid-bridge and the acceleration of the rupture of liquid-bridge. Results show that when the concentration of particle is as low as 0.1 wt %, the particles lead to a decrease in depinning force with an increase in particle concentration. At a given particle concentration, the droplet with smaller sized particles also has a less depinning force.

## DEPINNING OF WATER DROPLETS ON PILLARED SUPERHYDROPHOBIC SURFACES UNDER DYNAMIC ICING CONDITIONS

Mohammad Sarshar, Stevens Institute of Technology, USA; Chang-Hwan Choi, Stevens Institute of Technology, USA

In this study, the parameters affecting the anti-icing properties of superhydrophobic surfaces are discussed in association with the dynamics of water droplet in a receding motion. It is found that the depinning force of a receding droplet is the key parameter to determine the delay time against icing on superhydrophobic surfaces. Based on the experimental observation, a theoretical relationship between the depinning force and the morphology of superhydrophobic surfaces having cubical and circular pillars is developed with the consideration of the shapes and dynamics of the contact line of the receding droplet on the superhydrophobic surfaces. Results show that the chemistry of a surface, the size and pitch of the pillars, and the shape of the surface structures contribute to the depinning force and affect the slope of their linear relationship. Moreover, when the change in the shape of the contact line as well as the energy necessary to dewet a pillar in the receding movement is considered, the results show that the linear relationship between depinning force and maximal three-phase contact line effective at the droplet boundary can deviate from the linearity. At the end, the results suggest that the superhydrophobic surfaces that can minimize the formation of the effective three-phase contact line at the droplet boundary should have a higher chance for the droplet to escape from the surface in the freezing stage.

## CHANNEL FLOW OF PARTICLE SUSPENSIONS: MASS AND VOLUME FRACTION EFFECTS

Luca Brandt, KTH Royal Institute of Technology, Sweden; Iman Lashgari, KTH Mechanics, Sweden; Francesco Picano, University of Padova, Italy; Walter Fornari, KTH, Royal Institute of Technology, Sweden

We perform numerical simulations of suspensions of rigid spheres in plane a channel. An Immersed Boundary Method is employed to account for the dispersed solid phase, with collision and lubrication models. Analysis of the stress of the mixture of neutrally buoyant particles enable us to identify 3 different regimes, each dominated by one of the three contributions to the total momentum transfer. A laminar regime at low volume fractions and Reynolds number, where viscous stresses are the main responsible for the total stress, a turbulent regime at higher Reynolds numbers and moderate volume fractions, dominated by the Reynolds stresses, and a regime denoted as inertial shear-thickening associated to intense particle stresses at high volume fractions. To further disentangle the role of particle and fluid inertia, we perform additional simulations varying the particle volume fraction and the fluid to particle density ratio while keeping the total mass fraction constant and neglecting settling. The results indicate that changes of the density ratio between 1 and 10 at constant volume fraction do not alter the turbulent statistics significantly. On the contrary, simulations at constant mass fraction and different volume fraction display significant modifications, which indicates that, in the parameter range investigated, the excluded volume effect is the main responsible of the modifications of the flow in the presence of the particles.

## A MECHANISTIC MODEL FOR GAS-LIQUID PIPE FLOW WITH SURFACTANTS

Dries Van Nimwegen, Delft University of Technology, The Netherlands; Gijs Van Boven, Delft University of Technology, The Netherlands; Luis Portela, Delft University of Technology, The Netherlands; Ruud Henkes, Delft University of Technology, The Netherlands

A major problem in the production of natural gas is liquid loading, i.e. the accumulation of liquids at the bottom of a well at low reservoir pressures. To prevent liquid loading, surfactants are injected at the bottom of the well, which changes the multiphase flow in the well tubing such that a smaller gas velocity is required to transport the liquids to the surface. However, no predictive models for the effect of surfactants on gas-liquid pipe flows are available.

We present results of systematic experiments on the effect of surfactants on air-water flow in vertical pipes. The surfactants lead to the creation of foam, which has a lower density than water and can be transported by the gas more easily, which leads to a smaller pressure gradient at small gas flow rates. This effect is independent on the type of surfactant used. Furthermore, from the results we obtained a relation between the thickness of the film at the wall of the pipe and the interfacial friction between the gas and the film. Using the knowledge obtained from the experiments, we developed a mechanistic model for air-water flow with surfactants. This model is able to capture the trends in the pressure gradient observed in the experiments.

\*This work is sponsored by NAM, a Dutch subsidiary of Shell and ExxonMobil.

## DIRECT NUMERICAL SIMULATION OF TURBULENT CHANNEL FLOW OVER AN EVOLVING SEDIMENT BED

Aman Kidanemariam, Karlsruhe Institute of Technology (KIT), Germany; Markus Uhlmann, Karlsruhe Institute of Technology (IfH), Germany

We have investigated the formation and evolution of sedimentary patterns in horizontal channel flow. All the scales of the driving turbulent flow have been resolved via direct numerical simulation, while the sediment bed has been represented by a large number of freely-moving finite-size spherical particles. A fluid solver, which features an immersed boundary method, has been employed to accurately treat the moving fluid-particle interfaces, while inter-particle interactions have been accounted for by adopting a soft-sphere based discrete element model. The values of the relevant parameters have been chosen such that the shearing flow erodes the sediment bed and sets particles in motion. Consequently, a continued sediment erosion-deposition process takes place and eventually leads to the formation of patterns which are, on average, perpendicular to the mean flow direction and which migrate downstream. The flow over such evolving bedforms is characterized by distinct features, such as the recirculation and flow separation region downstream of the bedform crests. In this contribution, we analyze the turbulent flow and associated particle motion to address, for instance, the spatial variation of the bottom boundary shear stress and its correlation to the local particle flow rate.

## EXPERIMENTS ON INCIPENT BED LOAD UNDER A TURBULENT STREAM

Marcos Roberto Mendes Penteado, UNICAMP - University of Campinas, Brazil; Erick De Moraes Franklin, UNICAMP - University of Campinas, Brazil

The transport of grains within a granular bed sheared by a fluid flow is commonly found in nature and in industry. In nature, it is found in rivers, oceans, deserts, and other environments. In industry, it is found in petroleum pipelines conveying grains, in sewer systems, and in dredging lines, for example. When the sediments are sheared by moderate fluid stresses, they are carried by the flow as bed load. This paper presents an experimental investigation on the motion of individual grains of a granular bed sheared by a liquid flow. The experiments were carried in a closed conduit of rectangular cross section. An entrance length, consisting of a fixed bed of same granulometry as the loose bed, assured that fully-developed turbulent water flows were imposed over a flat loose bed. Prior to each test, the loose granular bed was smoothed and leveled. The tests were performed close to incipient bed load, and grains were carried by rolling and sliding over each other, forming a moving granular layer with a thickness of the order of the grains diameter, a case for which experimental data on grains velocities are scarce. At different water flow rates, the moving layer was filmed using a high-speed camera. The grains displacement and velocity fields were determined by post-processing the images with a numerical code developed in the course of this study. Finally, the motion of grains was correlated to the water flow conditions, and the bed-load transport rate was estimated and compared with semi-empirical transport rate equations.

## NUMERICAL 3-D SIMULATION OF DEVELOPING STRATIFIED GAS-LIQUID FLOW IN CURVED PIPES

Ali Haj, Amirkabir university of technology, Iran; Zohreh Mansoori, Amirkabir university of technology, Iran; Majid Saffar Avval, Amirkabir university of technology, Iran; Goodarz Ahmadi, Clarkson University, USA

Stratified gas and liquid flow in curved pipe is crucial for many industries. In this paper, developing laminar-laminar gas-liquid flow in curved pipe has been studied to predict velocity profile and pressure gradient. The three-dimensional steady-state momentum equation is discretized and solved using finite volume scheme, and computations are performed in the bipolar coordinate system on the curve for convenient mapping of the physical domain and better convergence. The numerical method for infinite curvature compares well with analytical solution.

## EXPERIMENTAL INVESTIGATION OF GAS-LIQUID FLOW IN A HILLY TERRAIN PIPELINE OF LARGE DIAMETER

Katerina Zabkova, University of Kassel, Germany; Andrea Luke, University of Kassel, Germany

Transport of multiphase and multicomponent systems requires knowledge about the occurring dynamic processes and flow instabilities. Long distance conveyance of multiphase systems is a challenge since dissolving, degassing, sedimentation and fouling processes have to be considered depending on its composition and its thermo-physical properties. These phase changes are strongly affected by temperature and pressure. Especially multiphase transport in hilly terrain in on- and offshore regions is more susceptible to flow dynamic processes and instabilities. The up- and down stream flow leads within multiphase systems to local accumulation and following blow out of the individual phases. In this paper flow behaviour of gas-saturated liquid/gas system in a hilly terrain pipeline of a large diameter is investigated in a wide operating range. The flow within the pipe is visualised using electrical capacitance tomography and a high speed recording through an integrated glass pipe. The effect of the pipe inclinations on the flow instabilities and pressure drops for the same operating conditions is analysed experimentally and theoretically.

\* Supported by Federal Ministry of Economics and Technology on the basis of a decision by the German Bundestag

## THEORETICAL MODEL OF THE BUBBLING REGIME IN A PLANAR CO-FLOW CONFIGURATION

Rocio Bolanos Jimenez, Universidad de Jaén, Spain; Alejandro Sevilla, Universidad Carlos III de Madrid, Spain; Carlos Martinez Bazan, Universidad de Jaén, Spain

We present an analytical description that models the bubbling regime of an air film discharging between two water streams with planar geometry. Based on the previous experimental and numerical characterization of this problem, the gas stream is modeled as a planar sheet divided into three different parts: a neck that moves downwards at the water velocity, a ligament upstream of the neck, and a forming bubble downstream of the neck, with uniform dimensionless half-thicknesses  $\eta_n(t)$ ,  $\eta_l(t)$ ,  $\eta_b(t)$ , and pressures  $\pi_n(t)$ ,  $\pi_l(t)$ , and  $\pi_b(t)=\pi_n(t)$ , respectively. Lengths are made dimensionless with the air thickness at the outlet,  $H_0$ , and pressures with twice the air dynamic pressure,  $\rho_a u_a^2$ . In a reference frame moving with the water velocity,  $uw$ , and imposing a negative pressure caused by the sudden planar expansion of the air stream at the outlet of the injector, a set of differential equations are obtained, that can be numerically integrated to obtain the temporal evolution of the different thicknesses and pressures, showing a good agreement with numerical and experimental results for a given value of the initial velocity of the collapsing neck. The latter is the only free parameter of the model, that shows a weak dependence on the control parameters, namely,  $We = \rho_a u_a^2 H_0 / \sigma$ , and  $\Lambda = uw / u_a$ , where  $\sigma$  is the surface tension coefficient.

\* Support from Spanish grants MINECO DPI2014-59292-C3-1-P and DPI2014-59292-C3-3-P, Junta de Andalucía P11-TEP749 and Universidad de Jaén UJA2013/08/05.

## BUBBLE ENTRAINMENT BEHAVIOUR DURING DROP IMPINGEMENT ONTO LIQUID SURFACE

Hisanobu Kawashima, Gunma University, Japan; Shun Fujishima, IHARA SCIENCE CORPORATION, Japan; Soichiro Ogawa, Gunma University, Japan; Tsuneaki Ishima, Gunma University, Japan

It has been well known that a behavior of free liquid surface such as drop impact as a jet, a splash, a rain drop and so on. Many previous studies have discussed the drop impingement behavior by using same fluid between a drop and free surface. In this study, a bubble entrainment motion during drop impact onto static fluid surface is observed with the conditions of same and different fluids for a drop and free surface. For the same fluid condition, water, oil, and HFE (Hydro Fluoro Ether) are tested for both a drop and a free surface. For the different fluid condition, HFE is dropped onto water surface. The impingement behavior is visualized by using a high-speed video camera. The relationship between the Weber number and Froude number is equal to the previous study results when the bubble is trapped under the free surface using the same kind of fluid. On the other hand, for the different liquid between droplet and free surface, the Weber number becomes smaller than that of used of liquid of the same fluid. Furthermore, the hollow shape of near free surface during drop impact is adjusted using the non-dimensional parameter regardless of the test condition.

\* Support from a grant of JSPC KAKENHI 15K05782 is gratefully acknowledged.

## MODELLING OF BUBBLE DYNAMICS AND EULER/LAGRANGE PREDICTIONS OF BUBBLE COLUMNS

Marcelo Muniz, Martin-Luther-University Halle-Wittenberg, Germany; Martin Sommerfeld, Martin-Luther-University Halle-Wittenberg, Germany

Bubble dynamics is still being a challenge on the mathematical modelling for CFD codes when applied on bubble columns. The flow in bubble columns is basically induced by the buoyancy force acting on bubbles. However, there are other important forces which depend on bubbles shape, size, orientation and its motion, such as drag, added mass, Basset force and transverse lift. In this framework, a CFD model is developed and implemented in an open source platform (OpenFOAM®), based on an Euler/Lagrange approach, considering all these forces. Flow field and turbulence of the carrier phase was modeled by Large Eddy Simulations (LES) approach, considering also the turbulence induced by bubbles. The effect of sub-grid scale turbulence on bubble motion was described by a stochastic single-step Langevin equation, based on Lagrangian and Eulerian time scales. Moreover, the bubble dynamics in the point-particle approach was modelled by stochastic variations of bubble shape and orientation according to experimental observations. The numerical simulations, including comprehensive parameter studies (e.g. using different forces correlations), were compared with experimental data from Sommerfeld and Bröder (2009).

\* Support from CNPq (National Council for Scientific and Technological Development – Brazil) and DAAD (German Academic Exchange Service) is gratefully acknowledged.

## NUMERICAL STUDY ON THE WETTABILITY DEPENDENT INTERACTION OF DROPLETS AND BUBBLES WITH SOLID STRUCTURES

Martin Woerner, Karlsruhe Institute of Technology (KIT), Germany; Xuan Cai, Karlsruhe Institute of Technology (KIT), Germany; Holger Marschall, Technische Universität Darmstadt, Germany; Olaf Deutschmann, Karlsruhe Institute of Technology (KIT), Germany

Simulations of the wettability dependent interaction of drops/bubbles with solid structures require accurate modeling of the moving contact line (MCL). Sharp interface models suffer from a paradox between MCL and no-slip conditions at the wall which is usually resolved by an empirical slip length. Phase field methods treat the interface as a region of finite thickness and resolve this paradox by a pure diffusive mechanism for the MCL. Recently, we coupled the two-phase Navier-Stokes eq. with a phase-field description for the interface evolution (Cahn-Hilliard eq.) and implemented this method in OpenFOAM®. We validated it for fundamental wetting processes and demonstrated its capability for 3D simulations with adaptive mesh refinement near the interface (DOI: 10.1002/ceat.201500089). Here, we highlight our advances on simulation of droplet dynamics on flat chemically patterned and on micro-structured substrates with regular roughness. We also present simulations of the rise of a bubble through a periodic open cellular structure (as used in innovative chemical reactors) and show that the bubble dynamics strongly depends on the structure wettability.

\*The funding by Helmholtz Energy Alliance "Energy-efficient chemical multiphase processes" (HA-E-0004) is gratefully acknowledged.

## WAKE FEATURES OF BIRDS DURING FLAPPING FLIGHT, CASE STUDY: STARLING, SANDPIPER AND ROBIN

Roi Gurka, Coastal Carolina University, X

A great deal of research focusing on flapping wings has been motivated by their high performance capabilities, especially in low Reynolds number configurations. Birds' unique characteristics results in high aerodynamic performance. The near wakes of three distinct birds: starling, sandpiper and robin have been investigated using long-duration time-resolved particle image velocimetry (PIV), combined with high-speed imaging. Time series of the vorticity fields have been expressed as composite wake plots, which depict segments of the wing beat cycle for various spanwise locations in the wake. The composite wakes feature several characteristics of the wake over the different phases of the wingbeat cycle. Comparison between the near wake fields of the three birds reveals remarkable similarity in their wake structures. We have identified over multiple wing beat cycles the presence of what appears to be an overlap of two distinct wake features during the transition phases from downstroke to upstroke and vice versa. It appears that these features produce net positive vorticity over the wing beat cycle. In these continuously shed concentrated vorticity regions, the majority of net positive circulation is accumulated, indicating this may be a key feature in producing lift, and thus the high aerodynamic performance observed by these low Reynolds flyers.



# **BREAKUP OF FINITE SIZE COLLOIDAL AGGREGATES IN TURBULENT FLOW INVESTIGATED BY 3D PARTICLE TRACKING VELOCIMETRY**

Matthaus Babler, KTH Royal Institute of Technology, Sweden; Debashish Saha, Eindhoven University of Technology, The Netherlands; Markus Holzner, ETH Zurich, Switzerland; Miroslav Soos, University of Chemistry and Technology Prague, Czech Republic; Beat Luthi, Photrack AG, Switzerland; Alex Liberzon, Tel Aviv University, Israel

Aggregates grown in mild shear flow are released, one at a time, into homogeneous isotropic turbulence where their breakup is recorded by 3D-PTV. The aggregates have an open structure with fractal dimension around 1.8 and their size is  $1.4 \pm 0.4$  mm which is large compared to the Kolmogorov length scale  $\eta = 0.15$  mm. 3D-PTV allows for the simultaneous measurement of aggregate trajectories and the full velocity gradient tensor along their pathlines which enables us to access the Lagrangian stress history of individual breakup events. We found no consistent pattern that relates breakup to the local flow properties. Also the correlation between the aggregate size and shear stress at the point of breakage is found to be weaker when compared with the correlation between size and drag stress. The analysis suggests that aggregates are mostly broken due to accumulation of drag stress over a time-lag of order Kolmogorov time scale. This is due to the fact that the aggregates are large which gives their motion inertia and increases the time for stress propagation inside the aggregate. It is also found that the scaling of the largest fragment and the accumulated stress at breakup follows an earlier established power law obtained from laminar nozzle experiments. This indicates that despite the large size and the different type of hydrodynamic stress, the microscopic mechanism that cause breakup is consistent over a wide range of aggregate size and stress magnitude.

# **THREE-DIMENSIONAL SCENARIO OF DRAG REDUCTION IN BUBBLY TAYLOR-COUETTE FLOWS**

Yuichi Murai, Hokkaido University, Japan; Tsubasa Ozaki, Hokkaido University, Japan; Yuji Tasaka, Hokkaido University, Japan

Our previous experiment of bubbly Taylor-Couette flow found a large impact of spiral bubble clouds in promoting drag reduction (Murai et al. 2008). The spiral bubble clouds are naturally formed without axial base flow of liquid phase, and enable to reduce momentum transfer the most sensitively among various two-phase flow patterns taking place into the flow system. In the present study, we investigate how the drag reduction is enhanced, by measuring liquid vortical flow structures affected by bubbles from 600 to 4000 in Reynolds number. Ultrasound velocity profiling coupled with particle tracking velocimetry has been set up to reveal the three-dimensional two-phase flow structure, only by which the scenario of drag reduction by bubbles can be understood. Our new findings are as follows; 1) when liquid phase organizes toroidal array of Taylor vortices, bubble phase can form a single spiral cloud in the case of dilute bubble suspension. 2) By increasing void fraction, the spiral bubble cloud drives liquid phase to transit to spiral Taylor vortices. 3) Between these two modes, toroidal and spiral modes appear intermittently resulting in high drag reduction rate, i.e. breaking down of the primary vortical structures that carry the azimuthal momentum of liquid phase in the radial direction. A numerical simulation based on Eulerian-Lagrangian bubbly flow model equation has also obtained this scenario qualitatively.

# **INVESTIGATION OF GAS PERFUSION THROUGH POROUS MEDIA WITH AND WITHOUT SUPER-HYDROPHOBIC COATINGS**

James W. Gose, University of Michigan, USA; Kevin Golovin, University of Michigan, USA; Anish Tuteja, University of Michigan, USA; Steven L. Ceccio, University of Michigan, USA; Marc Perlin, University of Michigan, USA

Super-hydrophobic (SH) materials have been used successfully to reduce skin-friction in laminar flows and therefore has guided researchers to apply SH materials in turbulent flows. Oftentimes, this has been unsuccessful at providing meaningful skin-friction drag reduction, and has even generated increased drag. This failure is frequently attributed to the wetting of an SH surface or equivalently the transition from the Cassie-Baxter to the Wenzel state. The result is fluid flow over an essentially roughened surface. In this investigation the researchers aim to perfuse small amounts of gas through porous media, including sintered and foam metals, to achieve skin-friction drag reduction in a fully-developed turbulent channel flow. As air is perfused through porous media, the solid – liquid interface transitions to a solid – liquid – gas interface. This can result in an interface that functions similarly to SH materials. Controlled air perfusion that provides the necessary gas replenishment at the interface may prevent wetting, and consequently eliminate or reduce the effect of the roughness on the flow. This latter possibility is investigated by supplying small amounts of gas to a porous media with and without SH coatings. To quantify the effectiveness of this method, pressure drop is used to infer friction drag along the surface in a fully-developed turbulent channel flow. PIV measurements are used to directly measure the mean flow along the surface.

\*Support from the Office of Naval Research (ONR) is greatly appreciated.

# **LINEAR VISCOELASTIC ANALYSIS OF BUBBLE SUSPENSIONS BY MEANS OF ULTRASONIC SPINNING RHEOMETRY**

Yuji Tasaka, Hokkaido University, Japan; Richard Rapberger, Johannes Kepler University, Austria; Konrad Dymowski, Technical University of Lodz, Poland; Yuichi Murai, Hokkaido University, Japan

Dilute bubble suspensions in unsteady shear flows take elastic characteristics originate with surface tension of bubble-liquid interface. In steady flows the surface tension effects are reflected in modifications of effective viscosity, and this variation against shear rate is determined by Capillary number and void fraction. But, in unsteady situations, there is no universal evaluation of its viscoelasticity. In the present study, we provide linear viscoelastic analysis on bubble suspensions of 1000 cSt silicone oil with around 1% of void fraction by means of newly established methodology, named ultrasonic spinning rheometry. This is based on ultrasonic velocity profile measurements in oscillating shear flows with a cylindrical configuration: equation of momentum conservation and Maxwell model of viscoelastic materials provide viscosity and elastic modulus for various shear rates at each oscillation frequency. We will introduce methodology of the present rheometry including its advantages, problems, and some ideas to overcome enhancement of measurement errors on numerical process. Then viscoelastic characteristics of the bubble suspensions for setting oscillation frequencies ranged from 0.5 to 2.5Hz will be discussed

# **GAS-LIQUID TWO-PHASE FLOWS IN A LAB-SCALE WATER ELECTROLYSIS CELL**

Wei Bai, SINTEF Materials and Chemistry, Norway; Abdelghafour Zaabout, SINTEF Materials and Chemistry, Norway; Alejandro Oyarce Barnett, SINTEF Materials and Chemistry, Norway; Magnus Thomassen, SINTEF Materials and Chemistry, Norway

In the present work, bubbly flows at anode side during the process of water electrolysis are studied using computational fluid dynamics (CFD). A two-phase model is adopted in the study, which consists of two main parts to deal with gas bubbles. Gas bubbles at large scale are resolved by using the volume of fluid (VOF) method, while small bubbles mainly due to nucleation are considered in a sub-grid model. The produced specie at anode, oxygen, is solved by a specie transport equation. For the purpose of validation of the two-phase model, the bubbly flows during water electrolysis are modeled in a lab-scale cell. In the experiments, different gas flow rates are employed in the test cell to simulate the process of water electrolysis at different current densities. High speed camera is used to capture gas bubbles moving in the lab-scale cell. An image processing method is then performed to the images of gas bubbles obtained from both the experiments and numerical simulations. Various bubble properties, such as equivalent bubble size, bubble velocity and average number of bubbles, are compared between the measurements and numerical simulations.

# **EFFECT OF SUBCOOLING ON FILM BOILING HEAT TRANSFER CHARACTERISTICS OF JET ON A SUPERHEATED PLATE**

Koji Hasegawa, Kogakuin University, Japan; Hiroyasu Ohtake, Kogakuin University, Japan; Masahiro Ueno, Kogakuin University, Japan

This paper presents the film boiling behavior of jet on a superheated plate using high-speed video camera and IR camera. Because of its cooling characteristics and usefulness, the boiling heat transfer and film boiling collapse temperature has for decades been a focus of investigation by researchers in the fields of steel manufacturing. This boiling phenomena are closely related to wettability and inception of wetting on the heated surface. Although many investigations on the wetting behavior and film boiling collapse have been reported over the past decades, there have been only few experimental findings. Determining the boiling mechanisms and elucidating the phenomena in film boiling collapse temperature are crucial. The purpose of our study is to investigate experimentally the boiling heat transfer characteristics and film boiling collapse temperature. In this paper, we recorded the film boiling collapse behavior and the temperature history by a high-speed video camera and IR camera, respectively. The liquid subcooling was 0, 10, 20, 30, 40, 50 and 75K. From the results, we estimated the dynamics of the film boiling of a jet on a heated plate of the silver (40 mm× 40 mm) and the experimental data is compared with the existing experimental correlation. In the case of lower subcooling, the film boiling collapse behavior good agreement with the existing model.

## EXPERIMENTAL STUDY OF WATER POOL BOILING ON A SINGLE NUCLEATION SITE, FROM ATMOSPHERIC PRESSURE DOWN TO LOW PRESSURE

Sandra Michaie, CETHIL UMR5008, Université de Lyon, CNRS, INSA-Lyon, Univ. L, France; Romuald Rulliere, CETHIL UMR5008, Université de Lyon, CNRS, INSA-Lyon, Univ. L, France; Jocelyn Bonjour, CETHIL UMR5008, Université de Lyon, CNRS, INSA-Lyon, Univ. L, France

Heat transfer during boiling is of great interest for cooling systems. In the case of sorption chillers or heat pumps using water as the refrigerant, specifically in their evaporator part, boiling occurs at low pressure levels. In these conditions, pressure and subcooling degree are no more homogeneous around the bubbles. These non-homogeneities lead to changes in the boiling phenomenon: bubbles size, shape and frequency are modified compared to what is usually observed at atmospheric pressure.

Experiments of water pool boiling were conducted at saturation state from atmospheric pressure down to 4.2kPa with an imposed heat flux of 2.7W/cm<sup>2</sup> at the heater. Nucleation took place on a single artificial site made at the center of the upward facing copper boiling surface located at the top of the heater. Videos of each experiment were recorded owing to a high speed camera in order to study the bubble dynamics for the different pressure levels.

It appears that as the pressure decreases, the size of bubbles increases and the detachment frequency of bubbles decreases. Close to atmospheric pressure, bubbles are near-spherical whereas at the lowest pressure range, bubbles have an oblate spheroidal shape. In the range of intermediate pressures, the bubbles get a mushroom shape due to the merging of the successive one or two bubble(s) sucked into the wake of the first one. After detachment bubbles collapse quite rapidly because of the subcooling degree.

## EFFECT OF OSCILLATORY FLOW FIELD ON BOILING IN SMALL DIAMETER TUBE AT ATMOSPHERIC PRESSURE

Manoj Kumar, Indian Institute of Technology Kharagpur, India; Arup Kumar Das, Indian Institute of Technology Roorkee, India, India; Prasanta Kumar Das, Indian Institute of Technology Kharagpur, India

Flow boiling in mini tube is experimentally observed to understand the effect of oscillation on the bubble dynamics. Indigenous experimental setup has been made using a transparent quartz tube having inner and outer diameter 4 mm and 6 mm respectively. Electrical heating around the tube is arranged for supplying the heat flux to the working fluid. Generated bubbles will collapse at a water cooled condenser at the top of the setup. The oscillations have been provided using permanent magnets over a diaphragm at the heating section. The frequency has been varied up to 5 Hz at different amplitudes in the diaphragm. We observe that bubble growth behaves differently when oscillation at particular frequency is implied. Initiation of boiling heat transfer is delayed as compared to unidirectional flow situation. The boiling also starts earlier at the higher frequency and lower amplitude as compared to low frequency and high amplitude. Bubble merging and coalescence has also been studied with other parameters that give further insight in to the process. Repeated pattern of blossoming and extinguishing of the bubbles is observed under oscillatory motion inside the pipe. Insights from the experimental observations may help in transfer processes and avoiding detrimental effects like cavitation.

## SPRAY IMPINGEMENT ON SPHERICAL TARGETS IN HOMOGENEOUS, ISOTROPIC TURBULENCE

Paul Andrade, Imperial College London, UK; Georgios Charalampous, Imperial College London, UK; Yannis Hardalupas, Imperial College London, UK

Collisions of droplets with particles in a turbulent carrier phase occur in many environments including spray dryers. The collision outcome may be deposition or re-atomisation. The liquid deposition rate during collisions depends on the relative droplet-particle size and velocity, the angle of collision as well as the carrier phase turbulence. Until now the contribution of the carrier phase turbulence on the deposition rate has not been experimentally quantified. An experimental investigation is conducted on the interaction between a polydispersed spray and a spherical target in homogeneous and isotropic turbulence without mean flow. The spray of low momentum water droplets with mean diameters around 50µm is created by an ultrasonic atomizer. The target diameter is between 0.5 and 2 mm. Turbulence is generated by a unique 'box of turbulence' facility, using loudspeakers to create interacting synthetic jet arrays in the centre of a cube. Turbulent Reynolds numbers, Re<sub>λ</sub>, up to 200 are considered. The droplet sizes and velocities are measured simultaneously using Phase Doppler Anemometry. The growth rate of the liquid film, which is deposited on the spherical target, is quantified through imaging until it detaches from the target due to gravity. The results will quantify the influence of initial droplet size and velocity distribution, local droplet number density and turbulence levels on the growth rate of the liquid deposit on the target and the time that it takes for the detachment event from the target to occur. The experimental findings will be discussed relative to current CFD models for liquid deposition.

## EFFECT OF WETTABILITY ON POOL BOILING ONSET

Marco Marengo, University of Brighton, UK; Benoit Bourdon, Euro Heat Pipes, Belgium; Fabio Villa, University of Mons, Belgium; Ileana Malavasi, University of Bergamo, Italy; Romain Rioboo, Euro Heat Pipes, Belgium; Emilie Bertrand, University of Mons, Belgium

The miniaturization of electronic components and their increasing integration, with the fascinating perspective of micro-robots, micro- and nano-systems for sensors and fluid manipulation, drive important technological advancements. Thermal management in micro- and nanoscale systems is a bottleneck in the further development of such technologies. Among the different ways to remove heat for these systems, passive phase-change heat transfer is one of the most efficient. It allows obtaining significant heat fluxes, spreading the heat from hot spots to larger heat sinks without need of pumping. Even if they are efficient, two-phase systems present drawbacks such as the gravity dependence or the activation of boiling. One of the major challenges is to reduce as much as possible the superheat temperature needed to activate bubble nucleation. The surface features, such as the topography and the wettability, have a strong effect on the onset of boiling and wettability becomes the predominant factor when the surface roughness is decreased under a given value. The paper aims at resuming authors' research activities in the last nine years to understand how the wettability is influencing the boiling. Different surface treatments such as chemical grafting or patterning have been applied on different surfaces to tune their wettability. A quasi-Leidenfrost regime has been discovered. A theoretical approach, based on MD simulations, has been proposed with an accurate qualitative agreement. The results are finally compared with the most important and recent experimental results in literature.

## EXPERIMENTAL VALIDATION OF ANALYTICAL MODELS FOR LIQUID FILM BREAK-UP

Hakon Line, TU Graz, Austria; Helfried Steiner, TU Graz, Austria; Guenter Brenn, Graz University of Technology, Austria; Philipp Engesser, Lam Research, Austria

In wet spin processing the complete coverage of the processed surface with operating liquid is generally a prerequisite for achieving the desired results. Much research has therefore been devoted to finding reliable criteria for a continuous wetting of the substrate. The underlying mechanisms for the occurrence of wetting discontinuities due to a film breakup into rivulets have been thus far mainly investigated for gravity driven falling films. Based on balances of momentum and energy in the continuous and broken states, these studies have yielded various analytical correlations for the minimum film thickness. This work extends these models to centrifugal force driven rotating films and assesses their scope and limits by a comparison against experimental data, which have been presently acquired for a comprehensive range of operating conditions and liquid/solid-surface combinations. As already shown by linear stability analyses for falling films as well as for spreading drops on rotating substrates in the literature, it is further demonstrated that the capillary fingering instability essentially triggers the breakup of the radially spreading liquid ridge. As such this type of instability appears to determine the maximum radial extension of the fully wetted surface in the presently considered rotating films, which are continuously supplied with liquid from a vertical jet, as well.

\*The authors gratefully acknowledge the support from the COMET K1 Competence Center ASSIC and Lam Research.

## A STOCHASTIC APPROACH TO COLLOIDAL PARTICLE COLLISION/AGGLOMERATION

Radu Eduard Maftei, INRIA, France; Mireille Bossy, INRIA, France; Jean-Pierre Minier, EDF, France; Christophe Profeta, Université d'Evry Val d'Essonne, France

In the present work, we address the problem of determining the collision and agglomeration kernels of colloidal particles carried by fluid flows when a stochastic model is used to represent colloidal particle transport. More precisely, the objective is to derive expressions for these kernels that, for the case of particle collisions, are defined as the collision rate divided by the particle concentration. The underlying goal is to obtain expressions of these kernels that remain valid even in complex situations, for example when the combined effects of Brownian motion, turbulence and wall-bounded flows, are met.

When the driving model consists in Brownian motion and each collision event results in perfect agglomeration, a definition of the collision kernel was proposed by Smoluchowski. Yet, this formulation needs to be generalized to more complex situations. In this work, we first introduce a similar definition when particles driven by a Langevin model (when both particle locations and velocities are jointly modeled). When perfect agglomeration is still assumed, it is shown that this new definition is consistent with the Brownian agglomeration kernel in the Smoluchowski limit. Then, the effects of non-perfect agglomeration events, such as collisions followed by specular reflection, are addressed in the context of Langevin models for colloidal particle transport.

## PARTICLE-WALL ADHESION MODEL FOR TURBULENT DISPERSE MULTIPHASE FLOWS WITHIN AN EULER-LAGRANGE LES APPROACH

Michael Breuer, Helmut-Schmidt University Hamburg, Germany; Naser Almohammed, Helmut-Schmidt University Hamburg, Germany

The present study is concerned with the particle-wall adhesion in turbulent particle-laden flows. In the framework of a four-way coupled Euler-Lagrange approach relying on the large-eddy simulation technique and a hard-sphere model with deterministic collision detection (inter-particle and particle-wall) a momentum-based wall adhesion model is developed. Taking the van-der-Waals force on dry electrostatically neutral particles into account, the conditions for the deposition of particles on bounding walls due to sticking or sliding collisions including friction and inelasticity are determined. The modeling assumptions are in accordance with a recently developed agglomeration model. The influence of the adhesive force on the deposition process is studied in a turbulent particle-laden channel flow under various condition, e.g. different sizes of the primary particles, different restitution coefficients, smooth and rough walls, ...). Furthermore, the agglomeration and the adhesion models are simultaneously taken into account. The results show that the number of agglomerates deposited on the walls is relatively low compared to the deposited primary particles. Finally, the new adhesion model is applied to study a practically relevant turbulent flow over curved walls.

## A NEW MIXTURE MODEL FOR COMPRESSIBLE MULTIPHASE FLOWS

Marc Henry De Frahan, University of Michigan, USA; Shahab Alahyari Beig, University of Michigan, USA; Bahman Aboulhasanzadeh, University of Michigan, USA; Harish Ganesh, University of Michigan, USA; Steven L. Ceccio, University of Michigan, USA; Eric Johnsen, University of Michigan, USA

Compressible multiphase flows are ubiquitous in many applications, including combustion efficiency, naval ship design, and biomedical procedures. Computational approaches to study these flows offer many advantages over experiments: they are cheaper, not limited by diagnostics, and offer control of the initial conditions. However, because of the flow characteristics, multiphase flows are challenging to simulate. Modeling the system, discretizing the evolution equations, and resolving the relevant length scales must be done carefully. Our approach combines several new computational models and methods. The flow is modeled using the multiphase Euler equations, which are solved using a high-order Discontinuous Galerkin method. Through a novel limiting procedure, spurious pressure oscillations at interfaces are avoided. The Message Passing Interface is combined with graphics processing units to form a novel high-performance computing paradigm. The problems of interest are single-bubble collapse dynamics, high-speed droplet breakup, and bubbly shocks in supersonic flow over a wedge. In this latter setup, air and water form a homogeneous bubble-liquid fluid with volume fractions close to 0.5. The supersonic flow over the wedge leads to the creation of a shockwave at the wedge's leading edge. We show good agreement over a wide range of Mach numbers between our simulations of the supersonic bubbly flow and experimental data (Eddington, 1970).

## A NEW HIGH ORDER MOMENT METHOD FOR POLYDISPERSE EVAPORATING SPRAY MODELING AND SIMULATION DEDICATED TO THE COUPLING WITH SEPARATED TWO-PHASE FLOWS IN AUTOMOTIVE ENGINE

Mohamed Essadki, CentraleSupléc, EM2C, France; Stephane De Chaisemartin, IFPEN, France; Marc Massot, CentraleSupléc, EM2C, France; Frederique Laurent, EM2C, CentraleSupléc, France; Adam Larat, EM2C, CentraleSupléc, France; Stephane Jay, IFPEN, France

Liquid fuel injection modeling and simulation in automotive engines face new challenges related to the need for predictive simulations of combustion regimes. Recently, derived from a statistical approach at mesoscopic level, a high order method of moments coupled to realizable, robust and accurate numerical methods, has been obtained and shown to describe properly the dynamics of polydisperse evaporating sprays and its coupling to a turbulent gaseous flow field. However, building up a global multi-scale model with the capability to resolve the whole injection process requires a major breakthrough in terms of both modeling and numerical methods. A new model for evaporating polydisperse sprays with easy coupling to the separated phase zone is proposed in the present contribution, as well as the specific numerical methods and implementation in the p4est library for adaptive mesh refinement and massively parallel computing. The key ingredient is a good choice of variables, which can describe both the polydisperse character of a spray as well the topology of an interface. After verification cases, some challenging two-phase simulations in terms of both physics and high performance computing will be shown.

\* Support from EM2C laboratory and IFPEN are gratefully acknowledged.

## COMPUTING COMPLEX MULTIPHASE FLOWS WITH BLUE

Damir Juric, LIMSI-CNRS, France; Jalel Chergui, LIMSI-CNRS, France; Seungwon Shin, Hongik University, Republic of Korea; Lyes Kahouadji, Imperial College London, UK; Omar K. Matar, Imperial College London, UK

BLUE is software for simulations of multiphase flows on several to over 100 000 parallel processes. A dual Front Tracking/Level Set approach accurately handles capillary forces and complex interface dynamics including changes of topology resulting from breakup. Modules for flow, mass and heat transport are designed with particular attention to parallel memory management. We demonstrate solutions in a variety of challenging multiphase applications in microfluidics, free surface and interfacial flows as well as interactions of multiphase flows with solid structures. We also describe the architecture and discuss the parallel performance of the code.

\*This research was supported by the UK Engineering and Physical Sciences Research Council (EPSRC) Programme

## NUMERICAL SIMULATIONS OF SUBCOOLED BOILING USING A TWO-FLUID/DQMOM METHODOLOGY

Klas Jareteg, Chalmers University of Technology, Sweden; Srdjan Sasic, Chalmers University of Technology, Sweden; Paolo Vinai, Chalmers University of Technology, Sweden; Christophe Demaziere, Chalmers University of Technology, Sweden

We present a formulation and implementation of a framework for simulating dispersed bubbly flows under subcooled conditions. The solver is based on a two-fluid methodology augmented by the Direct Quadrature Method of Moments (DQMOM). The latter allows us to retrieve the bubble size distribution in the system and thus resolve a size-dependent coupling between the dispersed vapor and the continuous liquid phases. The condensation of the vapor bubbles is formulated in terms of an internal phase space convection which results in a precise representation of the continuous bubble shrinkage associated with the subcooled conditions. We further suggest recommendations for the algorithm and the numerical implementations. The framework is compared to the multiple size group (MUSIG) method and we demonstrate the applicability of the proposed framework on a set of cases with realistic simulations of water vapor bubbles in a subcooled liquid phase. We characterize the advantage of the dynamic bubble sizes of DQMOM in terms of resolving the heterogeneity in the bubble size distribution throughout the domain. Furthermore, the proposed solver is evaluated in terms of computational performance and robustness as regards the proposed coupling algorithm.

## LDV ANALYSIS OF GAS-LIQUID SLUG FLOW IN A HORIZONTAL PIPE

Jignesh Thaker, S V National Institute of Technology Surat, India, India; Jyotirmay Banerjee, S V National Institute of Technology Surat, India, India

Interfacial flow structure of liquid slug is responsible for several erosion-corrosion problems in the piping systems of industries including nuclear, petroleum, chemical and geothermal plants. A Laser Doppler velocimetry (LDV) analysis is reported here for intermittent and transient characteristics of slug flow under varying inlet flow conditions in terms of superficial Reynolds numbers of gas (ReSG) and liquid (ReSL). In this direction, two-phase flow experiments are performed in a horizontal pipe and local axial velocity in the liquid slug is measured using LDV and fully automated traverse system. Based on these measurements, axial velocity profiles along radial direction are established in the liquid slugs for different inlet flow conditions. Such velocity profiles are observed to be asymmetric near the nose and tail of slug bubbles. Almost fully developed velocity profiles are observed at mid-section of the liquid slug. However within the liquid layer below a passing slug bubble, the velocity never develops into a quasi-fully-developed profile. At a given radial location the axial velocity gradually reduces towards the slug bubble tail. However, a rapid increase in axial velocity is observed near the slug bubble tail due to suction in the wake region formed by the fast moving gas bubble.

\* Support from Center of Excellence (COE) on Water Resources and Flood Management, S. V. National Institute of Technology, India (sanction letter no. NPIU/TEQIP-II/FIN/47 dated 21-06-2013) for LDV facility is gratefully acknowledged.

## EXPERIMENTAL ANALYSIS OF DOWNWARD GAS-LIQUID SLUG FLOW IN SLIGHTLY INCLINED PIPES

Romulo Rodrigues, Federal University of Technology - UTFPR, Brazil; Fabio Schneider, Unibrasil, Brazil; Marco Da Silva, Federal University of Technology - UTFPR, Brazil; Moises Marcelino Neto, Federal University of Technology - UTFPR, Brazil; Rigoberto Morales, Federal University of Technology - UTFPR, Brazil

Downward gas-liquid slug flow occurs in hilly-terrain pipelines and is frequently observed in oil and gas transportation lines. The onset of this flow type is due to instabilities generated by irregular pipe topography. Therefore, to understand the hydrodynamics of the slug flow is paramount in the design of crude oil production lines as well as in the project of equipment involved in oil and gas operations. An existing experimental rig in the NUEM/UTFPR labs was used to collect data. The goal of this work is to experimentally analyze slug parameters such as velocity and frequency in ducts with inclination angles from 0 down to 13°. The analysis was performed at different gas-liquid volumetric flow rates for which the slug flow regime was observed. A pair of wire-mesh sensors to evaluate the flow structure, thus obtaining void fraction temporal series was employed. From those temporal series, the slug parameters were obtained and comparative analysis with previous work could be made.

## EXPERIMENTAL RESEARCH OF CONDUCTANCE AND CAPACITANCE COMBINATION SENSOR FOR WATER HOLDUP MEASUREMENT IN OIL-WATER TWO-PHASE FLOW

Hao Wu, Tianjin University, China; Chao Tan, Tianjin University, China; Feng Dong, Tianjin University, China

Oil-water two-phase flow widely exists in the process of petroleum industry. The liquid holdup measurement in horizontal pipeline is very important and difficult. To measure the water holdup of oil-water two-phase flow, a conductance and capacitance combination sensor (CCCS) system with four conductance rings and two concave capacitance plates was designed and validated for its measurement performance through dynamic experiments. The conductance sensor and the capacitance sensor work alternatively in the same sensing volume with switch frequency of 500Hz. During the simulation with finite element analysis, the capacitance measurement results are influenced by the conductance probes. The conductivity of conductance probes will reduce the capacitance value. The influence has been verified by the static response test experiments. The theoretical correlations of conductance sensor and capacitance sensor were established. A real-time measurement method by CCCS system was provided based on the fusion of the conductance and the capacitance measurement without flow pattern recognition. The full scale error of the water holdup measurement is less than 5% based on the quick closing valves calibration. Therefore, the CCCS system has application prospect in petroleum industry.

\* Support from the National Natural Science Foundation of China (No. 61227006 and No. 61473206) and the Science and Technology Innovation Plan of Tianjin (13TJXSYJC40200) is gratefully acknowledged.

## CHARACTERIZATION OF NEARLY JAMMED GRANULAR FLOWS IN HIGH SHEAR SYSTEMS THROUGH MOLECULAR DYNAMICS SIMULATIONS

Mohammad Khalilitehrani, Chalmers University of Technology, Sweden; Srdjan Sasic, Chalmers University of Technology, Sweden; Anders Rasmuson, Chalmers University of Technology, Sweden

We investigate here a non-local behavior commonly observed in nearly jammed granular flows. We identify the presence of force networks and crystallization as the main origins of such a behavior and carry out Molecular Dynamics (MD) simulations (based on Hertzian soft sphere models) on a high shear disk impeller granulator. The non-local behavior of the system is characterized by a pair correlation function and the force probability distribution. The latter two are used to estimate macro-scale quantities as functions of micro-scale characteristics. The stability of the force chains is characterized by mean packing parameters (averaged coordination number, averaged normal force and volume fraction). It is observed that, by a slight departure from having equally-sized particles, a discontinuity takes place in the momentum transfer mechanisms and the structure of the force networks. This discontinuity indicates a certain degree of crystallization in monosized particle systems. Therefore, the effect of polydispersity on micromechanics of the system is considered. Suitable averaging and statistical methods are applied to link the MD simulations to a continuum description of the system. The simulation results show promising agreement to experimental data.

## DETERMINATION OF HYDRODYNAMIC PARAMETERS OF AIR-WATER SLUG FLOW USING ULTRASONIC TECHNIQUES VIA HILBERT-HUANG TRANSFORMS

Baba Musa Abbagoni, Cranfield University, UK; Hoi Yeung, Cranfield University, UK

Two-phase slug flow is the most common flow regime in horizontal flow pipe. It is intermittent, transient and its hydrodynamic parameters are difficult to predict. Determinations of the hydrodynamic parameters are required for many designs calculation in pipe and downstream equipment (Romero et al., 2012). In this study, experimental measurements of air-water flow on a horizontal flow pipe were conducted with a clamp-on (non-invasive) ultrasound Doppler sensor to determine slug flow hydrodynamic parameters. The Hilbert-Huang transform (HHT) is applied to the ultrasound signals to investigate a non-invasive measurement to the parameters of the two-phase slug flow (Huang et al., 1998). The clamp-on ultrasonic sensor measurement is validated by a synchronized measurement of the slug flow parameters using a pair of flush mounted conductivity probes, separated with a distance of 0.186m located at  $x/D = 166$ . The ultrasonic device is located midway between the pairs of the probes at  $x/D = 168$ . Experimental conditions covered included most parts of the slug flow regime. The liquid superficial velocity ranged between 0.4 to 2.0 m/s while the gas superficial velocity ranged between 0.05 to 2.7 m/s. The ultrasound signals were decomposed into the intrinsic mode functions (IMFs) using the HHT technique. The high oscillation signals were used for calculating the slug bubble velocities; low oscillation IMFs were summed up to produce a trend for each dataset. The trend is sensitive enough to detect the slug flow hydrodynamics and was analyzed simultaneously with the signals of the conductivity probes. Results are in good agreement with correlations in the literature.

## COUPLING DIFFERENT LEVELS OF RESOLUTION IN DEM SIMULATIONS

Daniel Queteschiner, Johannes Kepler University Linz, Austria; Thomas Lichtenegger, Johannes Kepler University, Austria; Stefan Pirker, Johannes Kepler University, Austria

Despite constantly growing computing power, discrete element method simulations of particulate systems can hardly reach the time and length scales required to describe industrial applications. Coarse-grained models have been developed to reduce computational costs and to make simulations of large granular systems feasible. However, these models cannot replace fine-grained simulations completely. Important details may be missed and in the worst case incorrect results are produced.

We present a novel technique for efficient multi-scale DEM simulations. The approach is designed for granular systems where full resolution is required only in spatially confined sub-regions whereas a lower level of detail is sufficient for the rest of the system.

Our method establishes two-way coupling between fine- and coarse-grained parts of the system by volumetric passing of boundary conditions. By this we are able to reach large time and length scales while retaining details of critical regions.

The method is tested by comparing the computed statistical properties of the coupled coarse-grained - fine-grained model with the corresponding properties of the fully resolved reference system.

## ON THE MOTION OF A SINGLE AIR SPOUT THROUGH A QUASI-TWO DIMENSIONAL GRANULAR BED

Ritwik Maiti, Indian Institute of Technology Kharagpur, India; Gargi Das, Indian Institute of Technology Kharagpur, India; Prasanta Kumar Das, Indian Institute of Technology Kharagpur, India

Granular aggregates can pack like a solid and flow like a fluid depending on the conditions it is subjected to. Granular flow thus exhibits a unique physics of its own and is yet a poorly understood field albeit its abundance in nature and importance in industry. The present study reports the behavior of air streams rising through a granule filled column and attempts to understand the dynamic interactions of granular aggregates with rising air. The experiments have been performed in a two dimensional channel (height 400 mm, width 120 mm and thickness 4 mm) filled with glass bead of size 400-600  $\mu\text{m}$ . Air is injected through a circular nozzle of diameter 4 mm located in the center of the channel floor. We observe that the air stream meanders through the medium and depending on the flow rate, exhibits different patterns. The shape and behavior of the air stream is a function of air flow rate, particle - particle interaction and particle - wall collisional force and the rate of fluidization depends on the rate of bubbling inside the medium. A complete understanding of air motion requires knowledge of fluid mechanics as well as gas-solid interaction mechanism.

## EFFECT OF MOLTEN STEEL FLOW IN MOLD ACCORDING TO INTENSITY AND POSITION OF ELECTRO-MAGNETIC FIELD

Sang-Woo Han, POSCO Technical Research Laboratories, Republic of Korea; Hyun-Jin Cho, POSCO Technical Research Laboratories, Republic of Korea; Seonyong Jin, POSCO, Republic of Korea; Martin Seden, ABB AB, Sweden; Sungmin Choi, POSCO, Republic of Korea

In order to manufacture the clean steel products, the occurrence rates of inclusion defects and cracks of slab produced from continuous casting process must be lowered. Molten steel flow of continuous caster mold can control the meniscus temperature at position of forming the initial solidified shell and the inclusion behavior, be affected by the port size and angle of the immersed entry nozzle, argon gas flow rate, electro-magnetic field. A number of researchers in continuous casting process have conducted a study on the effect of molten steel by the electro-magnetic stirrer using traveling magnetic field and electro-magnetic breaker using static magnetic field. This study is about the effects of molten steel flow according to intensity and position of electro-magnetic field of electro-magnetic controller that can simultaneously or separately apply the traveling magnetic field and static magnetic field. The simulation using CFD with MHD (Magneto-Hydro Dynamics) and the onsite testing methods can evaluate the behavior of inclusion according to intensity of the static magnetic field and the temperature uniformity by rotation flow at meniscus according to position of the travelling magnetic field.

## NUMERICAL INVESTIGATION OF ELECTROSTATIC EFFECTS ON PNEUMATIC TRANSPORT OF GRANULAR MATERIALS

Jun Yao, Xiamen University, China; Yanlin Zhao, China University of Petroleum-Beijing, China; Eldin Wee Chuan Lim, National University of Singapore, Singapore

Electrostatics is usually occurred in granular pneumatic conveying systems causing significant effect on its transport. However, due to its complexity and uncertainty, such effect has been always ignored. In this work, electrostatics of granules transporting in a pneumatic conveying system was considered and compared with other forces. The methodology of coupling Large Eddy Simulation (LES) with the Discrete Element Method (DEM) was applied for computational studies of pneumatic transport of granular materials through vertical pipes in the presence of electrostatic effects. The simulations showed that a thin layer of particles formed and remained adhered to the pipe walls during the pneumatic conveying process due to the effects of strong electrostatic forces of attraction towards the pipe walls. The close correspondence between particle velocity vectors and fluid drag force vectors was indicative of the importance of fluid drag forces in influencing particle behaviors. The electrostatic field strength developed during pneumatic conveying increased with decreasing flow rate due to increased amount of particle-wall collisions. Based on dynamic analyses of forces acting on individual particles, it may be concluded that electrostatic effects play a dominant role in influencing particle behaviors during pneumatic conveying through vertical pipes at low flow rates.

## INFLUENCE OF ANGULAR MOMENTUM ON THE FORMATION OF AIR-CORE INTAKE VORTICES IN CYLINDRICAL WATER TANKS

Nicolai Szeliga, Hamburg University of Technology, Germany; Daniel Bezecny, Hamburg University of Technology, Germany; Steffen Richter, Hamburg University of Technology, Germany; Marko Hoffmann, Hamburg University of Technology, Germany; Michael Schlueter, Hamburg University of Technology, Germany

The occurrence of air-entraining vortices in pump inlets of cooling water circuits is a safety hazard for the reliable operation of nuclear power plants. Vortex formation depends on various parameters, like inlet size and form, tank geometry, volume flow, submergence depth and induced momentum. In many studies that have been conducted to investigate the parameters listed previously, measures were taken to minimize the induced momentum (i. e. Jain et al. 1978), since it would impede the experiments. The experiments conducted at the Institute of Multiphase Flows are carried out to determine the influence of induced momentum on the vortex formation and stabilization. For this purpose the vortex formation in a cylindrical tank with varying angular inflow has been observed, using Particle Image Velocimetry (PIV) to measure the tangential and radial velocities on varying horizontal planes in the center of the tank. Additionally, the occurring air core lengths are recorded. Results show that a constant angular momentum not only increases the air-core formation but also stabilizes the vortex. A strong angular momentum significantly reduces the influence of surface tension and viscosity on the vortex formation in small-scale systems.

\*This work is kindly supported by the Federal Ministry of Education and Research (BMBF) under grant number 02NUK023A. The authors gratefully acknowledge this financial support.

## NUMERICAL ANALYSIS OF EROSION ON SUSPENDED HEAT TRANSFER SURFACES IN A 350MW SUPERCRITICAL CFB BOILER

Qiuyu Tu, Chinese Academy of Sciences, China; Haigang Wang, Chinese Academy of Sciences, China; Yunkai Sun, Chinese Academy of Sciences, China

For a supercritical circulating fluidized bed, with the boiler capacity increases, the cross-section area increases. In order to maintain the main circulating loop heat flux balance and constant temperature, it is necessary to install suspended heat transfer surfaces in the loop. A cold test rig of 350 MW supercritical circulating fluidized bed (CFB) has been implemented using computational fluid dynamics (CFD). The model is an industrial scale and has suspended surfaces installed on the top of the rectangular boiler and three parallel cyclone separators attached. The computational implementation is accomplished by the commercial software Fluent. The fluidization gas is modeled as continuum phase, while particles are modeled as discrete phase. Discrete particle model (DPM) is employed to represent the gas-particle interaction. Different parameters, including superficial gas velocity, particle numbers and diameters, are tested to simulate all kinds of conditions occurring in the real process, and their effects on the suspended surfaces erosion are analyzed to evaluate and optimize the surfaces location and geometry design. The effects of particle concentration, velocity and gas velocity near the surfaces on the erosion are given and analyzed.

\* Support from grant Strategic Priority Research Program of the Chinese Academy of Science (No. XDA07030100) is gratefully acknowledged.

## INTERFACE-RESOLVING SIMULATIONS OF GAS-LIQUID FLOWS IN A SOLID SPONGE STRUCTURE

Xuan Cai, Karlsruhe Institute of Technology (KIT), Germany; Martin Woerner, Karlsruhe Institute of Technology (KIT), Germany; Olaf Deutschmann, Karlsruhe Institute of Technology (KIT), Germany

Solid sponges (foams) combine large surface-to-volume ratios and good radial mixing/heat transfer with low axial pressure drop which makes them attractive packings for multiphase chemical reactors. The design and optimization of such reactors require the characterization of the flow and transport phenomena in the pores of the solid sponge. In this contribution we present interface resolving numerical simulations of the gas-liquid flow within a solid sponge. We employ a recently developed phase-field method (coupled with the single-field Navier-Stokes eq.) which was implemented in OpenFOAM® and validated for various wetting phenomena (DOI: 10.1002/ceat.201500089). The computational domain consists of a micro-CT-generated representative volume of a solid sponge which is mirrored to allow for periodic boundary conditions. Our simulations provide information for the gas-liquid interfacial area and the wetted sponge surface area for varying driving axial pressure drop and contact angle. The data are useful for characterization of the gas-liquid flow in solid sponges and for developing closure relations for CFD simulations with the Euler-Euler approach.

\*The funding by Helmholtz Energy Alliance "Energy-efficient chemical multiphase processes" (HA-E-0004) is gratefully acknowledged.

## INFLUENCE OF PARTIALLY WETTED MICRO-PARTICLES AT A CONTAMINATED INTERFACE ON COALESCENCE PROCESS.

Erik De Malmazet, EDF, France; Frederic Risso, IMFT- CNRS/Université de Toulouse, France; Olivier Masbernat, Tlmoine, France; Vincent Pauchard, The City College of New York-CUNY, USA

The combined effect of micro-particles and interface aging on coalescence of millimetre-sized water drops with an oil/water interface is studied over long times. The system is not pure and interface contamination grows with time, resulting in a slow but continuous decrease of interfacial tension over time, which is measured in situ using an original technique. Without added micro-particles, coalescence times are randomly distributed and uncorrelated to drop diameter or interfacial tension. In presence of 10 µm size hollow glass particles at the oil/water interface, coalescence times become more reproducible and show a clear dependence upon drop diameter and interface aging. Results are consistent with a classical drainage model assuming that the critical thickness scales as the micro-particle diameter, a result that tends to validate the bridging scenario. Interestingly, the film retraction speed during the coalescence process does not follow theoretical predictions in a planar geometry. High-speed imaging of the retracting film reveals that the hole rim is bending upward while retracting, resulting in a strong slowdown of retraction speed. This is caused by the difference of interfacial tension between oil/drop freshly formed interfaces and the aged oil/water interface.

### NUMERICAL STUDY OF DROPLET STATISTICS RESULTING FROM TURBULENT JET PRIMARY BREAKUP

Amirreza Movaghar, Chalmers University of Technology, Sweden; Mark Linne, Chalmers University of Technology, Sweden; Michael Oevermann, Chalmers University of Technology, Sweden; Luis Bravo, US Army Research Laboratory, USA; Alan Kerstein, , USA

Primary breakup to form drops near liquid surfaces is an important fundamental process to study since it determines initial properties of the dispersed phase such as ligament/droplet sizes and momentum. It affects mixing rates, secondary breakup, collisions, and separated flow within the dispersed flow region. Primary breakup can be regarded as one of the least developed model components for simulating and predicting liquid jet breakup, which is of paramount importance in many technical applications, e.g. fuel injection in engines and spray painting. This paper presents an investigation of droplet statistics as a result of primary breakup at the surface of a turbulent liquid jet in still air at standard temperature and pressure. We compare drop size statistics from one-dimensional turbulence (ODT) with results from large-eddy simulations (LES) using conservative Volume-of-Fluid methods simulating an injection event from a complex nozzle (Bravo et al., ICLASS 2015). ODT is a stochastic model simulating turbulent flow evolution along a notional 1D line of sight by applying instantaneous maps to represent the effect of individual turbulent eddies on property profiles resolving all time and length scales. It permits affordable simulations of high Reynolds and Weber number flow. The primary objective of this research is the validation of a recently presented extension of ODT for primary breakup (Movaghar et al., ICLASS 2015) and its capabilities to predict realistic drop size distributions from primary breakup by comparing it to LES results.

### 3D RSTM MODELING FOR PARTICULATE GRID-GENERATED TURBULENT AND SHEAR FLOWS

Alexander Kartushinsky, Tallinn University of Technology, Estonia; Medhat Hussainov, Tallinn University of Technology, Estonia; Igor Shcheglov, Tallinn University of Technology, Estonia; Sergei Tisler, Tallinn University of Technology, Estonia; Pawel Kosinski, University of Bergen, Norway; Boris V. Balakin, University of Bergen, Norway; Peep Lauk, Estonian Aviation Academy, Estonia; Karl Erik Seegel, Estonian Aviation Academy, Estonia

A novel 3D model based on RSTM closure of equations of carrier and dispersed phases was elaborated for behavior of solid particles in channel turbulent particulate flows.

The essence of the model is a rejection of the Boussinesq hypothesis, which connects the particles averaged velocity with their fluctuating velocity correlations via artificially imposed turbulent viscosity coefficients. The model is based on a direct calculation of normal and shear components of the Reynolds stresses for the dispersed phase in a similar way as for the carrier fluid.

The model was validated by investigating a turbulent dispersion of solid particles ejected from a point source into two kinds of turbulent flow: grid-generated turbulent flow and a uniform linear shear flow. Two cases of spatial orientation of the mean velocity shear were examined: in the direction of gravity and in the direction perpendicular to gravity. Numerical data on turbulent dispersion of particles and spatial displacement of maximum of distribution of their concentration show satisfactory agreement with experimental results.

\* Support from grant EMP230 is gratefully acknowledged.

### COLLECTING DEVICE OF VOLATILE ORGANIC COMPOUNDS IN WATER LIKE A NOSE: THIN LIQUID FILM CONTAINING ODORANT BINDING PROTEINS

Hitoshi Kida, Tokyo University of Agriculture and Technology, Japan; Yoshiyuki Tagawa, Tokyo University of Agriculture and Technology, Japan; Ryoichi Sato, Tokyo University of Agriculture and Technology, Japan; Masaharu Kameda, Tokyo University of Agriculture and Technology, Japan

A technique for the collection of volatile organic compounds (VOCs) in water is required for the practical use of biomimetic odorant sensors. We developed a thin film of water including odorant binding proteins (OBPs) that acts as a collecting device of vapor VOCs, similar to an animal nostril. OBPs transport hydrophobic odor molecules to olfactory receptors across the mucus or lymph. A thin liquid film was generated employing a rimming flow on the inner surface of a rotating glass cylinder. Two kinds of OBPs derived from Bombyx mori (Pheromone Binding Protein 1: PBP1, General Odorant Binding Protein 2: GOBP2) were used as additives in water. The sample gas included dilute eugenol, which simulates an explosive substance, was collected by the rimming flow device. The experimental results showed that the collection efficiency increased with the revolution rate of the cylinder. This was attributed to the spread of the liquid in the cylinder and an increase in the concentration of eugenol gas near the inside wall by centrifugal force. The collection efficiency further increased by adding both OBPs compared with an irrelative protein and PBP1 exhibited higher affinity for eugenol. Furthermore, the collection efficiency increased with the amount of OBPs.

### PARTICLE DISPERSION IN INHOMOGENEOUS TURBULENCE: AN ANALYSIS OF PDF-BASED MODELS

Christopher Stafford, Newcastle University, UK; David Swailes, Newcastle University, UK; Nilanjan Chakraborty, Newcastle University, UK

PDF equations provide a useful tool for studying the statistical distribution of particles in turbulent flow. The approach identifies two key contributions within the convective-diffusive representation of mass flux in the particle phase. These emerge from a density weighted 'body-force' term associated with statistical inhomogeneities of the turbulence, and a term contributing to the effective particle phase stress tensor. These terms are considered in the context of both homogeneous and inhomogeneous flows; the latter case being associated with the behaviour of particle-pair dynamics in homogeneous flows. Terms are evaluated using both Direct Numerical and Kinematic Simulations, and these provide reference values for assessing the performance of related closure approximations.

### LAGRANGIAN STOCHASTIC MODELLING IN LARGE-EDDY SIMULATION OF TURBULENT PARTICLE-LADEN FLOWS

Alessio Innocenti, University Pierre et Marie Curie, France; Cristian Marchioli, University of Udine, Italy; Sergio Chibbaro, University Pierre et Marie Curie, France

Large-Eddy Simulation (LES) in Eulerian-Lagrangian studies of particle-laden flows is one of the most promising and viable approaches when Direct Numerical Simulation (DNS) is not affordable. However applicability of LES to particle-laden flows is limited by the modeling of the Sub-Grid Scale (SGS) turbulence effects on particle dynamics. These effects may be taken into account through a stochastic SGS model for the Equations of Particle Motion (EPM) that extends the Velocity Filtered Density Function method originally developed for reactive flows, to two-phase flows. The underlying filtered density function is simulated through a Lagrangian Monte Carlo procedure, which consists of solving a set of Stochastic Differential Equations (SDE) along the trajectory of a particle. The resulting Lagrangian stochastic model has been tested for the reference case of turbulent channel flow, using a hybrid algorithm in which the fluid velocity field is provided by LES and then used to advance the set of SDEs in time. The consistency of the model has been assessed in the limit case of particles with very low inertia (very low Stokes number), when "duplicate fields" are available from both Eulerian and Lagrangian approaches. Tests with inertial particles have been performed focusing on particle preferential concentration and segregation in the near-wall region: upon comparison with DNS-based statistics, our results show improved accuracy and considerably reduced errors with respect to LES with no SGS model in the EPM.

### MULTIPHASE INVESTIGATION OF INTENSIVE QUENCHING WITH WATERJETS

Steffen Waldeck, Foundation Institute of Material Science, Germany; Udo Fritsching, Foundation Institute of Material Science, Univ. Bremen, Germany

In the heat treatment of metallic materials water jets are often used for intensive cooling. The workpieces are quenched from high temperatures to get the required process results and material or component properties. An important requirement of the cooling process is the maintenance of a spatially and temporally predefined course of heat transfer in order to avoid distortions and material damage. For the design of the cooling process with free water jets numerical simulations are used to predict the cooling history and reduce costs for experiments significantly. The challenge for the numerical simulation of heat transfer in boiling fluids is the modeling of the physical changes at the phase boundaries. A modified Euler-Modell is designed, in which the vapor film is build by a dense accumulation of vapor bubbles. The vapor phase is assumed to be a dispersed system. For the numerical simulation of quenching processes, source codes with several "User-Defined Functions" are implemented in ANSYS FLUENT, to describe the heat and mass transfer for the various boiling phenomena and to calculate the conservation equations for mass, momentum and energy for the phase transition. The mass flow and the biphasic flow structure are resolved locally and time-dependent. In this model, a relatively large grid spacing and time steps are sufficient for the numerical simulations. In this way also the quenching of complex geometries with free water jets can be simulated. The multiphase flow boiling model is used to analyse the quenching process of single and multiple interacting water jets (3D).

#### A SIX-EQUATION TWO-PHASE MODEL IN WENO SCHEME FOR PHASE TRANSFORMATION FRONTS

Wangxia Wu, Tsinghua University, China; Bing Wang, Tsinghua University, China; Wenbin Zhang, Tsinghua University, China

In this work, we model two-phase compressible flow with a six-equation single velocity model. This is a pressure non-equilibrium model so that each phase has its own pressure and temperature with a single velocity for both phase. We use this stiff velocity relaxation model simulate phase transformation fronts include heat and mass transfer with several relaxation steps. A modified Godunov-type scheme with a fifth-order weighted essentially non-oscillatory (WENO) reconstruction and a Harten-Lax-van Leer contact (HLLC) approximate Riemann solver is used for the solution of the hyperbolic part of the model. The instantaneous pressure relaxation processes are taken for mechanical relaxation. Hence it can ensure better features than reduced five-equation models which assume mechanical equilibrium initially for numerical approximations of non-conservative terms. Then temperature and Gibbs free energy relaxation are implemented respectively by the assumption that the mechanical properties relax much faster than the thermal properties. As thermal relaxation is only considered at phase transformation fronts, metastable states are allowed to appear far from these fronts. A number of numerical examples are provided and shown this method has a reasonable agreement with both the exact solutions and the experimental data.

#### DROP IN DROP DIAMETER FROM A VIBRATING CAPILLARY TUBE

Herve Duval, CentraleSupélec, Université Paris-Saclay, France; Aude Bertrandias, CentraleSupélec, Université Paris-Saclay, France; Marie-Laurence Giorgi, CentraleSupélec, Université Paris-Saclay, France

Whereas the impact of longitudinal vibrations on membrane emulsification (ME) is well documented, the effect of transversal vibrations (i.e. perpendicular to the membrane surface) has been less discussed. In this case, Arnaud (EP 1 551 540 B1) found that smaller drops with a narrower size distribution were generated. In order to understand the effect of transversal vibrations on emulsification, we focused on a simplified configuration where a single drop is formed from a vibrating capillary tube immersed in a stationary surrounding liquid. The effects of various parameters were investigated including the frequency and amplitude of vibration, the dispersed phase flow rate, the capillary inside diameter, the surface tension and both phase viscosities.

At set forcing frequencies, significantly smaller drops (up to 76% lower in diameter) are generated once a threshold amplitude is reached. A drop grows and resonates in first mode as soon as its resonance frequency and the forcing frequency coincide. During resonance, the drop detaches prematurely in stretching mode if its elongation ratio reaches a critical value function of the drop to pore diameter ratio. Otherwise, the drop grows further, leaves the first mode resonance range and detaches at a larger size in dripping mode.

The effects of the various parameters on drop size and amplitude threshold are well described by a harmonic oscillator model where the bound drop eigenfrequency is given by Strani & Sabetta's calculations and the damping term is modified to account for additional friction.

#### WALL EFFECT ON THE DYNAMICS OF PARTICLE MOTION IN A LAMINAR FLOW

Zhi-Gang Feng, University of Texas at San Antonio, USA; Jason Gatewood, University of Texas at San Antonio, USA; Yifei Duan, University of Texas at San Antonio, USA; Efsthios Michaelides, TCU, USA

This study examines numerically the effect of the presence of a plane wall on the motion of particles. The dynamics of particle motion are numerically studied using a resolved discrete particle method (RDPM). The particle Reynolds numbers are in the range 0.1 to 100. In the case when the particle moves parallel to the wall at a constant velocity, the drag and lift on the particle are numerically determined. Another case is when the particle settles near a wall. The migration velocity of the particle toward the wall is investigated. Observation on the wakes formed behind the particles are also reported. The simulation results at small Reynolds number are compared with experimentally measured data found in the literature as well as with the analytical solutions to this problem. It is observed in all the numerical computations that the two principal parameters that determine the effect of the plane wall on the motion of the particles are: a) the distance between the particle and the wall; and b) the particle Reynolds number.

#### EFFECT OF THE VAPOR FLOW ON THE DROP SPREADING AT THE IMPACT IN THE LEIDENFROST BOILING

Guillaume Castanet, Université de Lorraine, CNRS, France; Ophélie Caballina, Université de Lorraine, CNRS, France; Fabrice Lemoine, Université de Lorraine, CNRS, France

When droplets impact solid surfaces at high temperature, a vapor layer develops quickly between the liquid and the solid substrate (Leidenfrost effect). For modeling the deformation process undergone by the impinging droplet, one challenge is to take into account the frictions exerted on the droplet by the vapor flowing in the film. In the present study, the deformation of droplets impinging a smooth solid surface is experimentally investigated for liquids of different viscosities. The evolutions in time of the spreading diameter, the residence time, the maximum spreading diameter are determined by high-speed shadowgraphy. The experimental results are compared to a model that is derived from the mass and momentum balances applied to the rim and the lamella of the spreading droplet. Here, the momentum transferred to the rim by the liquid coming from the lamella is derived from an inviscid asymptotic solution for the flow inside the lamella. This model considers a lamella thickness which does not depend on the Reynolds and Weber numbers. Assuming that the shear stress is null at the liquid/vapor interface, the flow rate of the liquid entering the rim needs to be increased, especially for the less viscous liquids, to obtain satisfying agreements between the experiments and the numerical predictions. This suggests that the liquid at the bottom of the drops is dragged out by the vapor flow in the film.

#### KINEMATIC AND DYNAMIC CHARACTERISTICS OF A FALLING DROPLET

Konstantinos Bergeles, Imperial College London, UK; Yannis Hardalupas, Imperial College London, UK; Alex M.K. P. Taylor, Imperial College London, UK

Kinematic and dynamic characteristics of a falling droplet, as it reaches its terminal velocity, have been numerically investigated using OpenFOAM and the Volume of Fluid methodology. The droplet consisted of oil with Oh number equal to 0.044 and was injected into a stagnant environment with an initial velocity of 1m/s. The initial Weber, Reynolds numbers are 0.14 and 230 respectively. The numerical grid covering the computational domain consists of 12 106 computational cells and is translated along the gravity axis. Bounded, higher order discretization schemes along with time step control and temporal sub-cycling were used for the investigation of the flow both inside and outside the droplet. The investigation covered a length of 0.25s, in which unsteady phenomena appeared in conjunction with alternating vortex shedding. Attention is given to the motion of the intermediate vortex (rollex) between the Hill vortex and the wake of the droplet during the aforementioned alternating vortex shedding, which is missing from previous studies. The effects of the initial velocity and aerodynamic forces, as also the internal transient Hill vortex flow on the droplet oscillation are quantified. Moreover, a novel investigation of the temporal evolution of the droplet deformation, in which the centroids of the two droplet halves initially start oscillating out of phase and at latter stages in phase in a locking fashion with the vortex shedding, is also presented.

\* Support from Volvo Cars Corporation, Gothenburg is gratefully acknowledged.

#### STEERING ULTRASOUND CONTRAST AGENTS WITH ULTRASOUND USING THE PRIMARY BJERKNES FORCE

Alicia Clark, University of Washington, USA; Alberto Aliseda, University of Washington, USA

Ultrasound contrast agents (UCAs) are small micron-sized bubbles that increase the contrast in ultrasound (US) imaging due to their echogenicity. The trajectories of these microbubbles can be manipulated by the Bjerknes force, caused by the phase difference between the US pressure field and the volume oscillations of the microbubbles (Bjerknes, V. 1906). Our research focuses on quantifying the acoustic and hydrodynamic forces acting on insonified microbubbles in pulsatile flow by using high-speed imaging and a novel particle tracking algorithm. A computer-controlled pulsatile pump is used to create flow conditions similar to those found in human circulation, as characterized by the Reynolds and Womersley numbers. In our experiments, we varied the Reynolds number between 100 and 500 and the Womersley number between 1 and 4 to replicate conditions found in the intracranial vascular system. Microbubbles in complex flows, both with and without US, are imaged at high spatial (~1  $\mu$ m) and temporal (~1 ms) resolution. Image processing and particle tracking are used to extract quantitative information on the behavior of the microbubbles. We determined the coupling of ultrasound induced bubble volume oscillations with hydrodynamics forces (lift, drag, added mass, ...) from analysis of the bubble trajectories under a well-characterized carrier flow, given by a Womersley velocity profile experimentally measured using PIV. Nonlinear lift behavior dominates the trajectories in high shear regions, in competition with the Bjerknes force.

## PARTICLE RESUSPENSION BY A PERIODICALLY-FORCED IMPINGING JET

Giovanni Soligo, University of Udine, Italy; Wen Wu, Queen's University, Canada; Francesco Zonta, University of Udine, Italy; Cristian Marchioli, University of Udine, Italy; Alfredo Soldati, University of Udine, Italy; Ugo Piomelli, Queen's University, Canada

When hovering near arid grounds, the rotor of helicopters generates a downward jet that interacts with the soil and induces resuspension of sand grains, dust or dirt. This phenomenon, called rotary-wing brownout (RWB), can be extremely dangerous because it drastically reduces the pilot's visibility during landing or takeoff. To mitigate the impact of RWB, a detailed knowledge of the turbulent flow field generated by the recirculating rotor downwash must be accompanied by a detailed understanding of the particle resuspension mechanisms.

In this work, we investigate these mechanisms using a numerical approach that combines Large Eddy Simulation of the flow field with Lagrangian tracking of the particles. The impinging jet generated by the helicopter is modeled as a sequence of periodically forced azimuthal vortices. Particles are initially placed on a thin layer just above the ground. Depending on particle size and inertia, we determine whether or not particle resuspension occurs. In case of resuspension, we characterize from a statistical point of view the local interaction between particles and turbulence, as well as the tendency of particles to segregate and concentrate preferentially in specific flow regions. We also clarify the role of shear stress and vorticity, which are generated by the jet vortices when they impinge on the ground, on the inception of particle resuspension.

## MODIFICATION OF WALL TURBULENCE VIA THE EXACT REGULARIZED POINT PARTICLE APPROACH

Roberta Messina, Università Roma La Sapienza, Italy; Francesco Battista, Sapienza Università di Roma, Italy; Paolo Gualtieri, Sapienza Università di Roma, Italy; Carlo Massimo Casciola, Sapienza Università di Roma, Italy

Many technological and environmental applications involve particle laden flows e.g. pipelines or medical spray. The aim of this work is to understand how the presence of the particles modify the carrier turbulent flow. In the literature it is possible to find different numerical simulations of turbulent pipe and channel flows. However, the ensuing results are scattered and it is not possible to draw a definite picture of these kind of flows in presence of a substantial inter-phase momentum coupling. In the present contribution the inter-phase momentum coupling is achieved via the Exact Regularized Point Particle (ERPP) approach. Several simulation are performed in the geometry of the pipe flow for different values of the particle Stokes number and mass loading. For all cases the same pressure gradient of the uncoupled case (reference case) is used. Hence any effect of the particles on the carrier flow results in a modification of the mass flow rate. In fact, the presence of the particles substantially modify the turbulence as documented by the mean velocity profile, the mean momentum equation and the budget of the turbulent kinetic energy. A preliminary analysis shows an overall drag increasing and an attenuation of the turbulent fluctuations in the bulk while new peaks of the velocity variances appear in the near wall region.

## SCAVENGING OF SOLUBLE ATMOSPHERIC TRACE GASES DUE TO CHEMICAL ABSORPTION BY RAIN DROPLETS IN INHOMOGENEOUS ATMOSPHERE

Tov Elperin, Ben-Gurion University of the Negev, Israel; Andrew Fominykh, Ben-Gurion University of the Negev, Israel; Boris Krasovtsov, Ben-Gurion University of the Negev, Israel

We analyze effects of irreversible chemical reactions of the first and higher orders and chemical dissociation on the rate of trace gas scavenging by rain in the atmosphere with non-uniform latitudinal concentration and temperature distributions. We employ a one-dimensional model of precipitation scavenging of chemically active soluble gaseous pollutants that is valid for small gradients of temperature and concentration in the atmosphere. It is demonstrated that transient altitudinal distribution of concentration under the influence of rain is determined by the partial hyperbolic differential equation of the first order. Scavenging coefficients are calculated for wet removal of chlorine, nitrogen dioxide and sulfur dioxide for the exponential and linear initial distributions of trace gases concentration in the atmosphere and linear and uniform temperature distributions. Theoretical predictions of the dependence of the magnitude of the scavenging coefficient on rain intensity for sulfur dioxide are in a good agreement with the available atmospheric measurements.

Reference: Elperin, T., Fominykh, A., Krasovtsov, B., (2015) Journal of Environmental Radioactivity, 143, 29-39.

## SEDIMENT-LADEN FRESH WATER ABOVE SALT WATER: STABILITY ANALYSIS AND NONLINEAR SIMULATIONS

Eckart Meiburg, UC Santa Barbara, USA; Peter Burns, UC Santa Barbara, USA

When a layer of particle-laden fresh water is placed above clear, saline water, both double-diffusive and Rayleigh-Taylor instabilities may arise. We investigate this configuration by means of linear stability analysis and three-dimensional direct numerical simulations. The simulations show that the presence of particles with a Stokes settling velocity modifies the traditional double-diffusive fingering by creating an unstable 'nose region' in the horizontally averaged profiles, located between the upward moving salinity and the downward moving sediment interface. The effective thickness of the salinity (sediment) interface grows diffusively, as does the height of the nose region. The ratio of nose thickness to salinity interface thickness initially grows and then plateaus, at a value that is determined by the balance between the flux of sediment into the nose region from above, the double-diffusive/Rayleigh-Taylor flux out of the nose region below, and the rate of sediment accumulation within the nose region. For small ratios, double-diffusive fingering dominates, while for larger values the sediment and salinity interfaces become increasingly separated in space and the dominant instability mode becomes Rayleigh-Taylor-like. A scaling analysis based on the results of a parametric study indicates that the thickness ratio is a linear function of a single dimensionless grouping that can be interpreted as the ratio of in- and outflow of sediment into the nose region.

## PARTICLE FLOW IN GAS OR LIQUID - WHAT DO THEY HAVE IN COMMON?

Haim Kalman, Ben-Gurion University of the Negev, Israel

Particles flow in gas, Pneumatic conveying (PC) and particles flow in liquid, hydraulic conveying (HC) have significant applications in various industries. PC and HC differ only by the type of carrier fluid: one is compressible, the other is not. Even so, they are considered very different and have attracted different research groups rarely having a dialog. One of the main objectives of our research is to compare both systems and find common tool to analyse both PC and HC. The targeted topics are: particle final velocity, primary and secondary acceleration length, primary acceleration pressure drop, straight and bend pipe bend pressure drop. PC is commonly analyzed by showing a measured phase diagram or state diagram (Zenz diagram, which presents the steady-state pressure drop per length of pipe versus the superficial gas velocity. Each line presents a constant particle mass flow rate. HC is basically divided into two kinds of flows. For fine powders (sizes smaller than 40-70 micron) the flow is homogeneous (also called non-settling flow) and is treated as single-phase flow, probably non-Newtonian, with mixture properties. This kind of flow is not discussed nor mentioned in PC. Does it exist for very fine powders? For larger particles of HC the flow is called heterogeneous flow or settling flow where the flow is treated as two-phase flow as for PC. To better compare results of pressure drop on straight pipes we designed a new feeder for HC based on PC technology, thus, a rotary valve feeder. This enabled to control the particle mass flow rate and measure the liquid superficial velocity. We will show some preliminary results and analysis, presenting empirical relationships to include both PC and HC data for pick-up velocity, final particle conveying velocity and horizontal pipe pressure drop.

## AN UNIVERSAL BINARY FORCE RELATION ON A PARTICLE THAT ACCOUNTS FOR THE INFLUENCE OF NEAREST NEIGHBORS

Georges Akiki, University of Florida, USA; S Balachandar, University of Florida, USA; Thomas L. Jackson, University of Florida, USA; Yash Mehta, University of Florida, USA; Christopher R. Neal, University of Florida, USA; Siddarth Thakur, University of Florida, USA

The traditional approach in Euler-Lagrange point-particle simulations is to model the quasi-steady force on each particle in terms of particle Re and local particle volume fraction. Similarly added-mass and history forces are modeled in terms of relative acceleration and local volume fraction. Such models do not take into account the precise arrangement of all the particles in the neighborhood. Thus, as long as local particle volume fraction is the same the predicted force on the particle will be the same irrespective of whether the neighboring particles are mostly upstream, downstream or on the side. Here we present a new approach where we go beyond the standard "unary" approach. In this "binary" approach, not only the effect of the mean macroscale flow on the particle, but also the effect of the nearest neighbors, is taken into account. Direct numerical simulations of flow over a bed of finite-sized particles, at varying Reynolds numbers and volume fractions is used to develop and test this model. We plan to perform DNS and point-particle simulations with this binary model, as well as conventional unary model, and compare the results.

\* This work was supported by the U.S. Department of Energy under the Predictive Science Academic Alliance Program (Contract N. DE-NA0002378) and by the Defense Threat Reduction Agency (Basic Research Award N. HDTRA1-14-1-0028 to University of Florida).



## DNS OF TURBULENT CONVECTIVE FLOW BOILING IN A CHANNEL AT $Re_{\square}=400$

Daniel Caviezel, ASCOMP AG, Switzerland; Karol Swiderski, ETH Zurich, Switzerland; Chidambaram Narayanan, ASCOMP AG, Switzerland; Djamel Lakehal, ASCOMP AG, Switzerland

This paper presents first results of Direct Numerical Simulation (DNS) of subcooled water boiling in a channel at  $Re_{\square}=400$  (representing a case of a high mass flow-rate of 1250 Kg/s), where a part of the channel is heated with a constant heat flux, in the same setup as in the experiment of Phillips, Buongiorno and McKrell (MIT). Simulations were performed on 30 million grid cells, using about 2'000 parallel cores of the EU PRACE HPC clusters. Bubbles are generated by means of a novel boiling sub-grid model based on wall temperature, developed by the group, and then tracked using level sets from the wall cell to the core flow, where they condense. Turbulence is first generated upstream of the heated domain in an isolated channel flow then coupled with the domain of interest. Bubble nucleation is seen to occur in specific locations between the thermal streaks formed at the wall, where local temperature exceeds saturation. Heat transfer coefficients compare very well with the data for a range of heat flux values, varying from 100 to 1200 KW/m<sup>2</sup>, covering boiling and non-boiling scenarios. The resulting DNS flow database is being scrutinized to provide up-scaled closure models in phase-average simulations.

## STUDY OF BUBBLY FLOW THROUGH A PERFORATED PLATE USING A TWO-FLUID MODEL AND A DISCRETE ELEMENT METHOD

Carlos Pena Monferrer, Universitat Jaume I, Spain; Raul Martinez Cuenca, Universitat Jaume I, Spain; Jose Luis Munoz Cobo Gonzalez, Universitat Politècnica de València, Spain; Sergio Chiva, Universitat Jaume I, Spain

Many industrial applications involving two-phase flow systems require the use of filters or grids to manage its processes. Despite the existence of numerous numerical studies related to two-phase flow, bubbly flow through obstacles with holes or narrow gaps have not been carefully studied and analyzed. In this study two approaches at different resolution level as Two-Fluid Model (TFM) or a Discrete Element Method (DEM) coupled with the continuum phase are evaluated to analyze the appropriateness of these methods. For this purpose, the open-source package OpenFOAM has been used to compute the bubbly flow through a perforated plate. TFM has been used traditionally for engineering applications to predict two-phase flow and validated against experimental data in scenarios such as bubble columns or pipes. However this approach fails when the holes presented in the geometry are smaller than certain diameters of the bubble population size and the dispersed phase passes through them unrealistically. The same scenario is computed with DEM, which calculates the motion of each bubble and takes into account the dispersed phase blockage of the perforated plate. Average void fraction and velocity distributions across and downstream of the obstacle are discussed and compared with experimental results and finally a new implementation to the TFM is proposed in order to treat these scenarios properly.

## TRANSPORT AND HYDRODYNAMIC PHENOMENON IN A VIBRATING BUBBLE COLUMN

Shahrouz Mohagheghian, Oklahoma State University, USA; Brian. R Elbing, Oklahoma State University, USA

Simplicity, low operational cost and broad applications make bubble columns a valuable test apparatus for studying multiphase flows. Industrial applications, such as the production of synthetic fuels in the Fischer-Tropsch process, focused much of the work with bubble columns on mass transfer and controlling gas holdup. It was discovered in the 1960's that vibration increases the mass transfer rate. This study aims to understand the effect of vibration on mass transfer rate and bubble dynamics in an air-water system. The experimental column was fabricated from cast acrylic for clear optical access with an inner diameter of 10 cm and length of 122 cm. A custom vibrating table that could independently vary the vibration frequency and amplitude was designed and manufactured. The quality of vibrations was validated with an accelerometer over a range of frequencies (10-30 Hz) and amplitudes (1-10 mm). Gas flow rate was measured with a rotameter with a range of 0.4-5.0 l/min. Gas holdup and bubbles size distributions were measured using a quantitative flow visualization system. Validation of the experimental setup was performed by comparing gas holdup and bubble size distributions in the non-vibrating case with established data in the literature.

## EXPERIMENTAL INVESTIGATIONS ON THE LIFT FORCE FOR TURBULENT FLOWS WITH LOW MORTON NUMBER

Thomas Ziegenhein, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Dirk Lucas, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany

The lateral lift force has an important influence on the gas distribution in bubbly flows. For this reason reliable closure models reflecting this force are required for CFD-simulations of bubbly flows. In Direct Numerical Simulations as well in experimental investigation it was shown that the lift force strongly depends on the bubble size and even changes its sign depending on the bubble size. Tomiyama et al. (2002) obtained a correlation from experiments with single bubbles in a linear laminar shear flow for high Morton number systems, which is frequently used in CFD-simulations. In this work the lift force is determined experimentally in low Morton number systems with a turbulent background flow. Single bubbles move through a linear shear field generated in a flat column by asymmetric aerating. An averaged bubble trajectory is obtained from a long-term averaged gas volume fraction field along which the force balance including buoyancy, drag, virtual mass and lift is solved to determine the lift force coefficient. The additional parameters required, as relative velocity are obtained from the experiments. The dependency of the lift force coefficient on the horizontal bubble diameter is in good agreement with the data obtained by Tomiyama et al., however the Weltek correlation for the aspect ratio seems to be not valid for the pure system considered.

Tomiyama, A., Tamai, H., Zun, I., Hosokawa, S., 2002. Transverse migration of single bubbles in simple shear flows. Chem. Eng. Sci. 57, 1849-1858.

## MOTION OF SURFACTANT CONTAMINATED BUBBLES IN HOMOGENEOUS SHEAR TURBULENCE

Mitsuru Tanaka, Kyoto Institute of Technology, Japan

Fully resolved simulations are performed on the motion of spherical bubbles in homogeneous turbulence subject to a vertical flow that is uniformly sheared in the horizontal direction in order to understand the effect of turbulence on the transverse migration of bubbles. Bubbles are assumed contaminated by surfactants. An immersed boundary method is adopted to represent a spherical particle. A method which enables an efficient simulation of very light particles is introduced. We focus on the bubbles whose diameter is about 0.4mm and slightly smaller than those of vortex cores in turbulence. In addition to the lift force induced by the mean shear, turbulence plays significant roles in bubble transverse migration. Numerical results show that the mean transverse migration velocity in a turbulent shear flow is much lower than that of the laminar flow. Bubbles trapped in longitudinal vortex tubes, which are inclined in the transverse direction from the vertical direction, tend to rise along them. This contributes to the reduction of the transverse migration velocity. Other mechanisms are discussed, comparing with those in the case of small clean bubbles, in which the direction of the mean transverse migration is reversed in turbulence (J. Phys. Soc. Jpn. 82, 044401 (2013)).

## VISUALIZATION STUDIES ON THE SPRAY COOLING BEHAVIOR OF THE MICROMETRIC MULTIPLE SUCCESSIVE DROPLETS ABOVE THE LEIDENFROST TEMPERATURE IMPACTING ONTO HOT INCLINED SURFACES

Deendarlianto Deendarlianto, Faculty of Engineering, Gadjah Mada University, Indonesia; Yasuyuki Takata, Faculty of Engineering, Kyushu University, Japan; Masamichi Kohno, Faculty of Engineering, Kyushu University, Japan; Sumitomo Hidaka, Faculty of Engineering, Kyushu University, Japan; Takaaki Wakui, Faculty of Engineering, Kyushu University, Japan; Indarto Indarto, Gadjah Mada University, Indonesia

In metal industry, it is considered that the manipulation of solid roughness of the spray cooling will increase the quality of metal products and reduce the energy consumption. However, under practical conditions, the dispersion of the liquid results in the generation of numerous droplets that collectively can be difficult to study systematically. Here, the effect of surface roughness on the collision dynamics and heat transfer phenomena of multiple water droplets impacting upon heated inclined solid surfaces above the Leidenfrost temperature were studied experimentally. The surface roughness was ranged from Ra 0.04 up to Ra 10. The Droplet size was 500 (μm), and the substrate temperature decreased from 500 (°C) to 100(°C). As a result, it was found that the surface roughness has a significant effect on the spray cooling. The higher the surface roughness, the higher solid-liquid contact time during the multiple droplet impacting on the hot inclined wall, therefore, it will decrease the cooling time of a hot inclined wall.

## IMPLEMENTATION AND DEMONSTRATION OF A BOUNDARY CONDITION WALL FUNCTION FOR DIRECT PRECIPITATION FOULING CFD MODELLING

Sverre Gullikstad Johnsen, SINTEF Materials and Chemistry, Norway; Stein Tore Johansen, SINTEF, Norway; Bernd Wittgens, SINTEF Materials and Chemistry, Norway

In a previous paper we developed a generic modelling framework, for the diffusive mass transport through the turbulent boundary-layer of multi-component, reactive flows where one or more species may deposit on the wall. In the current paper, we describe the implementation of the modelling framework as a mass transfer wall function for coarse grid CFD modelling of direct precipitation fouling. The wall-function is formulated as a sub-grid model, where the simplified 1-dimensional governing equations are solved perpendicularly to the wall. This enables us, in principle, to model complex heat-exchanger geometries in two or three dimensions, since we reduce the computational cost of resolving the fine length-scales at which the boundary layer mass-transfer is determined. The mass-transfer wall function is implemented as a user-defined function (UDF) in the commercial CFD software ANSYS Fluent. To demonstrate the capabilities of the wall-function, it is employed on an axisymmetrical reactor geometry wherein a semi-artificial, 4-component, ideal mixture of perfect gases flows. One of the components, namely the reaction product of two of the other components, can sublime and deposit on the walls of the reactor.

\* Support from the Research Council of Norway and The Norwegian Ferroalloy Producers Research Association (FFF, 2015), through the SCORE project is gratefully acknowledged.

## MEASUREMENTS AND VISUALIZATION DURING CONDENSATION INSIDE A 3 MM DIAMETER CHANNEL

Marco Azzolin, Università degli Studi di Padova, Italy; Stefano Bortolin, Università degli Studi di Padova, Italy; Davide Del Col, Università degli Studi di Padova, Italy

In the last years, convective condensation has been extensively studied inside large tubes (e.g. 8-12 mm diameter) and two main mechanisms have been identified: shear stress dominated condensation (annular flow) and gravity force dominated condensation (stratified flow). Recent investigations carried out in mini- and micro-channels evidence the less and less important role of gravity force when decreasing channel dimensions, with a reduction of the stratified zone, an enlargement of the intermittent region and a sensible effect of the channel shape. An investigation in the macro-to-micro transition region, that includes both heat transfer, pressure drop and flow visualization, is currently missing. From this background, a new test section for the study of convective condensation inside a 3.4 mm diameter circular channel has been designed and built. The test section allows quasi-local heat transfer coefficient measurements and flow visualization. Prior to perform two-phase tests, the experimental technique has been validated with single-phase heat transfer measurements. Condensation experiments have been run at 40°C saturation temperature and the effects of mass velocity and vapor quality are studied. Flow pattern has been investigated with a high speed camera. Finally, the new experimental data are compared with available correlations in the literature to assess differences and similarities of the phenomena with larger and smaller channels.

\*The authors acknowledge the financial support of the European Space Agency through the MAP ENCOM-2 (AO-2004-096) program.

## THE PROPAGATION AND DEPOSITION OF A FINITE DRY GRANULAR MASS DOWN A ROUGH INCLINE

Keng Lin Lee, National Taiwan University, Taiwan; Fuling Yang, National Taiwan University, Taiwan

This work presents a theoretical analysis on the propagation and arresting process of a 2D finite granular mass in shallow configuration down a rough incline. The coherence-length constitutive model proposed by Ertas and Halsey (2002) is adopted to solve the bulk motion and local coherence length scale,  $l(x,t)$ , that characterizes granular cluster size which reflects spatial correlation between constituent particles. Under a priori assumption that deposition is negligible at early time of avalanche, flow depth profile,  $h(x,t)$ , turns out to be governed by an advection-diffusion equation and is solved using the matched asymptotic method under shallowness, which also determines a flow front trajectory,  $x_f(t)$ . The solutions reveal  $l(x,t) < h(x,t)$  in the front indicating the clusters can move freely and transport momentum flux in a flowing bulk. The trend of  $l(x,t)$  shows monotonic growing and becomes comparable to  $h(x,t)$  upstream, indicating clusters transmit basal decelerating impulse to decelerate the flow, giving rise to rear deposit. The critical location where  $l(x,t) \sim h(x,t)$  is solved to the leading order to determine a deposition front trajectory,  $x_d(t)$ . Under the constraint of conserved total mass, finite run-out distance,  $L_d$ , and arrested time,  $T_d$ , are estimated and used to construct a modified flow front,  $x_{fm}(t)$ , that incorporates the effect of rear deposition. The prediction of  $x_{fm}(t)$  is in a good agreement with the experimental data reported in Pouliquen and Forterre (2002).

## 3D SIMULATION OF NUCLEATE POOL BOILING USING DIFFUSE INTERFACE MODEL

Takuma Tanimoto, Mie University, Japan; Koichi Tsujimoto, Mie University, Japan; Toshihiko Shakouchi, Mie University, Japan; Toshiyuki Ando, Mie University, Japan

In various industrial applications, boiling flows generally appear, however, it is difficult to accurately predict using numerical simulations. For multiphase flow computations including the phase change, the interface between the different phases is tracked using several numerical schemes, such as the VOF method, front tracking method, level set method and MARS method. Recently as the interface tracking scheme, a diffuse interface model (DIM) including the Cahn-Hilliard equation is too much attention paid in the multiphase flow computations. In the present study, we propose the numerical scheme in which the DIM is used to track the interface; a temperature recovery method is introduced to represent the phase change. Further we examine how a micro-layer model on a wall affects the prediction of heat transfer performance. Compared to the experimental results such as the frequency of bubble departure, the diameter of bubble departure and the boiling curve, we demonstrate that the pool boiling is quantitatively reproduced using the appropriate micro-layer model.

## EULERIAN SIMULATION OF 3D FREELY-FALLING GRANULAR JET

Francois Audard, IMFT, France; Pascal Fede, INP Toulouse, France; Emmanuel Belut, INRS, France; Jean-Raymond Fontaine, INRS, France; Olivier Simonin, INP Toulouse, France

In many situations, the handling of powder leads to important and inconvenient dust emissions which are generally harmful for the health and safety of workers. In order to understand the dust spreading mechanisms, the typical case of a free falling granular jet is investigated. In the present study, 3D numerical simulations of the experiment of Ansart et al. 2009 are performed. The computational domain takes into account accurately the different parts of the geometry, composed of a bunker, of a free-fall chamber, of a plate and of different outlets. The effect of the physical coupling between the chamber and the silo is analysed by performing simulations with and without the silo. When the silo is present, it is observed that air enters from the chamber into the silo through its outlet, which may cause the spreading of the granular jet in the chamber. It can be noticed that such an effect is also observed in experiments. The spreading of the jet leads to a decrease of particles velocity while the surrounding air is entrained into the jet core. Without the silo, the lateral dispersion seen in the case with silo and in experiments is not obtained.

## COLLAPSE OF A WATER-SATURATED GRANULAR COLUMN IN AIR

Pascale Aussillous, Aix-Marseille University, CNRS, France; Clement Nobili, Aix-Marseille University, CNRS, France

We study experimentally the collapse of a granular column saturated with water. The granular column is initially stabilized by an air interface with an imposed Laplace pressure difference between the air and the interstitial fluid.

We study the dynamics of the column when the Laplace pressure difference suddenly vanishes. The collapse is initiated either by a light knock, imposing the volume of water constant, or by imposing a constant positive fluid pressure.

Similarly to the collapse of a fully immersed granular column, the morphology of the deposit is mainly controlled by the initial volume fraction of the granular mass. Different regimes are identified according to the initial packing and the way the collapse is initiated.

The initial loose packed columns give long and thin deposits with a fast dynamic that do not seem to depend on the collapse initiation. For dense packing, no motion is seen when the volume of water is kept constant, whereas the slow dynamics seem to depend on the imposed fluid pressure.

We compare the results to a depth-averaged two-phase continuum model, having a frictional rheology to describe particle-particle interactions, and taking into account the mechanisms of dilatancy, developed for the collapse of fully submerged granular columns.

# DIRECT AND INDIRECT MEASUREMENTS OF FRICTION COEFFICIENTS OF STEADY DRY GRANULAR FLOWS DOWN A ROUGH NARROW INCLINE OF ROUGH BED

Fuling Yang, National Taiwan University, Taiwan; Chiu Tingyen, National Taiwan University, Taiwan

Direct force measurement at the boundary of a steady dry granular flow down a narrow inclined chute with rough bed is conducted to investigate how basal friction coefficient,  $\mu_B$ , varies with flow bulk inertial number  $IB$  estimated by volume flow rate and flow height. Over  $IB < 0.2$ , the  $\mu_B$ - $I$  relation conforms to an almost linear relation reported in the literature. The corresponding wall friction coefficient,  $\mu_w$ , is also measured to show a depth-weakening profile and changes with  $I$  and a  $\mu_w$ - $I$  relation similar to  $\mu_B$ - $I$  is proposed. In addition, we also exploit high-speed imaging technique to achieve indirect measurement of internal friction coefficient,  $\mu$ , via a two-dimensional control volume analysis using the measured  $\mu_w$  and the depth profiles of solid volume fraction and non-Bagnold velocity,  $u(y)$ , at the wall. When this internal  $\mu$  is examined with respect to local  $I$  estimated via local slope of  $u(y)$ , it shows a similar linear trend with  $I$  but with greater dispersion, due to fluctuations in  $I$  loc, and of a lower magnitude. The difference between  $\mu_B$  and  $\mu$  is due to three-dimensional flow feature unresolvable from both measuring techniques.

# PERFORMANCE OF HEAVY OIL VISCOSITY REDUCER (HOVR) ON CRUDE OIL EMULSION

Darlan Bonetti, State University of Campinas, Brazil; Jorge Luiz Biazussi, State University of Campinas, Brazil; Carlos Eduardo Perles, State University of Campinas, Brazil; William Monte Verde, State University of Campinas, Brazil; Vanessa Cristina Guersoni Bizotto, State University of Campinas, Brazil; Antonio Carlos Bannwart, State University of Campinas, Brazil

This work aims to evaluate the effects of a Heavy Oil Viscosity Reducer (HOVR) in an experimental loop test for a crude oil sample (14 °API) with 20% (w/w) of synthetic marine water. A set of tests were performed to determine the viscosity, stress and shear rate of the emulsion under different flow conditions. The flow loop is equipped with two points of differential pressure and temperature measurement and a mass flow rate meter providing data for analysis efficiency. The data collected allowed analyzing the effect of concentration, temperature and shearing rate on the rheological curves. HOVR concentration of 1000 ppm, 1500 ppm and 2000 ppm at temperatures of 40 °C, 50 °C, 55 °C and 60 °C showed a significant effect, getting a maximum viscosity reduction of 90% at 40 °C with 1500 ppm. However, it was observed loss of efficiency in reducing viscosity for high concentrations. The increase in HOVR concentration (ARV) also caused changes in the rheological profile of the fluid, which acquired a dilatant behavior in concentrations above 1500 ppm. The same effect is observed when the emulsion temperature was increased above 50 °C. In high shear rate, the thermodynamic equilibrium of the HOVR located at the oil-water interface was changed, causing the observed degradation efficiency in viscosity reduction.

\* The authors acknowledge the financial support CNPq.

# A STUDY OF THE PERFORMANCE OF PILOT SCALE GRAVITY SORTERS USING A CFD-DEM COMPUTATIONAL FRAMEWORK

Ananda Subramani Kannan, Chalmers University of Technology, Sweden; Jacob Lund, Westrup A/S, Denmark; Jens Michael Carstensen, Videometer A/S, Denmark; Srdjan Sasic, Chalmers University of Technology, Sweden

Gravity sorters are utilized in the downstream processing of harvested grains in order to clean and enhance the grain quality for human consumption. In this paper, we undertake a detailed assessment into the performance of such sorters using a coupled CFD-DEM framework implemented in the OpenFOAM® environment. We look to establish and characterize the performance of gravity sorters cleaning harvested wheat at different operating conditions such as: deck tilt, fluidization conditions, deck vibrational intensity etc. Our simulations result in the identification of the optimal deck tilt and consequently the corresponding fluidization velocity and deck vibration intensity required for obtaining peak performance. A clear segregation between the light wastes present in the feed and the good product is noted in the virtual sorter, with the separation becoming poorer at steeper deck tilts. Finally, a regression model relating the operating conditions to the performance is developed in order to aid in making qualified guesses during the startup of such sorters.

\* This project is supported by the SEVENTH FRAMEWORK PROGRAMME OF EU, Industry-Academia Partnerships and Pathways (IAPP) - Marie Curie Actions. Grant no.: 324433.

# STUDY OF THE PROCESS OF ELECTROMAGNETIC IMPACT ON HIGH-VISCOSITY HYDROCARBON LIQUID IN A PIPELINE

Victor Kireev, Mavlutov Institute of Mechanics, Russian Federation; Ayrat Musin, Bashkir State University, Russian Federation; Rasul Zinnatullin, Bashkir State University, Russian Federation; Vladislav Miroshnichenko, Bashkir State University, Russian Federation

The profitability of a pipeline maintenance when highly paraffinic and high-viscosity crude oil is pumping (especially in the regions with low ambient temperature) may be sufficiently reduced. In such cases to raise the effectiveness of pumping various methods for thermal treatment of oil are used. One of the most perspective techniques to heat liquid in a pipeline is the high-frequency electromagnetic impact. A brand new scheme of the compact device for local radio-frequency heating of hydrocarbon liquid flowing through a pipeline is suggested in the present work. The results of experimental study of heat and mass transfer in high viscosity complex hydrocarbon liquid under heating by radio-frequency electromagnetic field are presented. Using numerical modeling the problem of laminar flow of high viscosity multicomponent hydrocarbon liquid in a pipeline is investigated. The model takes into account the influence of electromagnetic radiation and heat exchange with the environment. The mathematical model of the process is based on the convective heat transfer equation. The thermal effect of the electromagnetic field is considered by the introduction of volumetric heat sources. It is shown that the use of a point electromagnetic emitter can effectively heat the hydrocarbon liquid and maintain the optimal temperature regime of pumping while avoiding the undesirable effects of oil congelation and wax accumulation.

\* Support from the Ministry of Education and Science of the Russian Federation (grant #3.1251.2014/K) is gratefully acknowledged.

# CFD-PBM MODELLING OF AGGLOMERATION AND DEPOSITION IN PETROLEUM INDUSTRY

Boris V. Balakin, University of Bergen, Norway; Alexander Kartushinsky, Tallinn University of Technology, Estonia; Alex Christian Hoffmann, University of Bergen, Norway; Pawel Kosinski, University of Bergen, Norway

Turbulent flows laden with multiple reacting phases are common in, for instance, petroleum industry. The inter-phase reactions often result in formation of adhesive solid inclusions, which may lead to pipe plugging due to agglomeration and deposition of solids. The modern efforts to overcome the problem focus on prevention of the aforementioned phenomena by means of a flow management rather than a costly inhibition of the chemical reactions. The present paper describes a practical numerical model that can be used for design of petroleum lines so that the probability of plugging is reduced. The model is based on the principles of computational fluid dynamics (CFD) where the phases are treated as Eulerian continua, while the phenomena of solid phase nucleation, agglomeration and hydrodynamic fragmentation are accounted for by a population balance model (PBM). The combined CFD-PBM model accounts for the granular nature of the solid phase and used the up-to-date principles of the suspension rheology. The PBM-kernels are based on theoretical considerations on the formation and evolution of agglomerates in the multiphase flow and, contrary to the most of the population balance models reported nowadays, do not require any additional experimental determination of the kinetic constants. The model was validated against experiments on a four-phase flow of oil, water, natural gas and solid gas hydrates.

\* Support from grant EMP230 is gratefully acknowledged

# A COMPARATIVE STUDY OF TEMPORAL EVOLUTION OF AIR AND STEAM BUBBLES EMERGING OUT FROM AN ORIFICE IN A QUIESCENT WATER MEDIA

Thariq Ahmad.S, Indian Institute of Technology Kharagpur, India; Saikat Datta, Indian Institute of Technology Kharagpur, India; Sanat Kumar Singha, Indian Institute of Technology Kharagpur, India; Santanu Prasad Datta, Indian Institute of Technology Kharagpur, India; Prasanta Kumar Das, Indian Institute of Technology Kharagpur, India

Formation and evolution of a bubble or a train of bubbles from an orifice in a quiescent liquid media has a great importance in multi-phase flow and phase transformation. The bubble can be of either stable phase in case of gas bubble or metastable phase in case of vapor bubble. Using high-speed photography, a comparative analysis between the temporal variation of the evolution of air and steam bubbles emerging out from a small-diameter tube in a quiescent water media has been discussed in the present experimental study. Various intriguing patterns in the bubble shape, evolution, and merging are observed from the experiments. The evolution of steam bubbles are found to be more complicated than that of the air bubbles due to the occurrence of the vapor-liquid phase transition. Moreover, the final stable phase in the bubble evolution and the phase of the liquid as the media are the same, i.e. water, so, the steam bubble ultimately mixes in the water media and eventually disappears. However, this kind of transformation cannot be observed in case of air bubble due to the absence of phase transition throughout the process. The bubble evolution is also studied for different flow rates of air or steam.

## PHASE-FIELD SIMULATIONS OF THE BUOYANCY-DRIVEN DETACHMENT OF A WALL-BOUND PENDANT DROP

Andrea Lamorgese, University of Pisa, Italy; Roberto Mauri, University of Pisa, Italy

We investigated numerically the critical conditions for detachment of an isolated, wall-bound pendant droplet acted upon by surface tension and wall-normal buoyancy forces alone. Extending a previously developed diffuse interface model to describe three-phase contact line, we determined the critical Bond number as a function of the static contact angle. In addition, results of 3D simulations were compared to critical Bond numbers from a static stability analysis based on a numerical integration of the Young-Laplace equation. We argue that the discrepancy between our numerically determined static contact angle dependence of the critical Bond number and its sharp-interface counterpart is mainly due to the inability of the latter to describe the necking regime of drop detachment, where a sharpening of concentration gradients in the necking region produces an effective increase in (dynamic) surface tension, ultimately leading to a reduced tendency to detachment or an increase in the critical Bond number. Based on this dependence of surface tension on non-equilibrium conditions, we present a novel interpretation of phenomena such as Marangoni thermocapillary migration, diffusiophoresis and advancing / receding contact angles.

## VALIDATION OF TURBULENCE PARAMETERS AT THE INTERFACE OF HORIZONTAL MULTIPHASE FLOWS

Thomas Hoehne, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany  
Stratified two-phase flows are relevant in many industrial applications, e.g. pipelines, horizontal heat exchangers and storage tanks. The numerical simulation of free surface flows can be performed using phase-averaged multi-fluid models, like the homogeneous and the two-fluid approaches, or non-phase-averaged variants. The approach shown in this paper within the two-fluid framework is the Algebraic Interfacial Area Density (AIAD) model. It allows the macroscopic blending between different models for the calculation of the interfacial area density and improved models for momentum transfer in dependence on local morphology. A further step of improvement of modelling the turbulence was the consideration of sub-grid wave turbulence that means waves created by Kelvin-Helmholtz instabilities that are smaller than the grid size. A first CFD validation of the approach was done for an adiabatic case of the HAWAC channel. More verification and validation of the approach is necessary – more CFD grade experimental data are required for the validation.

## SCALE-UP OF MIXING PROPERTIES IN BUBBLE COLUMN REACTORS

Cecile Plais, IFPEN, France; Luca Gemello, Politecnico di Torino, Italy; Pedro Maximiano Raimundo, IFP Énergies Nouvelles and Université Grenoble Alpes, France; Ann Forret, IFP Énergies Nouvelles, France; Daniele Marchisio, Chemical Engineering, Politecnico di Torino, Italy; Frederic Augier, IFP Énergies Nouvelles, France

Bubble column reactors are used in many industrial fields such as biology, chemistry, refining, and water treatment. Resulting hydrodynamics is generally complex due to the simultaneity of different phenomena as macro-scale recirculations, instabilities, turbulence and interaction between phases. Prediction of global hydrodynamics is a complex but major issue for process modeling as it impacts strongly the performances of the considered gas-liquid contacting units. A possible way to properly identify scale-up properties is the use of CFD simulations.

To simulate large scale systems at high gas rates, Eulerian-Eulerian method constitutes a good compromise in terms of accuracy and computational time. It however requires the implementation of closure laws, such as those concerning phases interactions. In this work, different drag laws from literature have been evaluated, taking into account the relation between measured bubble diameters and relative velocities in different bubble columns. Their relevance once implemented in 3D RANS simulations is discussed.

Secondly, simulations have been performed wherein the diffusion generated by bubble induced turbulence modelled by Alm  ras et al. (2015) is implemented in addition to the diffusion resulting from the shear induced turbulence.

## TRIBO-CHARGING AND FLAME-SYNTHESIS TO CREATE SUPER-HYDROPHOBIC COATINGS WITH STRONG WATER JET RESISTANCE

Ilker Bayer, University of Virginia, USA; Eric Loth, University of Virginia, USA

A super-hydrophobic surface was created with tribo-charging and flame-synthesized nanostructured carbonaceous films to provide hierarchical roughness. However, their main drawback is that the films do not form a firm bonding with the surface upon which they are assembled, considerably reducing their real-time applications in liquid repellent surface technologies. The coating maintained sufficient texture and roughness to show remarkable resistance to saturation by high-pressure impinging water jets under normal or oblique conditions. Saturation conditions as a function of jet impingement pressure and oblique angles were interpreted using the theory of hydraulic jump due to Newtonian fluid jet impingement as well as scaling arguments developed for flow over superhydrophobic surfaces. The coatings can also be functionalized by a simple aqueous fluoropolymer treatment rendering them superhydrophobic also resistant to wetting by various oils.

## CFD SIMULATIONS OF A STIRRED-TANK REACTOR FOR MULTIPHASE SYSTEMS USING OPENFOAM

Xiaofei Hu, Iowa State University, USA; Alberto Passalacqua, Iowa State University, USA; Rodney Fox, Iowa State University, USA; Francesco Bertola, SABIC, The Netherlands

For industrial stirred-tank reactors, multiple impellers are usually preferred over single impeller for better gas utilization and maintenance of homogeneity. In this work, a stirred-tank reactor constituted by a cylindrical vessel with four baffles and three impellers is numerically investigated. A three-dimensional hexahedral mesh for the system is generated using a commercial mesh-generation software Pointwise. CFD simulations are performed with the open-source CFD software OpenFOAM. Steady state single-phase (liquid) simulations are performed with the realizable k- $\epsilon$  model. For gas-liquid and gas-solid-liquid systems, two-fluid model and multi-fluid model with the mixture k- $\epsilon$  turbulence model are used to solve the flow. The multiple reference frame approach is applied for all three systems to treat the rotation of impellers in the reactor. Simulation results of single-phase and two-phase systems are compared to experiments reported in the literature including flow patterns, unaerated and aerated power consumption, and overall gas holdup. Simulation results of the three-phase system, such as flow pattern and power consumption are presented. The effects of different operating conditions such as impeller rotation speed at 3.75 rad/s and 5.08 rad/s are also studied.

## INVESTIGATION OF LOCAL MIXING AND BUBBLE SIZE DISTRIBUTION IN A STIRRED BUBBLE COLUMN REACTOR

Annika Rosseburg, Technische Universit  t Hamburg Harburg, Germany; Jan Erik Schaefer, Boehringer Ingelheim Pharma GmbH & Co KG, Germany; Thomas Wucherpfennig, Boehringer Ingelheim Pharma GmbH & Co KG, Germany; Martina Berger, Boehringer Ingelheim Pharma GmbH & Co KG, Germany; Michael Schlueter, Hamburg University of Technology, Germany

For the large-scale production of pharmaceuticals, the mass transfer performance of aerobic fermentation processes is of increasing importance. Usually, mass transfer in large scale-bio processes is performed in stirred bubble column reactors that have been investigated intensively in the last decades. Despite this fact there are still some uncertainties concerning operation and scale up because most models have been derived from small-scale reactors and correlations, for instance, for the prediction of mass transfer rates, were only fitted to data on those smaller scales.

For further optimizations of large scale-reactors, a more accurate hydrodynamic characterization of large-scale gas-liquid flows is required. For this objective a large-scale reactor with optical access is used to get a deeper insight into local phenomena, such as local mixing and bubble size distribution.

First investigations show that the bubble size distribution and thus the specific interfacial area as well as the local mixing are crucial parameters for the reactor performance that can be influenced systematically.

## STUDY OF HYDRODYNAMICS AND MIXING INDUCED BY BUBBLE SWARMS IN HIGHLY VISCOUS FLUIDS

David Laupsien, University of Toulouse, France; Claude Le Men, University of Toulouse, France; Alain Line, University of Toulouse, France; Arnaud Cockx, University of Toulouse, France

Aeration is used to perform mixing of highly viscous fluids. The objective of this paper is to measure and understand macro-scale phenomena from a local analysis of bubble column hydrodynamics.

In this work, the oscillation of bubble swarms in viscous fluids in a pseudo-2D column reactor was experimentally investigated. Newtonian fluids with a viscosity in a range from 100 to 1000 times the viscosity of water were used to model bioreactor fluids. In such bubble columns a periodic flow behavior which is characterized by quasi sinusoidal trajectory of bubbles, can be observed. Large bubbles ascend in the middle while small bubbles descend along the column edges, which generate a density gradient. Vortex cells are formed at the top of the bubble column and further the phenomena of descending bubbles.

Bubble swarm behavior could be regarded for a large range of injection conditions by varying the injection type and flux. Different metrological methods were applied such as shadowgraphy, PIV as well as pressure and oxygen measurements.

A special attention was given to the frequency of the bubble swarm. This frequency could be quantitatively measured by following the mass center during the shadowgraphy, or by processing pressure, oxygen and PIV data.

## DIRECT NUMERICAL SIMULATION OF FLOW AROUND A ROTATING SPHERE AT HIGH MACH AND LOW REYNOLDS NUMBERS CONDITION FOR SUPERSONIC GAS-PARTICLE FLOW ANALYSIS

Takayuki Nagata, Tokai University, Japan; Taku Nonomura, Institute of Space and Astronautical Science, Japan; Shun Takahashi, Tokai University, Japan; Yusuke Mizuno, Tokai University, Japan; Kota Fukuda, Tokai University, Japan

In this study, direct numerical simulation of flow around a rotating sphere at high Mach and low Reynolds numbers flow is conducted in order to investigate rotation rate and Mach number effect into the fluid force coefficient and vortical structures. The simulation is carried out by solving three dimensional compressible Navier-Stokes equations directly. The free-stream Reynolds number (based on free-stream velocity, density, viscous coefficient and diameter of sphere) is set as 300, the free-stream Mach number is set to between 0.1 and 2.0, and nondimensional rotation rate defined by the ratio of free-stream velocity and maximum surface velocity is set to between 0.0 and 1.0. As a calculation result, we clarified following points: 1) the transition rotation rate of vortex structure is influenced by free-stream Mach number, 2) under the supersonic condition, the vortex structure is significantly reduced, (3) under the subsonic condition, with increasing rotation rate the drag coefficient is increased. On the other hand, under the supersonic condition, drag coefficient is reduced by means of increase of rotation rate, and (4) with increasing free-stream Mach number the change amount of lift coefficient due to rotation becomes reduced.

## A PSEUDO-DRY WET CONTACT MODEL FOR DISCRETE ELEMENT SIMULATION OF IMMERSED PARTICLE-WALL COLLISIONS

Cheng Chuan Lin, National Taiwan University, Taiwan; Fuling Yang, National Taiwan University, Taiwan

In solid-liquid flow simulations, the knowledge of how solid particles collide in viscous liquids is important and has been well studied in the sense of liquid dissipated particle kinetic energy. To address this issue, a wet coefficient of restitution, ewet, is measured and found to decrease monotonically and nonlinearly with particle Stokes number, St. Several theoretical models have been proposed to describe the immersed collision process and the ewet-St trend but precise explicit expression is yet unavailable. Different numerical simulations have also been developed but it always involves computationally expensive flow solvers that quickly consumes the computation power when multiple particles are present. Hence, this work proposes a pseudo-dry wet contact model with which an immersed wet collision process can be reproduced from a discrete element method without really invoking a flow solver. Liquid effects on dissipating particle kinetic energy, lubricating frictional contact, and modifying the contact time are embedded into the soft-sphere contact model parameters with empirical relations. The developed model is validated by direct comparison to the various ewet-St data reported in the literature.

## A STOCHASTIC MODEL FOR THE PREDICTION OF PARTICLE/PARTICLE INTERACTIONS IN THE FRAME OF THE EULER-LAGRANGE APPROACH

Boris Arcen, Université de Lorraine - LRGP, France; Anne Taniere, Université de Lorraine - LEMTA, France

The prediction of the interactions (collision and agglomeration) of inertial particles, such as solid particles and drops, is of great importance in many environmental and industrial processes. There are two main approaches to predict these interactions. The first approach, which is probably the most natural, consists in detecting the interactions between particles in a deterministic way. With this approach, the modelling uncertainties are reduced. The second approach is probabilistic and based on the knowledge of crucial statistical quantities such as the collision and agglomeration kernels. In the present contribution, we focus on the second approach, and propose a method to predict the collision and agglomeration of particles which can be used along with a stochastic particle Lagrangian tracking (Euler-Lagrange approach). We chose this framework since it is believed that complex mechanisms can be more naturally and simply implemented. The method proposed to model the particle interactions is based on the Lagrangian method given by Fox et al. (J. Comp. Phys., 2008). In the full-length paper, the method as well as its potential will be detailed. In addition, we will present several validation test cases. For instance, in order to assess the prediction of agglomeration, our numerical results will be compared to the experiment of Kim et al. (J. Aerosol Sci., 2003) and Kim et al. (J. Aerosol Sci., 2006) who measured the agglomeration of nanoparticles of NaCl in a fluid at rest, and in a stirred tank, respectively. A comparison with a QMOM method (Eulerian approach) will be also conducted.

## ANALYSIS OF SUBFILTER EFFECTS ON INERTIAL PARTICLES IN FORCED ISOTROPIC TURBULENCE

Bogdan Rosa, Institute of Meteorology and Water Management, Poland; Jacek Pozorski, Polish Academy of Sciences, Poland

Two-phase turbulent flows with the dispersed phase in the form of small, spherical particles are increasingly often computed with the Large-Eddy Simulation (LES) adopted for the carrier fluid phase, coupled to the Lagrangian tracking of the dispersed particles. An important issue, especially for particles of smaller inertia, is the modelling of subfilter fluid velocity seen by particles. This problem is of practical relevance, including an accurate prediction of particle velocity statistics and preferential concentration patterns. Our first model proposal in terms of a simple stochastic diffusion process for the subfilter fluid velocity seen by particle has further been extended to an anisotropic variant with possible correlations of subfilter flow velocity components. To enable further model development for inertial particles subject to gravity, we consider direct numerical simulations of homogeneous isotropic turbulence with a large-scale forcing. Simulation results, both without filtering and in the a priori LES setting, are reported and discussed in the paper. A full LES is also performed with the spectral eddy viscosity. Effects of the gravity field on the dispersed phase include a direction-dependent modification of the particle velocity variance and changes in the average settling velocity due to preferential sweeping. The filtering of the fluid velocity, performed in spectral space, is shown to have a non-trivial impact on these quantities.

\* The work has been supported by the National Science Centre (NCN, Poland) through the research project 2011/03/B/ST8/05677.

## NUMERICAL SIMULATION OF A SETTLING ICE BALL WITH COUPLED HEAT AND MASS TRANSFER

Agathe Chouippe, Karlsruhe Institute of Technology, Germany; Michael Stumpf, Germany; Markus Uhlmann, Karlsruhe Institute of Technology (IfH), Germany; Jan Dusek, University of Strasbourg, France; Alexei Kiselev, Karlsruhe Institute of Technology, Germany; Thomas Leisner, Karlsruhe Institute of Technology, Germany

We investigate the settling of an ice particle falling in a fluid composed of a mix of dry air and water vapor. We will focus on the fundamental configuration of a heavy spherical particle with fixed diameter in order to study possible freezing of water vapor in the wake, with direct application to the process of rain formation in clouds. We employ a numerical approach based on simulation of the Navier-Stokes equation with the spectral/spectral-element method of Jenny and Dušek (2004) for the coupling of particle motion and pressure-velocity field, while temperature and vapor content are transported by separate transport equations. The influence of both temperature and vapor on the flow is accounted for according to the Boussinesq approximation through the buoyancy term in the momentum equation using a similar method as Kotouč et al. (2009). We will analyze the structure of the velocity, temperature and water vapor fields, focusing on the influence of the wake structure upon the existence of super-saturation, as well as on the modification of the wake by both scalar fields.

References:

- M. Jenny and J. Dušek (2004), Journal of Computational Physics 194, 215-232.
- M. Kotouč, G. Bouchet, and J. Dušek (2009), J. Fluid Mech. 625, 205-248.

#### EXPERIMENTAL ANALYSIS ON THE CHARACTERISTIC AND MECHANISM OF THE SECOND DOMINANT FREQUENCY OF SUPERSONIC STEAM JET

Weixiong Chen, Xi'an Jiaotong University, China; Yingchun Wang, Xi'an Jiaotong University, China; Quanbin Zhao, Xi'an Jiaotong University, China; Daotong Chong, Xi'an Jiaotong University, China; Junjie Yan, Xi'an Jiaotong University, China

Condensation oscillation of steam jet is of high importance for industrial facility. In this study, the characteristic and mechanism of the second dominant frequency of supersonic steam jet is investigated, while the steam mass flux ranges 400-900 kg·m<sup>-2</sup>·s<sup>-1</sup> and water temperature ranges 20-70°C. The second dominant frequency is proved to be generated by the separated steam bubbles based on the bubble oscillation theory. Firstly, the second dominant frequency of supersonic nozzles in different design pressure ratio decreases with the rise of steam mass flux and water temperature. Then, a dimensionless correlation is proposed to predict the second dominant frequency. The prediction agrees well with the present experimental data, the errors are within ±16%. Furthermore, the energy of the second dominant frequency increases firstly and then decreases as the axial distance increases. Meanwhile, using the formula of predecessors to forecast the energy of the second dominant frequency, the results are well corresponding to the experimental data after the peak. However, before the peak, the energy of the second dominant frequency attenuates rapidly, and the predicted value is much bigger than the experimental data because of the influence of the steam core.

#### SPATIOTEMPORALLY RESOLVED HEAT TRANSFER MEASUREMENTS IN WAVY LIQUID FILM-FLOWS FALLING OVER AN INCLINED HEATED-SUBSTRATE

Christos N. Markides, Imperial College London, UK; Heiles Baptiste, Imperial College London, UK; Richard Mathie, Imperial College London, UK; Alexandros Charogiannis, Imperial College London, UK

We present an experimental technique that combines simultaneous planar laser-induced fluorescence imaging (PLIF) and infrared (IR) thermography, and its application to the measurement of unsteady and conjugate heat-transfer in harmonically excited, thin-film flows falling over an inclined, electrically heated substrate. Spatiotemporally resolved measurements are reported for the film height, film free-surface and liquid-solid interface temperatures, and heat flux exchanged with the heated substrate. Based on this data, local and instantaneous heat-transfer coefficients (HTCs) are also presented. Our results on the correlation between the local and instantaneous HTC and film height suggest considerable heat-transfer enhancements relative to equivalent steady-flows in thinner film regions, whereas in thick film regions, the HTC appears decoupled from the film height. This behavior was linked to unsteady/mixing transport processes that are not captured by the steady-flow analysis. Furthermore, at low Re, the mean Nu corresponds to 2.5 (same as for a steady flow), while for higher inertia, both the Nu and HTC exhibit significantly enhanced values. Finally, smaller film height fluctuations were associated with higher HTC fluctuations, while the amplitude of the wall temperature fluctuations was almost proportional to the amplitude of the HTC fluctuations.

\* Support from EPSRC grants EP/J006041/1 and EP/K008595/1 is gratefully acknowledged.

#### EXPLORING THE RHEOLOGICAL BEHAVIOUR OF ATHERMAL SUSPENSIONS OF ATTRACTIVE MICROPARTICLES

Eric Murphy, Iowa State University, USA; Gilson Lomboy, Iowa State University, USA; Sriram Sundararajan, Iowa State University, USA; Kejin Wang, Iowa State University, USA; Shankar Subramaniam, Iowa State University, USA

Cement is the most widely used man-made material. In slurry form and in absence of rheological modifiers, cement is a dense suspension of attractive microparticles. At even small shear rates, e.g. 10s<sup>-1</sup>, thermal effects are small, producing aggregated systems and with Peclet numbers O(10<sup>4</sup>). Such athermal attractive systems are interesting from their connection to (and difference from) granular, colloidal, and other soft matter systems. These systems can be characterized as having overdamped collisions/long ranged hydrodynamic forces, enduring frictional contacts, and interparticle potentials. We explore the rheology of microparticle suspensions using coupled Discrete element method and Fast Lubrication Dynamics simulations, with forces obtained from immersed atomic force microscopy experiments. We find that such suspensions quickly form physical gels, where forces are transmitted from particle to particle in shear through a large particle network much like dense granular assemblies. In this talk we touch on the origins of Bingham-fluid-like behavior through analysis of extracted stresses and explain how the dynamics of fluid, attractive potentials, and friction conspire to form two-particle correlated motion that is unique to the presence of all three interactions. We compare results from shear simulations with vane rheometer experiments and show that the FLD-DEM method is capable of capturing qualitative rheological trends.

\* Support from grant NSF CMMI Grant No. 0927660 is gratefully acknowledged.

#### ENHANCED RECOVERY OF URANIUM(VI) AND EUROPIUM(III) USING IONIC LIQUIDS IN SMALL CHANNEL

Qi Li, University College London, UK; Dimitrios Tsaoulidis, University College London, UK; Panagiota Angeli, University College London, UK

Small scale separation units aim to recover materials is an efficient way and have become an attractive choice for systems involving hazardous chemicals. For the separation of uranium and transuranic elements, the small scale operations are further intensified when ionic liquids are used as diluents instead of organic solvents (e.g. n-DD). In this work Eu(III) and U(VI) in nitric acid solutions are extracted by an ionic liquid ([C4min][NTf2]) using appropriate extractants (TBP, CMPO). The separations were carried out in the plug flow regime. Important hydrodynamic features of plug flow, such as film thickness, plug length and specific interfacial area were determined by high-speed imaging. Particle image velocimetry was used to obtain the velocity fields and circulation patterns within plugs, which enhance mass transfer. Mass transfer and extraction efficiency were measured under various nitric acid concentrations, mixture velocities and residence times. The overall volumetric mass transfer coefficients were found to be between 0.02-0.5 s<sup>-1</sup>, and are significantly larger than in conventional extraction units. Computational fluid dynamics simulations of mass transfer were also performed and compared to experimental data.

#### VISCOELASTIC FLOW MODELLING FOR POLYMER FLOODING

Shauvik De, Eindhoven University of Technology, The Netherlands; Frank Peters, Eindhoven University of Technology, The Netherlands; Johan Padding, Eindhoven University of Technology, The Netherlands; J. A. M (hans) Kuipers, Eindhoven University of Technology, The Netherlands

Polymer liquids are used in the oil industry to improve the displacement efficiency of oil from a reservoir. Surprisingly, it is not only the viscosity but also the elasticity of the fluid that determine the displacement efficiency. We aim to obtain a fundamental understanding of the effect of fluid elasticity, by developing a simulation methodology for the flow of non-Newtonian fluids through porous media. We simulate a 3D unsteady viscoelastic flow through a converging diverging geometry of realistic pore dimension using computational fluid dynamics. The primitive variables velocity, pressure and extra stress are used in the formulation. Viscoelastic stress is formulated using a FENE-P constitutive equation, which can predict both shear and elongational stress. A Direct Numerical Simulation approach using Finite volume method with staggered grid is applied. A novel second order Immersed boundary method is incorporated to mimic porous media. The effect of rheological parameters on flow characteristics is studied. The simulations provide an insight into 3D flow asymmetry at higher Deborah numbers. Micro-Particle Image Velocimetry experiments are carried out to obtain further insight. These simulations present, for the first time, a detailed computational study of the effects of fluid elasticity on the imbibition of an oil phase.

\*Support from Foundation for Fundamental Research on Matter (FOM), The Netherlands is gratefully acknowledged

#### ATOMIZATION OF NON-NEWTONIAN FLUIDS USING AIRLESS SPRAYERS: EFFECT OF VISCOUS DAMPING AND SHEAR THINNING ON LIQUID SHEET INSTABILITY AND ITS BREAKUP

Hiroki Watanabe, Meisei University, Japan; Shuhei Fujimoto, National Maritime Research Institute, Japan; Masayuki Ogata, Meisei University, Japan; Ichiro Kumagai, Meisei University, Japan

Laboratory experiments on atomization of shear thinning fluids (xanthan gum water solutions) using airless sprayer were conducted in order to understand the rheological influence on the liquid sheet instability in the atomization process. Flow visualization by backlight method using high speed video camera showed the spray morphology strongly depends on both the fluid rheology (concentration of xanthan gum water solutions) and the spray pressure while the mass flux has a weak rheological dependence in our experimental parameter range. The spatial and temporal variation of the liquid film instability in the airless spray was quantitatively examined by DFT analysis. For the airless spray using the low viscosity fluids (low concentration), a wide range from low to high frequency oscillation concerning the liquid sheet instability was observed. However the low pressure atomization with the high concentration fluids causes the viscous damping of the high frequency oscillation in the liquid sheet. Our experimental results provide fruitful information to improve the efficiency and the quality in the spray painting for the ship hull.

## COALESCENCE DURING BUBBLE FORMATION FROM TWO NEIGHBOURING PORES IN A NON-NEWTONIAN LIQUID

George Oikonomou, Aristotle University of Thessaloniki, Greece; Agathoklis Passos, Aristotle University of Thessaloniki, Greece; Aikaterini Mouza, Aristotle University of Thessaloniki, Greece; Spiros Paras, Aristotle University of Thessaloniki, Greece

This work studies the coalescence of bubbles formed into a static non-Newtonian liquid from two adjacent  $\mu$ -tubes (i.d. 110  $\mu\text{m}$ ), placed 170 & 710  $\mu\text{m}$  apart. The flow in this set-up simulates, in a simplified manner, the porous sparger operation of a bubble column and it aims to elucidate the phenomena occurring during bubble formation in a non-Newtonian liquid. The effect of the gas flow rate and the distance between the two  $\mu$ -tubes on both the coalescence time and the frequency of two rising bubbles was experimentally investigated. Various Newtonian and non-Newtonian liquids were employed while air was the gas phase. A fast video recording technique was employed for the visual observations of the phenomena occurring onto the  $\mu$ -tubes while the bubble size measurements were conducted with appropriate software. The experiments revealed that the coalescence frequency of two bubbles decreases both by increasing the gas flow rate and the distance between the  $\mu$ -tubes. For non-Newtonian and very viscous Newtonian liquids practically no coalescence was observed. This observance implies that low shear rates prevail during the formation of a bubble and hence the non-Newtonian liquid has a high viscosity. Additional CFD simulation were performed and their results are in good agreement with our experimental findings.

## A WALL-FUNCTION MODEL FOR NON-BROWNIAN HYDROSOL PARTICLES DEPOSITION

Magali Dupuy, CentraleSupélec, Université Paris-Saclay, France; Arunvady Xayasenh, CentraleSupélec, Université Paris-Saclay, France; Emmanuel Waz, Constellium, France; Herve Duval, CentraleSupélec, Université Paris-Saclay, France

We numerically analyze the motion of non-Brownian hydrosol particles in the near-wall shear layer of a turbulent channel flow in order to establish a suitable wall-function model. The liquid flow-field is described using the kinematic model proposed by Fan and Ahmadi (1995) whereas the particle behavior is obtained by Lagrangian tracking. The particle dynamic equation may include the buoyancy, the drag, the pressure gradient, the added mass and the lift forces. Appropriate friction coefficients may be added in order to take into account the hydrodynamic interactions between the particles and the wall. Deposition is achieved when the particle to wall separation distance is equal to the wall roughness.

For nonbuoyant particles, direct interception is the main deposition mechanism and the deposition velocity scales as the particle diameter (in wall units) to the power of 1.7. When wall/particle hydrodynamic interactions are taken into account, the deposition velocity is significantly reduced. In that case, the scaling law is modified by a correction factor function of the roughness height to particle diameter ratio. For buoyant particles, the deposition velocity remains close to the sedimentation velocity as long as the characteristic time of the drainage driven by buoyancy force is smaller than the residence time of the particle close to the wall. These results are recast to provide a wall-function model suitable for coarse-grid CFD simulations.

## DEFORMATION AND DYNAMICS OF COMPOUND CAPSULES IN SIMPLE SHEAR FLOW

Zhengyuan Luo, Xi'an Jiaotong University, China; Bofeng Bai, Xi'an Jiaotong University, China

A compound capsule consists of an elastic capsule with a smaller capsule inside. Such a compound capsule is an excellent structure for encapsulation, transport and release of active agents in industrial and biomedical applications such as drug delivery. It can also be taken as a model system of eukaryotic cells or parasite-infected red blood cells (the inner capsule representing the cell nucleus or parasite). Therefore, to study the deformation and dynamics of compound capsules will benefit the development of relevant technologies and the understanding of the motion of biological cells in blood flow. However, the fluid mechanics of compound capsules is largely unexplored due to the complex interactions among the viscous flows inside and outside the capsule and the elastic tensions in the inner and outer membranes. As such, we developed a front tracking-finite difference-finite element method to simulate compound capsules under flow and investigated the deformation and dynamics of compound capsules in simple shear flows. The effects of the presence of inner capsule and corresponding parameters, including mechanical properties of the inner and outer membranes and volume fraction of inner capsule, on the deformation and dynamics of compound capsules are quantitatively explored.

\* Support from grant 51425603 is gratefully acknowledged.

## SIMULATION OF GASIFYING PARTICLE, INTERACTION WITH FREE INTERFACE FLUID FLOW

Zeinab Moradi Nour, KTH, Sweden; Minh Do Quang, KTH, Sweden; Gustav Amberg, KTH, Sweden

A three phase flow, liquid-gas-solid is simulated by using lattice Boltzmann method (LBM) in which the finite size solid particle is gasifying. To capture the interface, a free energy method is applied proposed by Zheng et. al. (H.W. Zheng, C. Shu, Y.T. Chew, A lattice Boltzmann model for multiphase flows with large density ratio, JCP 218 (2006) 353-371). At equilibrium state, the gasifying solid particle is surrounded by a thin film of the gas and is immersed in the bulk fluid. By implementing a uniform inlet velocity into the domain or adding gravity, the system goes toward an off-equilibrium state which induces the solid particle motion. Moreover, there is a deformation in gas interface due to the particle motion and the other exerted forces. In the present study the interaction between solid phase with gas and liquid phases is investigated. Furthermore, the effect of the interface thickness, density ratio, mass flux rate of gasification and other important factors on the system is studied.

Support from grant Swedish Research Council (VR) is gratefully acknowledged.

## BREAK UP OF CAPILLARY BRIDGES: EFFECT OF SOLUBLE SURFACTANT

Nina Kovalchuk, University of Birmingham, UK; Emilia Nowak, University of Birmingham, UK; Mark Simmons, University of Birmingham, UK

Soluble surfactants are used in many industrial applications, for example to create foams or for emulsification. Size and stability of droplets formed in emulsification processes and in various microfluidic devices depend on a large extent on the kinetics of thinning and break-up of liquid bridges connecting drop with the bath of liquid. Addition of surfactant can influence the kinetics in two ways: i) by lowering the interfacial tension and ii) by the Marangoni stresses suppressing any dilational deformation of the interface. Despite a considerable amount of work already done in the field, there is still a lack of systematic experimental studies dealing with soluble surfactants. Comprehensive experimental data are necessary to develop and validate predictive models for optimization of emulsification and microfluidic processes. Here we present the results of an experimental study, where the break up of capillary bridges is investigated over a broad range of viscosities of inner and outer liquid phases, surfactant activities and concentrations. We observed that surfactant-laden bridges near the pinch-off point follow either inertial or viscous thinning kinetics depending on viscosities of liquids involved, similar to pure liquid bridges. However the effective surface tension changes over a wide range from the equilibrium value at the concentration of surfactant used to that of pure water, the values depend upon both the surfactant concentration and activity.

\* Support from grant MEMPHIS (EPSRC UK) is gratefully acknowledged.

## TOWARDS STOCHASTIC MODELLING OF INERTIAL PARTICLE DISPERSION IN WALL-BOUNDED TURBULENT FLOWS

Jacek Pozorski, Polish Academy of Sciences, Poland; Maria Knorps, Polish Academy of Sciences, Poland

The present work addresses the issue of two-phase turbulent flows with the dispersed phase in the point-particle approximation. The Eulerian approach is adopted for the carrier fluid and the Lagrangian description is considered for the dispersed particles and for the modelling of fluid dynamics along the particle trajectories. The first aim is to discuss known stochastic models for particle dispersion with a particular emphasis on their behaviour in the near-wall, viscosity dominated flow regions. The second aim is to propose possible extensions of the existing approaches and the criteria that should be satisfied by physically-sound closures for particle transport by turbulence. This problem is of importance in the statistical turbulence modelling (of the Reynolds-averaged type) and for the large-eddy simulations (with subfilter closures), including an accurate prediction of particle deposition and resuspension. One of the issues in the near-wall region is a suitable closure in the for the subfilter time scale of the fluid seen by the particles, needed as a parameter in Langevin-type stochastic models. Another issue to be addressed is the choice between the formulations based on the subfilter velocities with respect to formulae using the instantaneous fluid velocities at particle locations. The LES results for the particle-laden, fully-developed channel flow are presented and discussed.

\* The work has been supported by the National Science Centre (NCN, Poland) through the research project 2011/03/B/ST8/05677.

## EXPERIMENTAL AND NUMERICAL INVESTIGATION OF A PULSATED EVAPORATING SPRAY

Jorge Cesar Brandle De Motta, CORIA, France; Benjamin Robbes, CNRS, CentraleSupélec, Université Paris-Saclay, France; Francis Shum Kivan, CERFACS, France; Matthieu Boileau, CNRS - CentraleSupélec, France; Marc Massot, CentraleSupélec, EM2C, France; Eleonore Riber, CERFACS, France

While combustion of gaseous mixture has received a rather complete investigation, the combustion of sprays, which is very common in practical applications, still requires fundamental studies to understand and model the complex interaction between the flame, the turbulence and the dispersed liquid phase.

The present contribution investigates a laminar stationary and pulsated spray of heptane evaporating in a counterflow configuration. The experimental set-up has been designed to allow a quantitative comparison against numerical simulations in terms of droplet velocity, position and size. This configuration is simulated with an academic code using both Lagrangian and Eulerian multi-fluid description of the spray with a special emphasis on the effect of droplet size polydispersion. Finally, a semi-industrial unstructured code is used to provide a detailed description of the pulsated jet dynamics and to evaluate a spray model that is commonly used in the aero-engine industry.

This combined experimental and numerical database is intended to become a reference to validate models and methods for evaporating sprays.

\* This work is part of the NEXTFLAME project supported by the ANR through the grant ANR-13-BS09-0023-01.

## THE INFLUENCE OF THE REYNOLDS NUMBER AND THE PARTICLE SIZE ON PARTICLE-LADEN TURBULENT CHANNEL FLOW IN THE DENSE REGIME

Pedro Costa, Delft University of Technology, The Netherlands; Francesco Picano, University of Padova, Italy; Luca Brandt, KTH Royal Institute of Technology, Sweden; Wim-Paul Breugem, Delft University of Technology, The Netherlands

Turbulent flows laden with finite size particles appear widely in environmental and industrial contexts. Fundamental insight in these flows from a micro-scale perspective is very limited but has become in reach due to the continuous growth in computing power, together with the development of efficient numerical methods. We use interface-resolved direct numerical simulations (DNS) to study turbulent plane-channel transport of dense suspensions of neutrally-buoyant finite-size spheres. The flow is sustained at a fixed bulk Reynolds number of  $Re_b=12,000$ , with a solid volume fraction of 20%. We investigated particle sizes of 10 and 20 inner-scaled diameters, corresponding respectively to 640,000 and 80,000 fully-resolved spheres. The DNS results are compared with the corresponding unladen case and with the case where the effect of the particles is solely modeled by an effective viscosity. In the conference we will present detailed statistics of these simulations. In particular, we will demonstrate that properly accounting for finite-size effects allow us to predict the observed drag increase, which appears higher than what is expected considering only the suspension effective viscosity.

## EXPERIMENTAL INVESTIGATION OF THE INTERACTION BETWEEN A STATIONARY RIGID SPHERE AND A TURBULENT BOUNDARY LAYER

Rene Van Hout, Technion - Israel Institute of Technology, Israel; Jerke Eisma, Delft University of Technology, The Netherlands; Edwin Overmars, Delft University of Technology, The Netherlands; Gerrit Elsinga, Delft University of Technology, The Netherlands; Jerry Westerweel, Delft University of Technology, The Netherlands

Time resolved tomographic PIV measurements (acquisition rate 250Hz) were performed in a turbulent boundary layer (TBL) on the side wall of an open channel, water flow facility (cross section  $60 \times 60 \text{ cm}^2$ ,  $W \times H$ ), 3.5m downstream of the inlet at a bulk flow velocity of  $U_b=0.17 \text{ m/s}$  ( $Re_b=U_b H/\nu=102 \times 103$ ,  $\nu=0.99 \times 10^{-6} \text{ m}^2/\text{s}$ ,  $Re_\tau=891$ ). The measurement volume was a horizontal slab ( $6 \times 1.5 \times 6 \text{ cm}^3$ ,  $l \times w \times h$ ) extending from the wall, 30cm above the bottom. The setup comprised four high-speed ImagerPro HS cameras ( $2016 \times 2016 \text{ pixels}$ ), a high-speed laser (Nd:YLF, Darwin Duo 80M, Quantronix), optics/prisms and data acquisition/processing software (LaVision, DaVis8.2). Data were acquired with and without a stationary held sphere that had a diameter,  $D=6 \text{ mm}$  ( $D^+=51$ , where  $+$  denotes inner wall scaling), and was positioned at  $y^+=5.4$ , 10.3 and 37.5mm ( $y^+=46$ , 88 and 319) from the wall (measured from the sphere's center). Sphere Reynolds numbers based on  $D$  and the average streamwise velocity at the sphere's center were 714, 825 and 984, respectively. Detailed mean velocity and Reynolds stress profiles will be presented for the different cases. In addition, the instantaneous interaction between and modulation of the coherent turbulence structures in the TBL and those generated in the sphere's wake will be discussed.

\* Support from Israel Science Foundation grant 1596/14 and the J.M. Burgers center is gratefully acknowledged.

## TURBOPHORESIS; THE TURBULENT SORLET EFFECT

Nils Erland L. Haugen, Norwegian University of Science and Technology, Norway; Dhrubaditya Mitra, Nordita, Sweden; Igor Rogachevskii, Ben-Gurion University of the Negev, Israel

We study, turbophoresis – the clustering properties of heavy inertial passive particles in an inhomogeneous turbulent flow – by direct numerical simulation of inhomogeneously forced turbulence in a periodic box without walls. The forcing is a periodic function of one coordinate direction. The inertial particles cluster near the minima of the turbulent kinetic energy. Drawing an analogy with the Sorlet effect in near-equilibrium thermodynamics, we can describe the flux of particles as a sum of two fluxes, described by two turbulent transport coefficients: turbulent diffusion of particles and the turbophoretic coefficient. The second (turbophoretic) flux is assumed to be proportional to the gradient of turbulent kinetic energy. The ratio of these two coefficients would be analogous to the molecular Sorlet coefficient; hence we call this the turbulent Sorlet coefficient. Our numerical calculation show that such a description is a good description of our data. Furthermore, we find that the turbulent Sorlet coefficient is a non-monotonic function of the particle inertia (described by the Stokes number); i.e. beyond a critical Stokes number the clustering of particles decreases, but in a smooth manner.

## TRANSIENT CHANGES OF THE LIQUID LEVEL IN STRATIFIED FLOW ALONG THE PIPE

Dvora Barnea, Tel-aviv University, Israel; Yehuda Taitel, Tel-Aviv University, Israel

Substantial changes of the liquid holdup along the pipe may take place due to small changes in the gas or liquid flow rates. These changes can be attributed to the existence of multiple steady state solutions in stratified flow. It is aimed here to investigate the transient process of these changes applying the transient two fluid model used to analyze the interfacial stability and the simplified model used for the analysis of the structural stability. It is shown here that the two approaches for stability analysis are consistent and in fact complement each other. Transient simulations were performed using the method of characteristics on the two fluid model and the results are compared to the simulations of the simplified equations used for the structural stability analysis.

\* Support from the Israeli Science Formation, grant # 281/14 is gratefully acknowledged

## BUBBLE DYNAMICS AT A NON-FLAT WALL

Havazelet Shmueli, , Israel; Ruth Letan, , Israel; Gennady Ziskind, Ben-Gurion University of the Negev, Israel

The present study deals with three-dimensional numerical modelling of single bubble growth, detachment and motion as may occur in nucleate flow boiling at a non-flat wall. The model solved is based on macro-region formulation of the bubble and describes its growth from the initial state, leading in some cases to detachment from the wall and consequent motion. The model includes a simultaneous solution of conservation equations for the liquid and gaseous phases. A formulation of the dynamic contact angle between the bubble and the wall is added. The model is founded on the level-set formulation, which is used to track the three-dimensional interface. The numerical modeling is facilitated by using the multi-grid method. An original in-house numerical code, realized in MATLAB, discretizes the complete three-dimensional conservation equations. As a result, bubble detachment modelling becomes possible, including its shape evolution and interaction with the surrounding liquid. The main conclusion from the calculations is that the bubble shape and growth rate strongly depend on its location and on the wall orientation. New features, not possible for flat walls and special for this case, are revealed and discussed. It is demonstrated that under certain conditions, the bubble is obstructed by the surface geometry. It is also shown how a growing bubble affects the flow features, e.g. vortices, which result from the wall shape.



## EFFECTS OF GRAVITY AND ITS ORIENTATION OF BUBBLE DEPARTURE

Paolo Di Marco, University of Pisa, Italy; Nicolo Morganti, University of Pisa, Italy; Giacomo Saccone, University of Pisa, Italy

Buoyancy governs the dynamics of boiling mechanism and thus, boiling heat transfer capacity. In the present work, the effects of gravity are studied with the aim to understand how to handle microgravity applications; in particular, the replacement of gravity with electric field and their coupled effects are evaluated. A simple experimental apparatus has been built, with a continuous train of nitrogen bubbles (obtained at low flow-rate: ranging from 3 to 30 mm<sup>3</sup>/s) generated from an orifice submerged in the dielectric still liquid (FC-72). Electric field is provided with an axisymmetric electrode providing up to 4.3 MV/m. High-speed images are acquired (500Hz), recording bubble growth and detachment for gravity acceleration ranging from microgravity to hyper-gravity. The results of two ESA parabolic flight campaigns (PF-58 & 60) will be shown. Furthermore, the shear effects of gravity, obtained by tilting the experimental setup, have been studied in the same controlled conditions. Departure forces are evaluated in these different configurations and results in terms of detachment diameters and bubble departure frequency are shown and compare with the most recent correlations available in literature.

## BUBBLY FREE AND IMPINGING FORCED JETS: EXPERIMENTAL STUDY BY PIV/PFBI/PTV

Konstantin Pervunin, Kutateladze Institute of Thermophysics, SB RAS, Russian Federation; Mikhail Timoshevskiy, Novosibirsk State University, Russian Federation; Dmitry Markovich, Kutateladze Institute of Thermophysics, SB RAS, Russian Federation

An experimental investigation of bubble-saturated free and impinging turbulent jets was carried out using planar fluorescence for bubbles imaging (PFBI) technique combined with PIV and PTV methods. PFBI was used to visualize bubbles in the central section of the flows and determine the dispersed phase characteristics, such as bubble size distributions and local gas fractions. PIV and PTV were applied to measure the velocities of the continuous and dispersed phases, respectively. The bubbly jet flows were studied at different volume gas fractions (up to 4%). Basing on 10,000 measured spatial distributions of instantaneous velocity fields of the both phases and local gas content, turbulent characteristics of the flows were calculated, including mixed one-point correlations up to the third-order statistical moments. The flow manipulation was implemented by excitation of the jets with vibratory motions of an oscillator (periodic flow rate pulsations) at the frequency of the best flow receptivity to intensify large-scale vortex structures (LSV). In addition, the measurements were carried out with a conditional sampling approach to trace the LSV evolution. One major result is that an opposite effect of the volume gas fraction was discovered in the initial jet region, where turbulent fluctuations are substantially suppressed by the bubbles, and close to the wall, where turbulent fluctuations were significantly increased, in comparison with a single-phase flow.

\* The research was funded by a grant from the Russian Scientific Fund (Project No. 14-19-01685) through IT SB RAS.

## SUBCOOLED FLOW BOILING EXPERIMENTS: TOWARDS MODELLING THE NEAR-WALL BEHAVIOUR

Carlos E. Estrada Perez, Texas A&M University, USA; Junsoo Yoo, Idaho National Laboratory, USA; Yassin A. Hassan, Texas A&M University, USA

Subcooled flow boiling is of importance to many high-energy and heat transfer systems. Although many advances on the understanding of subcooled flow boiling have been accomplished on the past decades, the prediction of the liquid and gas phases behavior has been translated to merely empirical correlations or modifications to the single-phase models. The reason of models development stagnation is in part due to the scarcity of detailed and reliable experimental information. In this work, Particle Tracking Velocimetry (PTV) has been applied to subcooled flow boiling conditions on which the influence of the vapor bubbles on the neighboring liquid can be readily measured. Such boiling conditions include near-wall, near-nucleation site experiments to capture the flow development from a single-phase superheated sublayer to the incipience of boiling. Furthermore, PTV has been applied to regions downstream of the nucleation site on which a full bubble layer has been developed. Combining the results of such boiling condition, new liquid-vapor relationships are currently being developed, and contain enough detailed information for the development of mechanistic models while at the same time enough flow statistics for the improvement of empirical models. This data has the potential to be used as benchmark for both, DNS and CFD simulations.

## FLOW AND SHEAR INDUCED BY BUBBLES RISING THROUGH HOLLOW-FIBER MEMBRANES IN A PARALLEL ARRANGEMENT

Katsumi Tsuchiya, DOSHISHA University, Japan; Saori Yamada, DOSHISHA University, Japan; Naoki Umemura, DOSHISHA University, Japan; Yasushige Mori, DOSHISHA University, Japan

The liquid flow induced by rising bubbles—often associated with the bubble wake dynamics studied in unbound systems—has recently found its applications in wall-confined systems, viz., to enhance the operability of membrane bioreactors (MBRs) for aerobic wastewater treatment. The use of aeration—or “aero-scrubbing”—in MBRs, which effects shear stresses on the membrane surface, aims at improving the surface-cleaning efficiency. It is thus important to understand the flow and shear induced by bubbles—rising in a “quasi-2D” system—which signifies a degree of three dimensionality superimposed onto the inherently 2D-flow behavior, confined by two walls with small gap thickness. This study focuses on the flow and the wall shear stress induced by bubbles rising through a series of hollow-fiber membranes in a parallel arrangement. Both CFD and EFD analyses are conducted based on VOF evaluation and LDV—in compliment to PIV—measurements, respectively, of the bubble-rim-induced liquid shear rate in the vicinity of the confining wall. Due to confinement by the “bumped” flat surface, the rise of (spherical-cap) bubbles is found to be often restricted to be rectilinear; yet the flow structure is complicated, hampering the evaluation of the associated velocity profile or its variation near the system boundaries. The high wall shear stress is detected around the bubble, caused by rapid flow in the liquid film region between the bubble surface and the wall. The wall shear stress induced by the flow around the bubble is thus likely to play a crucial role in aero-scrubbing the membrane surface.

## HYDRODYNAMIC STUDIES OF CUTTING BUBBLES IN MICRO-STRUCTURED SQUARE BUBBLE COLUMN USING ULTRAFAST X-RAY TOMOGRAPHY

Krushnath Thiruvalluvan Sujatha, Eindhoven University of Technology, The Netherlands; J. A. M. (hans) Kuipers, Eindhoven University of Technology, The Netherlands; Yuk Man Lau, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Frank Barthel, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Uwe Hampel, TUDresden, Germany; Markus Schubert, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Niels G. Deen, Eindhoven University of Technology, The Netherlands

Bubble column reactors are used in the industry for both two phase (gas-liquid) and three phase (gas-liquid-solid) processes. In spite of their prevalence, the reactor performance is limited by the interfacial mass transfer and rate of heat removal. To overcome these disadvantages a novel reactor concept is proposed. It is a micro-structured bubble column reactor (MSBC), where the micro-structuring is realized by a wire-mesh. The wires are intended to efficiently cut the gas bubbles and have a positive impact on the gas-liquid contact area in the bubble column, depending on the wire-mesh geometry (mesh opening/wire diameter). The scope of this research is to study the effect of the wire-mesh on the bubble cutting in an air-water system using ultrafast X-ray tomography. Experiments are conducted in a MSBC for different flow configurations, by varying the superficial gas/liquid velocities to obtain results of bubble size distributions and velocities. \* We acknowledge the financial support from European Research Council (ERC) Grant StG 259521 and StG 307360.

## BOILING HEAT TRANSFER TO IMMISCIBLE MIXTURES FOR ADVANCED COOLING OF SEMICONDUCTORS

Haruhiko Ohta, Kyushu University, Japan; Keisuke Iwata, Kyushu University, Japan; Daisuke Yamamoto, Kyushu University, Japan; Satoshi Okayama, Kyushu University, Japan; Yasuhisa Shinmoto, Kyushu University, Japan

To satisfy the requirements of practical cooling of semiconductors, experiments on boiling of immiscible liquids mixtures are performed. The heat transfer characteristics are quite different from those of non-azeotropic miscible mixtures where the heat transfer deterioration is observed. Immiscible mixtures can realize i) significant increase of CHF by the addition of small amount of more-volatile liquid, ii) enhanced heat transfer by the bubbles of more-volatile component, iii) elimination of non-condensable air by the system pressure larger than the atmospheric keeping low liquid temperature, iv) reduction of surface temperature overshoot at the boiling initiation. To apply the proposed system for the cooling of automobile inverters, experiments on pool boiling of FC72/ethylene glycol aqueous solution as a non-freezing immiscible coolant are performed. The mechanisms of intermediate heat flux burnout relating to the behaviors of liquid-liquid interface and hydrodynamic instabilities and of the deviation from the equilibrium liquid temperature relating to the distribution of phases composed of immiscible liquids and their vapor. For the wide application of the cooling system, new experimental results on flow boiling of immiscible mixtures are also discussed.

## POOL NUCLEATE BOILING FOR SEAWATER CONTAINING MINERALS

Shinichiro Uesawa, Japan Atomic Energy Agency, Japan; Yasuo Koizumi, Japan Atomic Energy Agency, Japan; Hiroyuki Yoshida, Japan Atomic Energy Agency, Japan

In the accident of TEPCO's Fukushima Daiichi NPS, seawater was injected into nuclear reactors. As possible effects of the seawater on cooling performance of fuel rods and debris, the change in the physical properties of the coolant and minerals deposition on heat transfer surfaces are considered. In the present study, pool nucleate boiling experiments for artificial seawater were performed to examine the characteristics of the nucleate boiling of the seawater. In the experiments, a copper heat transfer surface at the bottom of the pool was heated by Joule heating. Spatial average surface temperature was obtained from the variation of the electric resistance of the heat transfer surface. The local and instantaneous temperature was measured with an infrared radiation camera. The boiling bubble behavior was captured with a high speed video camera. In comparison with results for pure water, the surface temperature shifted to higher region. Especially, the surface temperature became much higher when the heat transfer surface was covered heavily with deposited minerals. In addition, the boiling bubble became easy to detach from the heat transfer surface, and departure bubble diameter became large. At the same time, the temperature at the boiling position changed drastically.

## OIL-WATER PIPE FLOW DEVELOPMENT AFTER A VALVE

Heiner Schumann, Norwegian University of Science and Technology, Norway; Pankaj Chandra, Norwegian University of Science and Technology, Norway; Ole Jørgen Nydal, Norwegian University of Science and Technology, Norway

The development of oil-water flow downstream of a valve was experimentally investigated applying a medium viscosity oil ( $\mu_{oil} = 25 \text{ mPa}\cdot\text{s}$ ). Experiments were conducted in the Multiphase Flow Laboratory at the Norwegian University of Science and Technology. With an inner diameter of  $D = 60 \text{ mm}$  and a total length of  $50 \text{ m}$  the transparent test section allowed for flow measurements as far as  $L/D = 800$  downstream of the valve. Based on visual observations and pressure gradient measurements at three positions along the horizontal test section the separation of the flow is documented. Two separation regions, one where droplet settling was predominant, and one where coalescence was the major mechanism, could be identified. Fast droplet settling creates a dense packed droplet layer. This could be related to a considerable increase in the pressure gradient, which slowly declines as droplets coalesce. Different valve openings were tested. Input water fraction, mixture velocity and pressure drop over the valve (mixing intensity) were found to influence the separation behavior. The experiments were interpreted in the form of a 1D flow model which reproduces the flow behavior and predicts the development length of the flow.

## THE DISPERSED OIL-WATER FLOW VELOCITY MEASUREMENT BASED ON CONTINUOUS-WAVE ULTRASONIC DOPPLER

Xiaoxiao Dong, Tianjin University, China; Chao Tan, Tianjin University, China; Ye Yuan, Tianjin University, China; Feng Dong, Tianjin University, China

Oil-water two-phase flow commonly exists in industry, especially in petroleum transportation and production process. Because it relates to the economic benefit and safety operation, the oil-water flow velocity measurement is extremely important. In this work, a dispersed oil-water two-phase flow velocity measurement method based on continuous-wave ultrasonic Doppler (CWUD) is proposed. A system involved CWUD sensor and excitation-acquisition circuit was established. For correlating the relationship between the measured frequency shift and the average flow velocity of the dispersed oil-water flow, a theoretical correlation was built based on the velocity profile and force analysis of droplets. According to the flow regime, the dispersed oil-water flow can be classified into water-continuous and oil-continuous flow. In each flow regime, there are different flow patterns. Hence the modeling was taken all of these factors into account. The dynamic experiments were conducted with different flow velocities in a horizontal Plexiglas pipe with inner diameter of  $50 \text{ mm}$ . The theoretical correlation was proved correct and improved by using the measured results. The final results show that the proposed method can realize the oil-water average velocity measurement with a high precision.

\* The authors appreciate the support from the National Natural Science Foundation of China (No. 61473206 & No. 61227006) and Science and Technology Innovation of Tianjin (No. 13TXXYJC40200).

## CFD ANALYSIS OF OIL-WATER TWO-PHASE FLOW AT FLOW REGIME TRANSITION IN HORIZONTAL PIPES

Evgenii Burlutskii, University of Nottingham, UK; Herve Morvan, University of Nottingham, UK; Kiran Kumar, University of Nottingham, UK

Dispersed oil-in-water two-phase flow in a horizontal pipe at low values of droplet phase volume fraction is analyzed numerically using CFD solver ANSYS Fluent in which a three-dimensional mathematical model of oil-water two-phase flow in pipes is developed. The Euler-Lagrange scheme is employed to resolve the interaction between the fluid (water) and droplet (oil) phases. The flow of the continuous fluid phase is calculated by solving the steady-state Reynolds-averaged Navier-Stokes (RANS) conservation equations that are coupled with a high Reynolds number  $k$ -turbulence model. The Lagrangian method is used to resolve the motion of oil droplets. The drag, gravity and shear-lift forces are taken into account. The shear-lift force is modelled and incorporated into the governing equations through the User Define Function (UDF) option as the external program. The two-way coupling procedure is employed. The model is validated through experiments in which the frictional pressure drop in oil-water dispersed flow in vertical and horizontal pipes is measured. The effect of mixture flow velocity on droplets dynamics in horizontal pipeline is investigated numerically. The transition from dispersed two-phase flow to stratified flow regime is analysed for the case of horizontal pipe. Different pipe diameters are also investigated by keeping the flow Reynolds number constant.

## EXPERIMENTAL INVESTIGATION AND MODELLING OF DISPERSED OIL-WATER PIPE FLOW

Iara Hernandez Rodriguez, University of São Paulo (EESC/USP), Brazil; Oscar Mauricio Hernandez Rodriguez, University of São Paulo, Brazil

Dispersed oil-water flow has been investigated in two horizontal pipes made of stainless steel and acrylic with  $82.8$  and  $26 \text{ mm}$  ID, respectively. The experiments in the steel pipe were conducted in the multiphase-flow test facility of Shell in the Netherlands, while the tests in acrylic pipe were performed in the Laboratory of Thermal and Fluids Engineering (LETf) of USP, Brazil. New experimental data on pressure gradient and holdup were obtained and the occurrence of drag reduction without addition of any drag reducer agent was detected. A significant slip ratio was observed, the oil droplets flowing faster than the continuous water phase. There is experimental evidence of the existence of a thin water film surrounding the turbulent oil-water dispersion. Flow measurements were carried out close to the acrylic-pipe wall with the application of a visualization technique and the water-film thickness was quantified. A phenomenological model was proposed to explain both the drag reduction and the slip ratio observed. Comparisons were made between film-thickness data and model predictions, with good agreement. A new correlation is proposed for slip-ratio estimation in turbulent oil-water dispersed flow.

\*The authors are grateful to FAPESP for financial support and research grant and to Shell for providing its facilities and financial support.

## NEW GAS HOLDUP EXPERIMENTAL DATA IN LARGE COUNTER-CURRENT BUBBLE COLUMNS

Giorgia De Guido, Politecnico di Milano, Italy; Giorgio Besagni, Politecnico di Milano, Italy; Fabio Inzoli, Politecnico di Milano, Italy; Laura Pellegrini, Politecnico di Milano, Italy

A new correlation for the gas holdup in bubble columns has been developed based on experimental data obtained in a counter-current bubble column of  $0.24 \text{ m}$  inner diameter and  $5.3 \text{ m}$  height. The annular gap and the open tube (both with a pipe sparger and a spider sparger) configurations have been tested for comparison. In all the runs, air has been used as the dispersed phase and various fluids (tap water, aqueous solutions of NaCl, a water-ethanol mixture and solutions of water-monoethylene glycol of different concentrations) have been employed as the liquid phase to study the influence of the liquid properties on the gas holdup. Gas superficial velocities in the range  $0.004$ - $0.26 \text{ m/s}$  have been investigated and, for the runs with water moving counter-currently to the gas phase, the liquid has been recirculated up to a superficial velocity of  $-0.11 \text{ m/s}$ . The bubble column design, the liquid velocity as well as the liquid properties have turned out to affect the gas holdup. Since no correlation has been found in the literature that can correctly predict the whole dataset, a new correlation has been developed. The proposed correlation allows to achieve a good agreement with the new experimental data.

## ULTRASOUND REFLECTOR RECOGNITION AND TRACKING TECHNIQUE FOR GAS BUBBLES IN LIQUID

Antonin Povolny, Tokyo Institute of Technology, Japan; Hiroshige Kikura, Tokyo Institute of Technology, Japan; Tomonori Ihara, Tokyo Institute of Technology, Japan

Measuring of gas bubble behavior in liquid is important for number of applications of gas-liquid two-phase flows. If flow is contained in transparent containers or pipes, optical methods can provide detailed information on bubble behavior. However, in many cases, transparent material can't be used due to high pressure, temperature or because the flow installation is already working. In those cases, ultrasound techniques can be used. Ultrasound Reflector Recognition and Tracking Technique (URRTT) has been developed as a new technique. It uses ultrasound transducers that emit an ultrasound beam into the liquid. The technique detects bubbles that cross this beam. Specifically, trajectories of their surfaces in direction of the ultrasound beam are obtained. If results from simultaneous measurements by more transducers (therefore more beams) are combined and bubbles tracked from one beam to another, secondary data can be obtained such as velocities perpendicular to the measurement line or bubble diameters. Experiments are conducted and results are compared to data obtained from image processing of High Speed Camera footage. The aim of experiments is to prove the concept of URRTT, provide data for the development of this method and also to detect limits and measuring performance of this method. Results obtained by URRTT can be of high value since each detected bubble is measured individually and thus, difference in the bubble behavior based on the size, velocity or history of the bubble can be described.

## TOMOGRAPHIC INTERROGATION OF GAS-LIQUID FLOWS IN INCLINED RISERS

Lokman A. Abdulkareem, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Josep Escrig Escrig, University of Nottingham, UK; Sebastian Reinecke, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Buddhikha Hewakandamby, University of Nottingham, UK; Barry James Azzopardi, University of Nottingham, UK

Measurements have been made with Electrical Capacitance Tomography (ECT) and a Wire Mesh Sensor (WMS) using air-silicone oil (viscosity 5 mPa s) on a 67 mm diameter pipe. The experiments were carried out in a facility at Nottingham University different inclinations. The cross-sectional distribution of gas fraction is obtained from the WMS and from the ECT the latter after post-processing. The combined data is used to provide information on the size, frequency and velocity of large bubbles (Taylor bubbles and spherical cap bubbles) and how they are influenced by the phase flow rates and the pipe inclinations. The boundary for spherical cap bubbles is identified. In addition, WMS output presents the results for a radial void fraction, as well as bubble size distribution, in particular that gives the percentage of void fraction in relation to bubble diameter. It is concluded that moving from horizontal to inclined to vertical, there is an increase in the percentage of smaller bubbles. There are also large diameter bubbles forming for all the inclinations. The cross-section averaged void fraction and its variation in time were measured. Taylor bubble frequency increases with increasing of liquid flow rate and inclination angle of the pipe.

## THE EFFECT OF AIR PRESSURE GRADIENTS ON THE DISCHARGE OF GRANULAR MATERIALS FROM A SILO

Yixian Zhou, IRSN, France; Pascale Aussillous, Aix-Marseille University, CNRS, France; Pierre Ruyer, IRSN, France

In the hypothetical conditions of a reactivity initiated accident in a nuclear power plant, some of the cylindrical rods, that contain the fuel, could break. If fuel fragmentation occurs, hot fuel particles and pressurized gas could be ejected out of the rod and then interact with the surrounding fluid. The violence of this interaction depends particularly on the discharge rate toward the fluid. This flow configuration is modeled thanks to a lab-scale device.

In this paper, we focus on the experimental study of the effect of an air flow on the discharge of a closed-top silo composed of spherical glass beads. The measured parameters are the mass flow rate and the pressure along the silo, whereas the controlled parameters are the size of particles, the size of orifices, and the flow rate of air which changes from counter-current to co-current. A simple analytical model is derived from first principles to describe the grains and air flows interaction and to predict the flow rate of particles.

## FLOW REGIME AND MASS TRANSFER STUDY IN STRUCTURED PIPE REACTORS USING ELECTROCHEMICAL MICRO-PROBES

Marco Altheimer, ETH Zurich, Switzerland; Dustin Becker, ETH Zürich, Switzerland; Philipp Rudolf von Rohr, ETH Zurich, Switzerland

Electrochemical micro-probes are used to study the flow regime and liquid-solid mass transfer in structured chemical pipe reactors. A novel electrode mounting technique is presented, allowing new geometries to be investigated. Instead of mounting the micro-electrodes flush directly with the wall, the electrodes are mounted flush with a small pin. This pin is then simply plugged to the structured chemical pipe reactor and by ensuring a minimal effect on the structure geometry, enabling the possibility to investigate geometries without access for grinding and polishing the electrodes from the inside. The novel mounting technique is validated in an empty pipe, where it is shown that the onset of turbulence is close to the theoretical value of  $Re = 2300$ . Additionally, a variety of pin lengths are studied in the structured reactor to determine the influence of the pin length on the transition. The onset of fluctuations are determined for liquid single-phase flows and the difficulties of measuring in gas-liquid two-phase flows are discussed. The liquid-solid mass transfer is measured for a liquid single-phase flow and a gas-liquid two-phase flow. The results are compared and the influence of the gas phase is discussed.

\*We gratefully acknowledge financial support from the Swiss Confederations innovation promotion agency (CTI) in cooperation with DSM Nutritional Products.

## DEVELOPMENT OF A DISCRETE GRAIN BREAKAGE MODEL TO PREDICT EROSION MECHANISM

Rahul Bharadwaj, ROCKY - DEM, USA; Karolline Ropelato, ESSS, Brazil; Alexander Potapov, ROCKY - DEM, USA; Brent Nelson, ESSS, USA

Erosion is a complex mechanical process in which material is removed from gas pipelines due to repeated multiple impacts of solid particles, such as silica, during oil and gas production. This phenomenon has been investigated by many researchers over the years and this can lead to substantial environmental issues and financial losses. The numerical approach to simulate the erosion problem has a strong dependency on empirical constants, thus making it less of a predictive tool. This study presents, for the first time, the development of a grain breakage model to study the erosion mechanism in a fundamental perspective. The erosion rate on a block of brittle material at various impact angles and solid particle shapes (silica) are predicted using this model. In addition, the feasibility of using Discrete Element Method (DEM), along with CFD to predict erosion rates is also investigated. Lastly, a futuristic perspective of using coupled DEM-CFD as a predictive tool to study erosion issues is also demonstrated.

## DENSE GRANULAR FLOW IN A BUBBLING FLUIDIZED BED AS A VISCOPLASTIC FLUID

Srdjan Sasic, Chalmers University of Technology, Sweden; Klas Jareteg, Chalmers University of Technology, Sweden; Henrik Strom, Chalmers University of Technology, Sweden

We propose and implement a numerical framework for a continuum description of highly dense regions of a bubbling gas-solid fluidized bed. The particulate phase is understood as a pressure sensitive viscoplastic fluid, described through a  $\mu(I)$  inertial rheology approach. Effort is made to show that such flows are sensitive to gradients in a number of fields (e.g. volume fraction, velocity) and that non-local effects may be needed when formulating the stress tensor of the particulate phase. We show that the behavior of dense regions of a fluidized bed may belong both to quasi-static (solid-like) and intermediate (liquid-like) granular-flow regimes. In addition, we look in the paper at a number of relevant questions related to numerical implementation of the proposed framework, in particular at the use of viscosity-regularization methods (e.g. type of the method used and convergence). To validate the results of our simulations, we use Particle Image velocimetry (PIV) to obtain detailed information on the dynamics of the particulate phase in the bed.

## NUMERICAL STUDY OF MULTI-SIZED GRANULAR PARTICLES IN ROTATING TUMBLER

Ziwei Zhang, China University of Petroleum-Beijing, China; Liang Ge, , China; Nan Gui, Institute of Nuclear and New Energy Technology, China; Zhenlin Li, , China

This study investigates the mixing characteristics of multi-sized granular particles in a three-dimensional rotating tumbler by DEM simulation. Comparisons of particle displacement, velocity, potential energy and statistical information are carried out with three ratios of diameters ( $d_1:d_2=1:1, 1:2, 1:3$ ) in a rotating tumbler under three rotating speeds ( $\omega=1.0, 2.0, 3.0$  rad/s). Periodic and intermittently pulsed variation of the gyration radius and velocities are illustrated by quantitative analysis. Moreover, the information entropy analyses based on the radial distribution function and local concentration functions are proposed to determine the most effective factors in improving the performance of mixing of granular particles. In addition, the results for the tumblers with and without frictions on the front and rear walls are compared, and the enhanced diffusion process by friction is indicated.

\*The support from the national natural science foundation of China (Grant Nos. 51576211 & 51106180) is gratefully acknowledged.

## A TWO-DIMENSIONAL CFD MODEL FOR GAS-LIQUID FLOWS IN A COIL-WOUND HEAT EXCHANGER

Thomas Acher, Linde AG, Germany; Manuel Knaup, The Linde Group, Germany; Stephan Goll, The Linde Group, Germany; Hans-Jorg Zander, The Linde Group, Germany; Stefan Lenz, The Linde Group, Germany; Christiane Kerber, The Linde Group, Germany

Due to their large specific heat transfer area accompanied by a compact design, coil-wound heat exchangers are widely used in various process plants (e.g., LNG plants). Depending on the application, two-phase flows may occur at both, the tube- as well as the shell-side of the apparatus.

One-dimensional thermohydraulic correlations are commonly used for the design of coil-wound heat exchangers. Fluid and thermodynamic effects in the apparatus which result in radial parameter variations are inaccessible to these conventional design tools. Therefore, a two-dimensional CFD model has been developed which accounts for the prevalent thermohydraulic phenomena. The shell-side as well as different tube-side sections are represented by hydraulically independent but thermally coupled simulation domains. The gas-liquid dynamics in each flow region is modeled based on an Euler-Euler approach. Unlike classical Euler-Euler models, local phase fractions and fluid properties are calculated from species relations as well as pressure and temperature fields. This hybrid model framework is augmented by locally evaluated correlations for pressure drop and heat transfer to account for apparatus internals and thermal coupling.

## SAND TRANSPORT IN SUBSEA MULTIPHASE FLOW PIPELINES

Yan Yang, Changzhou University, China; Shuli Wang, Changzhou University, China; Chuang Wen, Changzhou University, China

Most of oil and gas reservoirs worldwide are unconsolidated and the sand production and transport is almost an inevitable situation in the subsea petroleum industry. This paper was focusing on the transport of sand through pipelines in the subsea system. The liquid-solid two phase flow in the pipeline transport process was numerically calculated using the computational fluid dynamics (CFD) technique. The simulations were performed by employing Eulerian granular model, which was validated by comparison with the experimental data. The sand particle size range used in our studies was within 50-300 microns, and these particles most likely moved in a saltation mode. The results showed that the slurry velocities distributed uniformly at the vertical cross-section in a low concentration of fine sand. But in higher concentration, the slurry velocities near the pipe top were obviously larger than those close to the bottom. A plenty of sand deposition were observed at  $x = 30 D$  and the sand bed began to form in the pipe bottom. When the sand diameter reached 300  $\mu m$ , the height of the sand bed was even approximately equal to one third of the pipe diameter. The sand deposition was obviously observed while the inlet velocity was approximate 0.8 m/s for 50  $\mu m$  sand.

## CFD MODELLING OF GRAVITY SEPARATORS USING PID CONTROLLER

Manoj Vani, ANSYS Inc, India; Saurish Das, Eindhoven University of Technology, The Netherlands; Hemant Punekar, ANSYS Inc., India; Srinivasa Mohan, ANSYS Inc, India

Gravity Separators are used in oil and gas industry where oil and/or gas are separated from water through stokes settling. CFD simulations can help design and optimize these equipment. Accurate boundary conditions is a necessity for meaningful simulation, however, for gravity separators the pressure values at the outlets are generally not known simply because they are not measured. The valves downstream of the outlets are operated through controllers to ensure the desired interface level. The interface level is dictated by the required quality of the separation.

To eliminate necessity of explicitly knowing the outlet pressure value, an approach using PID (Proportional-Integral-Differential) controller has been developed to mimic the real life controllers. PID controller is a feedback controller that takes in the interface level as an input and adjusts the outlet pressures until the desired interface level is obtained. This is an automated approach that makes the simulation faster and improves the accuracy. Results of test cases and an industrial strength case are presented and discussed.

## COMPREHENSIVE STUDY OF ELECTRO-CHLORINATION PROCESS IN AN UNDIVIDED CELL: A COMPUTATIONAL FLUID DYNAMICS APPROACH

Maryam Momeni, Aalborg University, Denmark; Helge Grann, Grundfos Holding A/S, Denmark; Karin Dooleweerd, Grundfos Holding A/S, Denmark

For more than a century, chlorine has been used to disinfect drinking water and thereby avoid epidemics. However, there are some inherent challenges in using chlorine gas as a disinfectant due to transportation and storage of the gas. On site electro-chlorination, where sodium hypochlorite is generated electrolytically from salt and water is an alternative where no hazardous chemicals are required and the produced hypochlorite solution can be used and stored safely. The Grundfos product, Selcoperm, is based on those principles. In this work, a CFD frame of CFX 16.2, ANSYS is employed in order to model and analyze a full multiphase electrolysis process in a single cell and an experimental setup provides data to validate the model. The model is supported by a precise size distribution of gas bubbles resulted by experiments and the effects of convection, migration and diffusion on the mass transport of the reacting species are considered. Results indicate that the model-predicted cell performance has a good agreement with the experimental results. Both model and experiments represent that at low flow rate, evolution of dispersed gas bubbles have more pronounced impact on flow regime and consequently electrical field and current density distribution in the cell.

## EXPERIMENTS FOR CORE-ANNULAR FLOW (TO STUDY THE LEVITATION MECHANISM AND THE EFFECT OF BENDS)

Eduard Ingen Housz, Delft University of Technology, The Netherlands; Gjsbert Ooms, Delft University of Technology, The Netherlands; Ruud Henkes, Delft University of Technology, The Netherlands; Mathieu Pourquie, Delft University of Technology, The Netherlands; Anton Kidess, , The Netherlands

An experimental set-up has been built to study the flow of a high-viscosity-liquid core (oil) surrounded by a low-viscosity-liquid annulus (water) through a 6 m long tube with a diameter of 21 mm. In the experiments the levitation mechanism is analysed: this is the counterbalancing of the upward buoyancy force on the core (due to a density difference between the core liquid and the annular liquid) by hydrodynamic forces generated because of the movement of waves at the core-annular interface. Also the influence of bends on the stability of core-annular flow is studied. To that purpose bends with different values for the radius of curvature are used in the experimental set-up. The pressure drop over the tube is measured as function of the oil and water throughput and compared with the pressure drop of oil only flowing at the same oil throughput. Special attention is given to the measurement of the shape of the waves at the interface. The experimental results are also compared with the results of numerical simulations of the flow.

#### AN IMPROVED SHARP INTERFACE METHOD FOR SPH

Mingyu Zhang, *Institute of Applied Physics and Computational Mathematics, China*; Xiaolong Deng, *Beijing Computational Science Research Center, China*

It is important to deal with interface accurately and stably in numerical simulation of multiphase flows with clear interface. Smoothed Particle Hydrodynamics (SPH), one of the successful meshfree particle methods, can easily handle multiphase flows, especially with strong deformation and topological change, but usually not accurately and stably enough in many cases. In our previous work (Zhang & Deng, JCP 302 (2015) 469-484), a sharp interface method (SIM) for SPH is developed to improve the accuracy. The Level set method is used to track the interface position and helps to get more accurate geometric calculations. A ghost fluid method is introduced to handle the discontinuity, in which the interface states are calculated by satisfying the jump conditions and extended to the ghost fluid particles. It helps to get smooth and stable calculation near the interface. Based on it, an improved version of SIM for SPH is developed, in which the interface position is determined by the Lagrangian nature of SPH method. Therefore it can guarantee mass conservation, and is more stable than the previous version. In this talk, the improved version of SIM for SPH will be presented in detail. The performance of the developed method will be shown by benchmark tests and compared with the previous version.

\* Support from Natural Science Foundation of China (Nos. 11372050, 91230203, and 11371068) is gratefully acknowledged.

#### GROWTH AND SPECTRA OF GRAVITY-CAPILLARY WAVES IN COUNTERCURRENT AIR/WATER TURBULENT FLOW

Francesco Zonta, *University of Udine, Italy*; Miguel Onorato, *University of Torino, Italy*; Alfredo Soldati, *University of Udine, Italy*

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,  $E(k_x)$ , 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.

\*CINECA 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 Friuli Venezia Giulia under grant PAR FSC 2007/2013 is gratefully acknowledged.

#### CFD STUDY ON DROPLET FORMATION IN NON-NEWTONIAN FLUIDS FLOWING THROUGH T-JUNCTION MICROCHANNEL

Arnab Atta, *Indian Institute of Technology Kharagpur, India*; Somasekhara Goud Sontti, *Indian Institute of Technology Kharagpur, India*

Microdroplet production is a rapidly growing interdisciplinary field of research in chemical and biological sciences. Precise control of droplet volumes and reliable manipulation of individual droplets are of immense importance in numerous microfluidic applications. Based on available literature, it is apparent that droplet formation of Newtonian fluids in T-junctions has been extensively studied. However, the studies on non-Newtonian fluids in microchannels are scarce. In this study, the effect of various rheological properties of non-Newtonian fluids on droplet formation in an oil-water system is explored. Numerical investigations in a 3D T-junction under various flow conditions are carried out considering oil as a power-law fluid. Systematic analyses reveal that for a particular operating condition, formation of water droplets decreases with decreasing power-law index. However, the droplet diameter is found to be increasing with decreasing power-law index until an optimum value is reached. This work helps us to explain the effects of rheological parameters on the droplet breakup dynamics of a Newtonian/non-Newtonian microscale flow in a 3D T-junction. For controlled droplet sizes, several parametric studies are carried out with varying viscosity and surface tension of the fluids to identify the optimum conditions.

#### EXPERIMENTAL STUDY ON THE DEVELOPMENT OF SURFACE GEOMETRY UNDER FINGERING INSTABILITY

Takahiro Ito, *Nagoya University, Japan*; Masatoshi Yamashita, *Nagoya University, Japan*; Kazuyuki Nakagawa, *Nagoya University, Japan*; Yoshiyuki Tsuji, *Nagoya University, Japan*

Contact line of the front edge of liquid film expanding on an inclined plate can be destabilized, leading to formation of rib-like or saw-tooth like geometry of the contact line, regarded as viscous fingering. Such phenomena have much effect on the capability of the liquid coating, or the quality of coating. In this study three-dimensional measurement technique of the surface geometry has been developed and applied for the measurement of the fingering liquid film to elucidate the destabilization process and the growth of the fingers. The technique is based on the coding of the color to the local gradient vector of the surface, first proposed by Zhang (1996). The experiment was held by using aqueous mixture of ethylene glycol and glycerol as test liquid, on the transparent acrylic plate surface coated with hydrophilic coating material. The inclination angle of the acrylic plate from the horizontal is set between 4 degrees and 10 degrees. We found that the formation of the saw-tooth type geometry is made by the reacceleration of the contact line of the root of the developing finger, which was related with the cross sectional geometry of the surface near the contact line. The experiments also revealed that the dynamic (instantaneous and local) contact angle on the moving contact line remarkably deviates from the estimation based on the Ca number there, based on Voinov equation (Voinov, 1976), for the side region between the tip and the root of the finger.

#### EXPERIMENTAL INVESTIGATION OF DROPLET GENERATION IN A MICROFLUIDIC T-JUNCTION USING NEWTONIAN NANOFLUIDS

Katerina Loizou, *University of Nottingham, UK*; Edward Lester, *University of Nottingham, UK*; Buddhikha Hewakandamby, *University of Nottingham, UK*

The utilisation of microfluidics to generate droplets is a promising field that has received significant attention. Different applications involve the use of particle dispersions to generate droplets, however work done to characterise the droplet generation mechanism in such systems is limited. The effect of Newtonian nanofluids on the droplet generation mechanism and on the size of droplets in a microfluidic T-junction is examined. In this work, Hydroxyapatite nanofluids of different concentrations have been utilised as the continuous phase while deionised water was used as the dispersed phase.

High-speed videos have been employed to visualise and measure the volume of the resulting droplets. Two distinctly different modes of droplet generation in a microfluidic T-junction are observed. The differences between the observed mechanism and the droplet generation mechanism when pure liquids are employed are outlined in detail. It is postulated that as the concentration of particles in the nanofluid increases, the effect of the adsorption of particles on the emerging interface of the dispersed phase filament forming a particle layer is less dominant. At higher particle concentrations the shear stress exerted is thought to play a pivotal role on the droplet generation mechanism.

#### EFFECT OF THE LUBRICATION FILM DYNAMICS ON THE DROPLET MOTION IN A HELE-SHAW MICROCHANNEL

Jose-Maria Fullana, *Université Pierre et Marie Curie (Paris 6), France*; Yue Ling, *Université Pierre et Marie Curie (Paris 6), France*; Stephane Popinet, *CNRS UMR 7190, France*; Christophe Josserand, *Institut D'Alembert, France*

Droplet-based microfluidics is a promising tool for performing biomechanical and chemical assays and thus a comprehensive understanding of droplet dynamics in a confined microchannel is crucial. For a droplet with the horizontal diameter larger than the channel height, a lubrication film is formed between the droplet and the channel wall. The thickness of the film can be two orders of magnitude smaller than the channel height. Furthermore, the time step dictated by the surface tension is very small due to the low capillary number. Therefore, numerical simulation of a droplet in microchannel is indeed very expensive. In the present study, 3D simulations are conducted with the two-phase flow solver GERRIS, in which the interface is resolved by the Volume-of-fluid method and an adaptive mesh is used to refine mesh only near the interface and within the lubrication film. A disjoining pressure is introduced to incorporate the interaction between the interface and the channel wall. The dynamics of film is shown to have a significant impact on the droplet motion. The film thickness and the interface configuration are in turn dictated by the hydrodynamics and the disjoining pressure. The hydrodynamics and the disjoining pressure dominates for the regimes of large and small Ca, respectively. There exist a regime of intermediate Ca, in which both the hydrodynamics and disjoin pressure are significant, and interesting interfacial instabilities are observed.

\* Support from grant ANR-13-BS09-0011 is gratefully acknowledged.

#### BREAKUP OF DROPLETS IN BRANCHING MICRO-CHANNELS

Luis Portela, Delft University of Technology, The Netherlands; Cees Haringa, Delft University of Technology, The Netherlands; Conrad Jong, Delft University of Technology, The Netherlands; Duong Hoang, Delft University of Technology, The Netherlands; Chris Kleijn, Delft University of Technology, The Netherlands; Michiel Kreutzer, Delft University of Technology, The Netherlands; Volkert Van Steijn, Delft University of Technology, The Netherlands

The segmented flow of droplets and bubbles in channels of micrometer size occurs in numerous (bio) chemistry applications and an important question is whether (and how) a droplet (or bubble) entering a branching micro-channel breaks. In this work, we address this question using the T-junction as a paradigm. We performed experiments at low capillary numbers, with droplets of different sizes entering a T-junction. Our experiments show (i) the existence of stable non-breaking droplets (trapped inside the T-junction) at sufficiently low speeds, and (ii) that the critical speed at which the droplets break differs considerably from the scaling rules proposed in the literature. We developed a semi-empirical model that takes into account the 3D nature of the problem (with the flow around the droplet, through the gutters), which is not done in the scaling models proposed in the literature. Our model agrees well with the experimentally observed scaling and explains the change of scaling at low capillary numbers.

#### EXPERIMENTAL ANALYSIS OF GAS-LIQUID DISPERSION IN MECHANICALLY STIRRED TANK BY MEANS OF QUANTITATIVE ELECTRICAL RESISTANCE TOMOGRAPHY

Antonio Busciglio, Alma Mater Studiorum – Università di Bologna, Italy; Giuseppina Montante, Alma Mater Studiorum – Università di Bologna, Italy; Alessandro Paglianti, Alma Mater Studiorum – Università di Bologna, Italy

Electrical Resistance Tomography (ERT) has been widely adopted to infer fundamental dispersion properties in highly concentrated multiphase systems, where other techniques are either impossible to use (e.g. optical techniques), or very expensive (CT, PEPT, especially at large scales). At the actual state of development, quite satisfying qualitative results can be obtained by ERT (sufficient for the accurate assessment of regime transitions or blending times) but the ability to reconstruct quantitative data has still to be fully addressed. In this paper, a practical method for assessing quantitative data from ERT system is proposed and validated through experimental data.

This implied to quantify the effect of iterative vs non-iterative conductivity tomogram algorithms, determination of the optimal iteration number for iterative procedures and the choice of the correct conductivity-holdup relation. The general method proposed is applied to the dispersion of gas in a mechanically stirred tank agitated by a pitched blade turbine, finally resulting in affordable data for CFD models validation.

\*The financial support of MIUR "Progetto SO.FI.A. CTN01\_00230\_450760-Sostenibilità della filiera agroalimentare italiana" is gratefully acknowledged.

#### COMPARISON BETWEEN DIFFERENT METHODS FOR THE SOLUTION OF THE BIVARIATE POPULATION BALANCE EQUATION

Antonio Buffo, Aalto University, Finland; Alopaeus Ville, Aalto University, Finland

Population balance modeling is nowadays a widely used tool in many areas of chemical engineering and process industries, due its capability of capturing the dynamics of systems characterized by dispersed elements in a continuous phase, such as particles, droplets and bubbles. A number of industrially relevant problems, (e.g., granulation of powders, synthesis of nanoparticles, mass transfer and chemical reactions in multiphase systems, etc.) requires the formulation of a multi-dimensional Population Balance Equation (PBE), in which two or more properties of the population are taken into account to properly describe the investigated system.

In this work, different methods for the solution of the bivariate PBE are considered in order to assess both the accuracy and the computational time of each numerical scheme. The investigated numerical methods include Direct Simulation Monte Carlo (DSMC), High-order Moment conserving Method of Classes (HMMC) and Quadrature-based Moment Methods (QBMM). The comparison between different methods has been performed on a number of relevant test cases, considering coalescence and breakage problems, growth, nucleation, mass transfer and chemical reaction.

#### PARTICLE DISTRIBUTION IN UNBAFFLED STIRRED VESSELS

Alessandro Tamburini, Università degli Studi di Palermo, Italy; Andrea Cipollina, Università degli Studi di Palermo, Italy; Francesca Scargiali, Università degli Studi di Palermo, Italy; Giorgio Micale, Università degli Studi di Palermo, Italy; Michele Ciofalo, Università degli Studi di Palermo, Italy; Alberto Brucato, Università degli Studi di Palermo, Italy

Suspension and distribution of solid particles into liquids via mechanical agitation is frequently encountered in a number of industrial processes. Such operation is usually performed within cylindrical tanks provided with "baffles" (metal strips vertically deployed along vessel walls), although there are many cases in which these may be undesirable (e.g. due to ease of cleaning, risk of giving rise to dead zones with viscous liquids etc). In recent years, a growing awareness of the potential advantages of unbaffled vessels with respect to baffled ones has become widespread. However, there still is a poor knowledge of the fluid dynamics of these reactors, especially for multiphase systems, that inhibits exploiting their full potential. The present work is devoted to providing a first insight into the local solid particle distribution within these systems via experiments and CFD simulations. Experiments were carried out on a lab-scale unbaffled stirred tank by making use of a recently introduced technique named Laser Sheet Image Analysis: effects due to particle diameter, density and stirrer type (i.e. Rushton turbine and marine propeller) were investigated. As a difference from the axially stirred tank, the data collected with radially stirred vessels showed the presence of two stable toroidal attractors for the solid particles, whose concentrative effect increases as particle inertia increases. This singular behavior was also investigated by the CFD simulations and a good agreement with experiments was found.

#### A CONDITIONAL QUADRATURE METHOD OF MOMENTS FOR TURBULENT MIXING COUPLED TO A POPULATION BALANCE EQUATION

Alberto Passalacqua, Iowa State University, USA; Ehsan Madadi Kandjani, Iowa State University, USA; Jeffrey Heylmun, Iowa State University, USA; Rodney Fox, Iowa State University, USA

A conditional quadrature method of moments (CQMOM) was developed to solve turbulent mixing problems associated with population balance equations (PBE) by approximating the evolution equation of the composition probability density function (PDF) for the moments of the PBE. The well-established Interaction-by-Exchange-with-the-Mean (IEM) model was used to close the PDF evolution equation. The new CQMOM methodology was coupled to a quadrature-based PBE solver for univariate problems, which relies on the extended quadrature method of moments (EQMOM). The two procedures were implemented into the OpenQBMM framework, which leverages the OpenFOAM® CFD toolbox. The CQMOM approach for mixing problems was first tested considering two consecutive-competitive reactions to verify the implementation and validate the proposed approach. The coupled turbulent mixing-PBE approach was then used to investigate polymer aggregation in a multi-inlet vortex reactor (MIVR), typically used to perform flash nano-precipitation for the production of nanoparticles used in pharmaceutical applications. Results showing the predicted mixture fraction, reactant and product concentration fields and the polymer particle size distribution are shown.

\* Support from the SI2-SSE award NSF – ACI 1440443 is gratefully acknowledged.

#### MODELLING OF THE FLOW REGIME TRANSITION WITH THE SMOOTHED PARTICLE HYDRODYNAMICS

Michał Olejnik, The Szwedalski Institute of Fluid-Flow Machinery, PASci, Poland; Kamil Szewc, The Szwedalski Institute of Fluid-Flow Machinery, PASci, Poland; Jacek Pozorski, Polish Academy of Sciences, Poland

The Smoothed Particle Hydrodynamics (SPH) is a meshless, particle-based method of fluid flow modelling. Its main advantage over Eulerian approaches for multiphase flows computations lies in the fact that shape of the interface can be determined explicitly from positions of the particles of different phases, therefore there is no need for its artificial reconstruction. While SPH gives accurate results for simulations of single bubbles/droplets, its performance in more complex cases is still object of research.

In the present study we investigate the applicability of SPH to the modelling of the gas-liquid flow in a channel. Our aim is to simulate various flow regimes, i.e. bubbly, slug, annular and churn flows, and to capture the transition among them. We present qualitative and quantitative results of SPH simulations compared with reference data. The difficulties in the multiphase SPH include spurious interface fragmentation and unphysical coalescence. We discuss them together with possible remedies.

\* This research was supported by the EU FP7 Nugenia-Plus project (grant agreement No. 604965).

## A COMPARATIVE ASSESSMENT OF LATTICE BOLTZMANN AND VOLUME OF FLUID (VOF) APPROACHES FOR GENERIC MULTIPHASE PROBLEMS

Siddhartha Mukherjee, Delft University of Technology, The Netherlands; Ahad Zarghami, Delft University of Technology, The Netherlands; Cees Haringa, Delft University of Technology, The Netherlands; Sasa Kenjeres, The Netherlands; Harry Van Den Akker, University of Limerick, Ireland

The goal of this paper is a comparative assessment of two CFD solvers for simulating multiphase flows: a solver based on the pseudopotential Lattice Boltzmann Method (PP-LBM) and a finite-volume based solver using the Volume of Fluid (VOF) method. After a short review of the PP-LBM and the VOF models, we compare the accuracy and computational efficiency of these solvers for two generic two-phase flow cases. An identical case setup for both formulations allows for a direct comparison of the strengths and weaknesses of both methods. First, stationary droplets were simulated with varying density ratios, where the parasitic currents (leading to a spurious Reynolds number) in LBM were seen to be lower than in VOF by 2 to 3 orders of magnitude. On the other hand, compared to LBM, VOF helps maintain a very sharp interface for all density ratios. In terms of computation time, LBM clearly outperforms VOF for the stationary cases by an order of magnitude faster approach to equilibrium values. Lastly, a comparison is presented for a falling droplet compared to experimental results from literature, and the unique features and differences of the two methods are analysed in details.

\* The financial support from the Institute of Sustainable Process Technology ISPT and Shell is gratefully acknowledged.

## EULERIAN TRAJECTORY METHOD AND IMPINGEMENT MODEL FOR NON-SPHERICAL ICE CRYSTALS

Ellen Norde, University of Twente, The Netherlands; Edwin Van Der Weide, University of Twente, The Netherlands; Harry Hoeijmakers, University of Twente, The Netherlands

In this study an Eulerian method has been applied to compute trajectories and impact of non-spherical ice crystals. This research is carried out in the framework of EU-project HAIC. One of the objectives of this project is to understand and (numerically) model the physics of High Altitude Ice Crystals.

In the trajectory computation the particle motion and heat transfer have been adapted to accommodate the non-spherical shape of the ice crystals. Particle melting has been accounted for by applying a phase change model that combines melting and evaporation.

On impact with a surface ice crystals can either stick, bounce or fragment into smaller particles. A dimensionless parameter has been utilized to distinguish the onset of these sticking or bouncing regimes. In case of fragmenting particles the secondary particles are re-injected into smaller bins of the initial particle size distribution.

The Eulerian method employs a finite-volume-method for multi-block structured grids achieving second order spatial accuracy. Results will be shown and discussed for impact of ice crystals on a NACA-0012 airfoil.

\*This project has received funding from the European Union's Seventh Framework Programme for research, technological development and demonstration under grant agreement n°ACP2-GA-2012-314314.

## GAS ENTRAINMENT DUE TO A PLUNGING LIQUID JET IN FREE-SURFACE DOWNWARD ANNULAR-CHANNEL FLOW

Marcel Cavallini Barbosa, University of Sao Paulo, Brazil; Luis Enrique Ortiz Vidal, Sao Carlos School of Engineering, University of Sao Paulo, Brazil; Oscar Mauricio Hernandez Rodriguez, University of São Paulo, Brazil

Air entrainment is a common phenomenon in industrial applications. For example, self-aeration in spillways and open-channels, aeration in bubble column reactors and gas entrainment from Taylor bubbles in gas-liquid slug flows in pipelines. The phenomenon is related to the rate of kinetic energy dissipation produced, bubble coalescence and breakup and it is responsible for the diameter and size distribution of the bubbles. The proper knowledge of the air entrainment phenomenon plays an important role in the design and operation stages of these applications. In this paper, we present a study of air entrainment due to a plunging liquid jet. A revision of the available information from literature is presented. Experiments in an inclined annular duct with downward free-surface water flow were conducted. Air entrainment occurs when the free-surface water flow impacts on a static water-air interface at the bottom of the duct at a controlled liquid flow rate. Bubble-distribution measurements at several positions are performed using the Optical Reflectance Measurement (3D-ORM) technology. Experimental results, as bubble size distributions and average Sauter diameters, are compared to theoretical correlations. The presented findings can be used to further improve current knowledge on bubble generation mechanisms in multiphase flows.

\*The authors would like to thank STATOIL and ANP for the financial support and CNPq for the research grants.

## ACCURATE MODELING OF MOVING CONTACT LINE IN TWO-PHASE IMMISCIBLE FLOWS

Hanna Holmgren, Uppsala University, Sweden; Gunilla Kreiss, Uppsala University, Sweden; Martin Kronbichler, Technische Universität München, Germany; Simon Sticko, Uppsala University, Sweden

In simulations of moving contact line problems (i.e. flow problems involving two immiscible fluids that are in contact with a solid) it is necessary to introduce slip to avoid a singularity in the stresses. However, when the dynamics of the moving contact line is driving the flow, in for example capillary-driven flows, introducing slip in an accurate way is not straight-forward. Common problems are inaccuracies in the model and grid effects.

We present a second order accurate method to impose a slip velocity at the contact line. The method presented here is based on the so-called hydrodynamic model for steady movement of a contact line, introduced in [1]. The hydrodynamic model consists of an analytical expression for the fluid velocity field close to a moving contact line. This expression is derived from the creeping flow approximation of the Navier–Stokes equations and by imposing appropriate boundary and interface conditions. In this work, the velocity field from the hydrodynamic model is used to impose a slip boundary condition at the solid and to advect the contact line. Numerical results will be presented. Further, appropriate level set reinitializations for moving contact line problems will be discussed.

[1] C. Huh, L. E. Scriven, Hydrodynamic Model of Steady Movement of a Solid/Liquid/Fluid Contact Line, J. of Colloid and Interface Science, 1971.

## SINGLE ELLIPSOID IN SIMPLE SHEAR FLOW: INFLUENCE OF INERTIA

Tomas Rosen, Royal Institute of Technology (KTH), Sweden; Yusuke Kotsubo, University of Tokyo, Japan; Minh Do Quang, KTH, Sweden; Cyrus K. Aidun, Georgia Institute of Technology, USA; Fredrik Lundell, KTH, Sweden

The orientational dynamics of non-spherical particles in fluid flows depends on their reaction to local velocity gradients. For macroscopic particles, the velocity gradients can be strong enough for inertial effects to influence the results. To get fundamental understanding of the evolution of the orientational distribution of suspended particles due to inertia, we consider the behavior of one single ellipsoidal particle in simple shear flow. The case of a single spheroidal particle (an ellipsoidal particle with rotational symmetry) has been studied extensively in the recent years both theoretically and numerically. Less focus has been given to the tri-axial ellipsoidal particle, which will therefore be the main focus of this work. In the absence of inertia, it is well known that the nearly prolate tri-axial ellipsoid can perform chaotic orbits in shear flow. It has also previously been seen that this chaotic motion is inhibited by sufficient particle inertia, which promotes a drift towards rotation around the minor axis. In this work, we find that fluid inertia is an even stronger inhibitor of chaos and that the rotation around the middle axis stabilizes even at low particle Reynolds numbers. At moderate Reynolds numbers, the nearly prolate tri-axial ellipsoid actually behaves similarly to the perfectly prolate one. Thus, slight tri-axiality of the prolate spheroid does not induce fundamental changes of the dynamics (from predictive to chaotic) if fluid inertial effects are present.

## INTRODUCING A MULTISCALE MODELLING TOOL FOR THE SIMULATION OF POLYURETHANE FOAMS

Pavel Ferkl, University of Chemistry and Technology Prague, Czech Republic; Mohsen Karimi, Chemical Engineering, Politecnico di Torino, Italy; Daniele Marchisio, Chemical Engineering, Politecnico di Torino, Italy; Juraj Kosek, University of Chemistry and Technology Prague, Czech Republic

This study reports on the development of a software tool for the simulation of reactive expanding polyurethane (PU) foams. The foaming is described by the kinetics of the reactions and the evolution of cell sizes during the process. A simplified kinetic scheme is applied to describe the gelling and blowing reactions, whereas a Population Balance Equation (PBE) is solved to determine the evolution of bubble size distribution (BSD). Using a C++ interface library the macro-scale model is linked with a meso-scale detailed modeling tool designed for the prediction of bubble growth rate. The bubble growth tool models the interface between individual bubbles and their surroundings to produce the required growth rate for the PBE. The PBE, solved with the Quadrature Method of Moments (QMOM), instead tracks the evolution cell size during the foaming process. The developed tool is tested for twelve different PU recipes to predict the temperature and density evolution of the foam under different conditions. The agreement level observed between the numerical predictions and experimental data assured the successful application of the modeling tool for the simulation of polyurethane foams.

\* Support from European Commission under the grant agreement 604271 (Project acronym: MoDeNa; call identifier: FP7-NMP-2013-SMALL-7) is acknowledged.

## MODELING OF THE TRANSIENT TERRAIN-INDUCED AND SEVERE SLUGGING PROBLEMS USING THE DRIFT-FLUX MODEL

Konstantin Sinkov, Moscow Institute of Physics and Technology, Russian Federation; Andrey Kharlashkin, Lomonosov Moscow State University, Russian Federation; Pavel Spesivtsev, Schlumberger Moscow Research Center, Russian Federation

The simultaneous flow of gas and liquid in the pipelines and wellbores may lead to slug flow regime formation. This regime is characterized by significant periodic variations of pressure and flowrates in space and time. Due to the multiple issues caused by oscillating pressure and flowrates, prediction of slugging phenomenon is an actual problem for modeling of two-phase pipe flows. The present study is focused on terrain-induced and severe slugging in configurations of pipelines common in the oil and gas industry. The transient drift-flux model is applied to simulate slug flows in the hilly-terrain pipelines and horizontal pipeline-riser systems. The model is tested against previously published experimental data for air-water flows. By varying the specific set of tuning parameters a close match with the laboratory data is obtained. The applications of the model to the field scale problems involving the flow of hydrocarbon oil and gas and complicated by mass transfer between phases is also presented.

## MODELING AND PREDICTION OF GAS-SOLID TURBULENT MULTIPHASE FLOWS USING LAGRANGIAN AND EULERIAN APPROACHES

Dennis Dunn, Arizona State University, USA; Kyle Squires, Arizona State University, USA

Eulerian methodologies have been under increasing development for modeling multiphase flows comprised of dispersed particles. These techniques overcome some of the challenges encountered using Lagrangian approaches, especially for moderately dense dispersions where a range of complex inter-particle interactions are possible (e.g., collisions, coalescence, etc.). A quadrature-based moment method known as the Conditional Quadrature Method of Moments (CQMOM) is employed in the present effort to model dispersed phase transport within an Eulerian framework. The model represents Particle Trajectory Crossing (PTC) in a 3-D velocity space using a minimum two-point quadrature assumption. The present work provides direct comparisons between Eulerian and Lagrangian descriptions of the dispersed phase with (and without) particle collisions in a turbulent boundary layer and turbulent channel flow. Predictions generally compare favorably to previous results from the literature, including the prediction of flatter mean velocity profiles for larger Stokes number particles and when collisions are considered due to the increased momentum and energy transfer. CQMOM predicts the accumulation of particles at the wall, but has a more restrictive resolution of its discretized velocity space and thus has a tendency toward exhibiting reduced effects of PTC as compared to Lagrangian methods.

## A SPATIAL PARTICLE CORRELATION-FUNCTION ANALYSIS IN NON-ISOTHERMAL DILUTE PARTICLE-LADEN TURBULENT FLOWS

Enrica Masi, INP Toulouse, France; Pascal Fede, INP Toulouse, France; Olivier Simonin, INP Toulouse, France

In dilute gas-solid turbulent flows, as that encountered, for example, in pulverized coal combustion processes, the correct prediction of the non-isothermal/reactive particle-laden turbulent mixture relies on the accuracy of the modeling of the local and unsteady particles' behavior, which affects the hydro-thermodynamic coupling and the heat transfer and transport in and between the phases and at wall. In very dilute mixtures composed of highly inertial solid particles, such a local and unsteady behavior is the result of the particles' interactions with very distant and independent turbulent eddies, namely with different dynamic and thermal turbulent scales. Such interactions strongly modify the local particle velocity and temperature distributions, changing the local evolution of the properties of the dispersed phase. Their knowledge is thus crucial when modeling unsteady particle-laden turbulent flows. In this work, the focus is on the particle temperature and heat flux. Their characterization is provided by means of an analysis of the two-particle correlation functions in the frame of the direct numerical simulation of non-isothermal homogeneous isotropic, forced turbulent flows. Several Reynolds and Stokes numbers are investigated. Moreover, different scalar forcing are also analyzed and their impact on the particles' statistics discussed.

## VOLUME-AVERAGED EQUATIONS FOR DIRECT NUMERICAL SIMULATION OF PARTICLE-DISPERSED FLOW UNDER A GRID RESOLUTION COMPARABLE TO PARTICLE DIAMETER AND KOLMOGOROV LENGTH SCALE

Toshiaki Fukada, Osaka University, Japan; Shintaro Takeuchi, Osaka University, Japan; Takeo Kajishima, Osaka University, Japan

For the simulation of the turbulence modulation by particles with a diameter comparable to the Kolmogorov length scale, we propose a new Eulerian interaction force model by taking the local stress distribution into account through averaging over fractions of the surface of the particles. The length scale of the averaging volume is comparable to the particle diameter in order to directly capture the turbulence structure excluding the vicinities of particles. The interaction force model is constructed based on analytical solutions of two fundamental flows: uniform and shear flows. In particular for a uniform flow case, the stress component due to the nonlinear term is found to be essential for a nonuniform distribution of the interaction force. According to the test simulation of two fundamental flows with grid width comparable to the particle diameter, the present interaction force model considerably improves the prediction of the flow field around the particle and the mechanical work. Also, the residual stress originating from the volume averaging of the nonlinear term is modelled as an expansion of that for a linear velocity field. The model reflects the qualitative trend of the residual stress field obtained by highly-resolved DNS data.

## FULLY EULERIAN SIMULATION OF 3D TURBULENT PARTICLE LADEN FLOW BASED ON THE ANISOTROPIC GAUSSIAN CLOSURE

Macole Sabat, Laboratoire EM2C, CNRS, CentraleSupélec, France; Aymeric Vie, Laboratoire EM2C, CNRS, CentraleSupélec, France; Adam Larat, EM2C, CentraleSupélec, France; Marc Massot, CentraleSupélec, EM2C, France

Moment methods are of great interest for the simulation of particle or droplet-laden flows because of their intrinsic statistical convergence and their efficiency in terms of High Performance Computing compared to Lagrangian methods. In this work, we investigate the use of Kinetic-Based Moment Method with an Anisotropic Gaussian closure (AG). By solving all moments up to second order, this model reproduces statistically the main features of Particles Trajectories Crossing. The resulting hyperbolic system of equations has well-posed realizability properties. This system is solved using a realizable MUSCL-HLL scheme guaranteeing a physical meaning for the results. The present strategy is evaluated for the Direct Numerical Simulation of 3D turbulent particle-laden flows by using ASPHODELE solver for CORIA for the gas phase, and MUSES3D solver from EM2C for the Eulerian disperse phase. The results are compared to reference Lagrangian Simulation as well as Mono-Kinetic Eulerian method. By investigating the impact of numerical scheme and mesh refinement, we show that the AG is a predictive method for moderately inertial particles and is a good candidate for complex simulations in realistic configurations. Thus, AG is used to develop a fully kinetic Large Eddy Simulation formalism.

\* Support from SAFRAN-SNECMA is gratefully acknowledged.

## MODELING OF MACROSCALE SCALAR TRANSPORT IN BUBBLY SUSPENSIONS

Peter Spelt, Laboratoire de Mécanique des Fluides et d'Acoustique, France; Aurore Loisy, Université Claude Bernard Lyon 1, France; Aurore Naso, CNRS, France

In bubbly flows the transfer of heat and mass is strongly coupled to the dynamics of bubbles and to the disturbances they create in the liquid. When the suspension is homogeneous, scalar transfer can be described at the macroscale by an average transport equation where the closure is performed by an effective diffusivity tensor. However the relation between microscale dynamics and effective transport properties seems unclear. This study aims at improving the understanding and the modeling of macroscale scalar transport in bubbly suspensions.

We conducted 3D DNS of rising bubbles in otherwise quiescent liquid for several regimes, corresponding to Reynolds number ranging from 0 to 40 and yielding spherical, ellipsoidal, and skirted bubble shapes, and for a wide range of Péclet numbers.

First, we show how inertia and finite volume fraction affect the effective diffusivity of an idealized suspension consisting of bubbles regularly arranged on a cubic lattice. The influence of the gas-to-liquid diffusivity ratio and of a scalar discontinuity across the interface (Henry's law) is also examined. We then consider disordered systems and discuss the effect of the suspension microstructure.

\* Supports from ANR (TEC2), GENCI-CINES (Grant 2015-x20152b6893) and FLMSN are gratefully acknowledged.



### THE IMPACT LOAD AND THE POWER DENSITY DUE TO BUBBLE COLLAPSE IN 1D BUBBLY FLOWS AS INDICATORS FOR A MATERIAL EROSION MODEL

Georgios Bergeles, CITY UNIVERSITY, UK; Manolis Gavaises, City University London, UK

A mathematical model is presented and implemented in a CFD methodology for the collapse of bubbles in 1D-bubbly-cavitating flows created in a vibratory horn type facility. Bubbles are created and collapse during the horn oscillation; the peak pressures on the material specimen above a threshold created inside the 1D bubbly flow, the power density of the collapsing bubbles as also the number of collapsing events are monitored during the external pressure forcing; the effect of the forcing frequency and amplitude as also the initial gas bubble size on the peak pressure and on the power density are evaluated. The impact pressure above a threshold, defining the Erosion Aggressiveness Index (EAI), is compared to the material yield stress and identifies if the material surface is to be eroded, whilst the power density, defining the Erosion velocity Index (EVI), is compared to the material fracture energy, is correlated with the material erosion rate. The initial bubble size and the bubble size density distribution are accounted for in the proposed Indexes. The model results are compared with available experimental data and indicate among others the importance of the liquid quality in the execution of the horn type erosion tests of materials. \* Support from grant No 324313/FP7/2007-2013 is gratefully acknowledged.

### SPATIAL TRANSITION OF SYNTHETIC VOID WAVES IN A HORIZONTAL BUBBLY CHANNEL FLOW

Hyun Jin Park, Hokkaido University, Japan; Yuji Tasaka, Hokkaido University, Japan; Yuichi Murai, Hokkaido University, Japan

There are relatively small number of studies on horizontal bubbly channel flows in comparison with vertical systems in spite of their great importance on modifications of heat, mass and momentum transfers. As a fundamental experiment, spatial transitions of bubble swarm, generated by periodically injected air in a short duration time, traveling in a horizontal channel flow were investigated statistically using through-beam photoelectric sensors installed at several locations of the channel in the main flow direction. Each bubble which compose the bubble swarm has different advection velocity, because it is affected by mutual interactions, such as bubble-liquid and bubble-bubble interactions. This velocity difference causes modification of spatial bubble density distribution in the swarm and small bubbles coalesce into a large bubble at a high bubble density region. On the other hand, the large bubble is fragmented by strong shear of the flow. As a result, behaviors and conditions of the bubble swarm change continuously in the downstream direction. We discuss the spatial transition of bubble swarms using distributions of projection void fraction, bubble size and advection velocity obtained by the investigation.

### A COMPUTATIONAL STUDY OF SURFACTANT INDUCED ENHANCED HEAT TRANSFER IN FILM BOILING

Kannan Premnath, University of Colorado Denver, USA; Samuel Welch, University of Colorado Denver, USA

The surface tension between liquid and vapor phases of a fluid is strongly influenced by local surfactant concentration. This computational study is directed towards understanding mechanisms of enhanced heat transfer in film boiling caused by surfactant addition to the working fluid. A change in surface tension results in a change in the critical Rayleigh-Taylor wavelength leading to different bubble release patterns and a change in heat transfer rates. Enhanced heat transfer resulting from the change in critical wavelength has been studied experimentally and computationally by utilizing an electric field and taking advantage of the dielectric properties of the fluid and vapor phases. The presence of surfactants results in the additional mechanism of Marangoni convection due to the presence of tangential gradients in surfactant concentration along the phase interface. The presence of surfactants represents an opportunity to exploit a new mechanism (Marangoni convection) in order to enhance the heat transfer rate in boiling. We augment the CLSVOF method with bulk energy and diffusion equations along with a phase change model and an interface surfactant model at the interface to perform two-dimensional simulations studying this mechanism.

### MULTIPHASE PHENOMENA IN UNDERWATER EXPLOSION NEAR THE FREE SURFACE

Weibing Zhu, Harbin Engineering University, China; Hong Chen, Harbin Engineering University, China; Boo Cheong Khoo, National University of Singapore, Singapore

As a bubble moves near different boundaries, the motion characteristic will change largely compared with the condition of free field, especially near the free surface: there will appear a complex phenomenon because of the interaction. This paper is focused on the numerical simulation of complex the phenomenon. The process of underwater explosion near free surface is reproduced. Different typical phenomena near the free surface are calculated. The effect of different explosion depth and maximal radius on the initial water column formation is studied. Finally, the three-dimensional numerical model is also established based on the Real Ghost Fluid method; with Level set method tracking the interface, and the Harten-Lax-van Leer contact (HLLC) scheme was used for the approximate Riemann solvers. High-order accuracy is achieved using a weighted average flux (WAF) scheme. Numerical examples show that the proposed method is robust for two-dimensional and three-dimensional problems.

### TWO-PHASE FLOW AND HEAT TRANSFER CHARACTERISTICS IN NON-UNIFORMLY HEATED HELICAL COILED TUBES

Xiao-Juan Niu, Xi'an Jiaotong University, China; Liang-Liang Fan, Xi'an Jiaotong University, China; Shuaishuai Luo, Xi'an Jiaotong University, China; Liang Zhao, Xi'an Jiaotong University, China

The circumferential heat flux always distributes non-uniformly in the heat exchange tube for some engineering applications. Little research has been done on the vapor-water two-phase flow and heat transfer characteristics under non-uniformly heated conditions. To study the two-phase flow and heat transfer characteristics in a helical coiled tube with one-side heating, a numerical simulation is conducted in both uniformly and non-uniformly helical coiled tubes. The volume of fluid model (VOF) is adopted to calculate the interface between the two phases, and the mass transfer model is used to calculate the mass transfer source term in the phase-changing process. The simulation results show that the temperature difference between the outer and the inner wall is more significant in the helical coiled tube with one-side heating than that with two-side heating; the velocity and pressure drop in the one-side heating tube are lower than those in the two-side heating tube; The local heat transfer coefficient mainly depends on the vapor distribution, and the position of the lowest heat transfer coefficient around the cross-section is different in the non-uniformly and the uniformly heated helical coiled tubes. The study is helpful for the design of the non-uniformly heated helical coiled heat exchanger.

### MEASUREMENT OF EBULLITION CYCLE CHARACTERISTICS IN SUBCOOLED NUCLEATE BOILING OF WATER FLOWING IN A NARROW VERTICAL GAP

Henryk Anglart, KTH Royal Institute of Technology, Sweden; Heiko Kromer, ABB AB Corporate Research, Sweden; Aleix Fonellosa Caro, KTH Royal Institute of Technology, Sweden; Tor Laneryd, ABB AB Corporate Research, Sweden; Rebei Bel Fdhila, ABB AB Corporate Research, Sweden

A new experiment to investigate the vapour bubble size and the detachment frequency in subcooled nucleate boiling of water in 3 mm vertical gap and at atmospheric pressure has been performed. The main goal is to investigate the onset and the local characteristics of subcooled nucleate boiling during cooling of sensitive electronic systems or industrial energy transformation equipment. To this end a single-side heated test section with length of 400 mm and a rectangular cross section of 3 mm x 100 mm is employed. The boiling characteristics are obtained from high-speed camera recordings of selected heater areas through front transparent channel wall made of polycarbonate. In total 15 nucleation sites have been investigated using sampling rate 5000 fps, resolution 1024x768 pixels and recording time 2 s per site, and employing constant heat flux 3 W/cm<sup>2</sup>, water mass flux 58 kg/m<sup>2</sup>s and inlet temperature 90 °C. A sophisticated software package was developed to process images and retrieve nucleation site density, bubble size history and detachment frequency. The local flow and heat transfer conditions were obtained from a CFD conjugate-heat transfer model. As a result, the relationships between the bubble size at the departure and the detachment frequency have been obtained for all nucleation sites. It is concluded that no such single relationship exists which is valid for all nucleation sites and that the local conditions have to be accounted for.

## ELECTROSTATIC CONTROL OF IMPACT DYNAMICS OF A DROPLET ON SOLID SURFACES: SUPERHYDROPHOBIC BEHAVIOR

Saikat Datta, Indian Institute of Technology Kharagpur, India; Arup Kumar Das, Indian Institute of Technology Roorkee, India, India; Prasanta Kumar Das, Indian Institute of Technology Kharagpur, India

A coupled electrohydrodynamic model is used to numerically explore the phenomenon of droplet impact on a charged surface. The spatio-temporal electric field is obtained from the solution of Poisson and charge conservation equation. The electrostatic forces are coupled with hydrodynamics by incorporation of Maxwell stress term in the two-phase momentum equation. We varied the geometry of the foot print of the charged region to control the contact line mobility. The electrical stress generated near the three phase contact-line due to the presence of electric field constrains its motion in the desired direction and causes a change of impact behavior. We observed that a hydrophilic surface shows hydrophobic characteristics when the droplet is collided with it at a concentric ring shaped charged region. As the phenomenon is mainly governed by the interplay between electrostatic, capillary and inertia forces it is explored through the parametric variation of applied voltage, equilibrium contact angle and impact velocity. A regime map is also generated to identify the favorable conditions of droplet detachment based on the governing parameters. The outcome of the present study can be utilized to develop a vast range of engineering surfaces for various applications like drop wise condensation, anti-corrosion and self-cleaning.

## FORMATION OF A LIQUID RING VIA DROP IMPACT

Seungho Kim, Seoul National University, Republic of Korea; Myoung-Woon Moon, Republic of Korea Institute of Science and Technology, Republic of Korea; Ho-Young Kim, Seoul National University, Republic of Korea

It is generally conceived that water drops bounce off superhydrophobic surfaces upon impact. Here we show that the impacting drop can form a thin liquid film (spherical cap shape) and reside on superhydrophobic surfaces with the help of a superhydrophilic annulus surrounded by water-repellent background. The drop is pinned by the hydrophilic annulus after initial spreading, but the film gets destabilized and ruptures as its volume decreases due to evaporation. The rupture begins with a hole, which grows until the water film is ejected from the inner superhydrophobic region while leaving a liquid deposit on the hydrophilic annulus, a "liquid ring". The ejected water drop jumps off the solid substrate with a different take-off angle depending on the geometries of predefined annulus pattern. We visualize such novel drop motions with a high-speed camera and quantify their dynamics by combining experimental measurements and theoretical models.

## WET GAS MEASUREMENT BY V-CONE THROTTLE DEVICE

Denghui He, Xi'an University of Technology, Xi'an Jiaotong University, China; Bofeng Bai, Xi'an Jiaotong University, China

Wet gas flow through V-Cone throttle device has been investigated by experiment and numerical simulation. The phase distribution and pressure drop of wet gas along the V-Cone throttle device were concluded. The pressure recovery length downstream of V-Cone was determined as well as the dynamical characteristic and variation of the vortex behind V-Cone were analyzed. The results show that the V-Cone throttle device has excellent performance on the measurement of wet gas. We proposed a parameter called two-phase mass flow coefficient (TPMFC) to correct the "over-reading" of the V-Cone throttle device on measurement of gas flow rate. The influencing parameters of TPMFC were discussed and based on which a new wet gas measurement model was developed. Comparisons with existing models demonstrated that the new model predicted the gas flow rate of the wet gas more accurate. To reflect the influences of the liquid on the measurement, the pressure loss ratio of the V-Cone throttle device was proposed. By incorporating the wet gas measurement model, the wet gas correlations to simultaneously meter the gas flow rate and liquid flow rate were concluded.

\* Support from the National Natural Science Foundation of China under Grant No. 51276140 is gratefully acknowledged.

## A NOVEL DROPLET DRYING MODEL APPLIED TO A LABORATORY SPRAY DRYER

Ali Darvan, Martin-Luther-University Halle-Wittenberg, Germany; Martin Sommerfeld, Martin-Luther-University Halle-Wittenberg, Germany

Spray drying for powder production is widely used in many industrial areas. The quality of produced powders is strongly governed by dryer configuration and operational conditions. Therefore, numerical calculations of the spray drying process were conducted for many years in order to support dryer design and lay-out. However, a dream is still the numerical prediction of the dried powder properties. This is the result of the numerous physical effects influencing powder formation in a spray dryer: response of droplets and particles to turbulence, temperature history of the droplets, drying characteristics, as well as droplet coalescence and agglomeration of particles. A novel solution droplet drying model was developed which accounts for the temperature distribution inside the droplet and the solids diffusion in the liquid phase. This model was thoroughly validated for single droplet drying under different conditions and for various solution properties. Now the model was implemented in an Euler/Lagrange code and two-way coupled calculations were compared to measurements done for a simple spray dryer configuration, namely spray drying in a pipe flow. The profiles of spray droplet velocity showed good agreement with the measurements. However, the averaged droplet temperature and the solids content were over-predicted, which is at present further analysed.

## THE APPLICATION OF THE DEVELOPED IMAGE PROCESSING ALGORITHM TO STUDY THE BEHAVIOR OF WATER DROPLETS IMPINGING ON THE TILTED HEATED SURFACES

Hadiyan Kuntoro, Gadjah Mada University, Indonesia; Akmal Irfan Majid, Gadjah Mada University, Indonesia; Deendarlianto Deendarlianto, Faculty of Engineering, Gadjah Mada University, Indonesia; Yasuyuki Takata, Faculty of Engineering, Kyushu University, Japan; Masamichi Kohno, Faculty of Engineering, Kyushu University, Japan

Spray cooling is commonly implemented in the manufacture of steel, iron and aluminum products. This technique is applied by spraying liquid droplets over high temperature surfaces to cool down the surface temperature rapidly, e.g. in quenching processes. A comprehensive study of the liquid droplets impinging on the high temperature surfaces plays is crucial to improve the industrial processes. An algorithm of image processing technique was developed to study the behavior of water droplets impinging on the tilted heated surfaces. The heated stainless steel surfaces (SUS304) with 3 variations of surface roughness ( $R_a = 0.04, 3$  and  $10$ ) were employed. The surfaces with diameter  $30$  mm were tilted  $15^\circ$  to the normal plane and the surface temperatures were varied from  $150$  up to  $500^\circ\text{C}$ . The droplet diameter and the impinging velocity were kept constant around  $700\ \mu\text{m}$  and  $4\ \text{m/s}$ , respectively. The experiments were aimed to develop a high quality database of water droplets impinging on the tilted heated surfaces. The image processing algorithm was able to measure the important parameters of the dynamic behaviors of water droplets, namely apparent contact angle, spreading diameter, contact time and moving distance. As a result, it was found that (1) the rougher the heated surfaces, the longer the contact time of water droplets on the heated surfaces; (2) the higher the temperature of the heated surfaces, the shorter the contact time of water droplets on the heated surfaces.

## COMPARISON BETWEEN EXPERIMENTAL CHARACTERISATION OF THE MIXING OF IMMISCIBLE FLUIDS IN STATIC MIXER USING PLIF AND CFD MODELLING USING OPEN FOAM.

Federico Alberini, University of Birmingham, UK; Giuseppe Forte, University of Birmingham, UK; Domenico Daraio, University of Birmingham, UK; Alessio Alexiadis, University of Birmingham, UK; Mark Simmons, University of Birmingham, UK

The performance of Kenics (KM) and Sulzer (SMX+) designs of static mixer has been investigated for the mixing of immiscible fluids using in situ optical measurements. The fluids used are water as the continuous phase and either silicone oil in the presence of surfactant (Sodium Lauryl Ether Sulfate), or Lytol mineral oil in the presence of a nonionic surfactant (Span 80). The dispersed phase volume fraction was between  $0.0072\%$  and  $0.028\%$  and the superficial velocities ranged from  $0.16$  to  $0.91\ \text{m/s}$ . The pipe diameter was  $0.0127\ \text{m}$  and  $6$  or  $12$  mixing elements were used for each mixer type. Planar Laser Induced Fluorescence (PLIF) has been used to obtain images of the droplets formed in a traverse section across the mixer outlet. Image analysis methods have been developed, for the blending of miscible fluids. The analysis enables drop size distributions to be obtained as a function of the number of mixing elements, interfacial tension and superficial velocity. A CFD turbulent model in multiphase system has been developed using Open Foam. A  $k-\omega$  turbulent model has been used combined with multiphase solver. The CFD modeling has been used to extrapolate more detailed information about the fluid flow properties on the interface between the two immiscible fluids. This has enabled better understanding of immiscible fluid mixing inside the static mixer and explains the trends of drop size distributions obtained experimentally.

## REFRACTIVE INDEX MATCHING METHOD TO STUDY GASOLINE SPRAY-WALL IMPINGEMENT

Nicolas Lamarque, Continental Automotive France, France; Giacomo Piccinni Leopardi, Continental Automotive France, France; Christoffer Schmit, Continental Automotive France, France; Jerome Helie, Continental Automotive France, France

Spray-wall impingement, which can occur in gasoline direct injection (GDI) engines, generate liquid films upon either the piston or the cylinder walls. They are responsible for a bad local mixture formation, which causes pollutant formation, especially particulate matter. The refractive index matching method is an optical measurement technique, which enables, after fine calibration, to quantitatively characterise the liquid film surface and depth (here, from a tenth to a few micrometres). This method relies on the use of a transparent plate (quartz or sapphire) with a uniformly roughened side. As liquid fills in the surface troughs, the illuminated plate scatters less light towards the camera, indicating how thick the film is.

The experimental set-up is here used to study a GDI prototype in a laboratory (atmospheric pressure) and evaluate the effects of different factors, such as fuel pressure, injected mass, wall temperature, injector-wall distance. High-speed recording enables to observe the film generation and its vaporisation, especially for wall temperatures higher than the fuel saturation temperature. Eventually, the main benefits and drawbacks of this technique are underlined.

\* Partially financed by Agence Nationale de la Recherche in the framework of ASTRIDE project

## COMPUTATIONAL STUDY OF DENSE GRANULAR FLOWS IN STIRRED REACTORS

Lokman Bennani, IMFT, France; Herve Neau, IMFT, France; Cyril Baudry, EDF R-D, France; Jerome Lavieville, EDF R-D, France; Pascal Fede, INP Toulouse, France; Olivier Simonin, INP Toulouse, France

In chemical engineering applications, reactors featuring rotating parts are common practice. As these rotating parts are present in order to enhance chemical reactions, it is essential to take them into account when performing predictive numerical simulations. This aspect can be particularly challenging, even more so when complex industrial geometries are to be treated. In this communication the rotating mesh numerical methodology of NEPTUNE V3.0 (an Eulerian n-fluid multiphase flow CFD code) is presented. The method is based on splitting the domain into static and rotating parts. The information between rotating and static parts is passed thanks to a non-conformal mesh matching technique. The methodology is first validated, both numerically and experimentally using the classical rotating drum case. The high degree of compaction of the flow is taken into account thanks to a frictional stress tensor. The method is then pushed further and used to investigate the hydrodynamics of dry granular beds in stirred vessels. The results show that the rotating mesh method can effectively treat such configurations, hence offering interesting insight concerning the dynamics of the flow.

## DRAG REDUCTION DUE TO ROTATION WITHIN GRANULAR MEDIA

Wonjong Jung, Seoul National University, Republic of Korea; Sung Mok Choi, Seoul National University, Republic of Korea; Wonjung Kim, Sogang University, Republic of Korea; Ho-Young Kim, Seoul National University, Republic of Korea

We present the result of a combined experimental and theoretical investigation of granular drag reduction induced by rotation of self-burrowing seeds of *Pelargonium* species. The helically coiled awn, which attaches to the seed of *Pelargonium* species, reversibly coils and uncoils in response to the change in environmental humidity. Such coiling-uncoiling motion of the awn gives the self-burrowing seed a thrust with rotation against the soil. We find that the rotation of the seed dramatically reduces the granular drag, which greatly facilitates its self-burrowing. We here aim at elucidating and quantifying drag reduction in digging of the self-burrowing seeds. We demonstrate that the surface area normal to advancing direction is the key parameter determining the granular drag on a rotating intruder. We then develop a partial slip model that explains how the rotating motion of the intruder extenuates the effective area where the granular hydrostatic pressure is applied.

## DYNAMIC BEHAVIOUR OF TWO-PHASE FLOW THROUGH ORIFICES USING PARTICLE IMAGE VELOCIMETRY

Olufemi Eytape Bamidele, University of Guelph, Canada; Chris Vines, University of Guelph, Canada; Wael Ahmed, University of Guelph, Canada

Two-phase flows through orifices are found in several nuclear power generation systems. The complex nature of the two-phase flow through orifices is characterized by the deformable interfaces, turbulence, phase interaction, and compressibility of the gas phase. The aggressive nature of two-phase flow in such geometry causes failures in many boiler blowdown, steam extraction and condensate systems due to variety of degradation mechanisms such as flow accelerated corrosion, cavitation erosion, and liquid impact erosion. This severely affect both safety and reliability of these systems and sometime lead to fatalities and huge economic loss. Although, two-phase flow through different piping components have been studied in the past, the dynamics behavior of two-phase flow on the developing region before and after the orifices has not been fully understood. Therefore, the present study aims at understanding the fundamental behavior of two-phase through geometries such as orifices using the Particle Image Velocimetry (PIV). In this paper, the two-phase flow redistribution, local void fraction and liquid turbulence characteristics are experimentally investigated for different flow conditions. The present study showed how the PIV can be used to characterize gas-liquid two-phase flows through piping components. The bubble velocity, instantaneous flow velocities, streamline and vorticity fields for slug flow were obtained utilizing data processing algorithms for phase discrimination.

## LARGE EDDY SIMULATION OF SAND WAVE EVOLUTION

Ali Khosronejad, University of Minnesota, USA; Fotis Sotiropoulos, Stony Brook University, USA

Here begins the body of the abstract.

We have employed a coupled hydro-morphodynamic numerical model, the St. Anthony Falls Laboratory Virtual StreamLab (VSL3D), to perform large-eddy simulation (LES) of stratified turbulent flow over a mobile sand-bed in a laboratory setting. The VSL3D numerical model is used to elucidate the complex interaction of turbulence, mobile bed, and channel geometry, which eventually leads to formation and growth of small-scale sand waves with a maximum amplitude of about 6cm. Using coupled hydro-morphodynamic module of VSL3D, we simulate sand wave initiation, growth and evolution in the channel. Major characteristics of the computed sand waves in this study, including the early cross-hatch and chevron patterns to fully grown 3D sand waves, are shown to be in good qualitatively and quantitatively agreement with the experimental data of Venditti et al.(2005). As we show in this work, numerically captured sand waves continuously grow in amplitude and wavelength. The process that eventually lead to creation of energetic coherent structures in the form of horseshoe vortices. Additionally, upward transport of low momentum fluid (near the mobile bed) and suspended sediment leads to formation of boil events at the water surface.

Here ends the body of the abstract.

## ANALYTICAL AND PRACTICAL ANALYSIS OF FRICTIONAL-KINETIC MODEL FOR DENSE AND DILUTE GAS-SOLID FLOWS

Pascal Fede, INP Toulouse, France; Francois Audard, IMFT, France; Emmanuel Belut, INRS, France; Jean-Raymond Fontaine, INRS, France; Olivier Simonin, INP Toulouse, France

Granular flows are encountered in many industrial or geophysical flows. When the solid volume fraction becomes large, the particle-particle contacts become long and frictional effects may appear. In the literature of Euler-Euler numerical simulations, many different models can be found for taking into account the frictional effects by introducing frictional pressure and viscosity models. In the present paper, an analysis of the behavior of several models on a simple case of homogeneous sheared granular flows is conducted. In a second part, the models are implemented in NEPTUNE\_CFD code and 3D numerical simulations of bins discharge are performed. The computed particle velocities and solid mass flow rates are compared to those measured in the experiments of Djermane et al. 2014 and to the Beverloo law for different particle diameter. For the highest diameter, the solid mass flow rate is successively captured by the different models, while it is systematically underestimated when the diameter of particles is reduced.

## DEVELOPMENT OF AN INDUSTRIAL SCALE MICROBUBBLE GENERATOR FOR THE PURPOSES OF AEROBIC WASTE WATER TREATMENT

Akmal Irfan Majid, Gadjah Mada University, Indonesia; Deendarlianto Deendarlianto, Faculty of Engineering, Gadjah Mada University, Indonesia; Wiratni Wiratni, Gadjah Mada University, Indonesia; Indarto Indarto, Gadjah Mada University, Indonesia; Enggar Diwangkara Brata, Gadjah Mada University, Indonesia; Baskoro Adiputro Purwono, Gadjah Mada University, Indonesia; Alva Edy Tontowi, Gadjah Mada University, Indonesia

The rapid growth of residences and commercial industrial establishments requires efficient waste-water treatment. In aerobic biological waste water treatment, microbubble generator (MBG) is an alternative solution that is easy to implement due to good purification capabilities and simple construction. The present paper examines the possibility to use MBG as an aerator for the waste water treatment from laboratory to the industrial scale. Some basic studies on microbubble characteristics have also been conducted to determine the optimal configuration of MBG, bubble dynamics. Development of advanced measuring technique to visualize the microbubbles is also reported. For the purposes of aerobic waste water treatment, industrial scale MBGs based on orifice and porous pipe types were used in this experiment. The dissolved oxygen (DO) rates were evaluated in some particular points in the coverage area. The influences of air-liquid flow rates and MBG configurations to the dissolve oxygen rates were also carried out. The optimal configuration was able to reduce the averaged chemical oxygen demand of waste until 45% and supply enough oxygen for the aerobic bacteria growth. The effective working time for pumping was also determined to minimize the operating cost. In comparing to the available commercial diffuser system, the developed microbubble generator system consumes less energy.

## PHYSICAL AND MATHEMATICAL MODELLING OF MULTIPHASE FLOW IN THE NEAR-WALL REGION OF ENTRAINED FLOW REACTORS

Giovanni Tretola, Consiglio Nazionale delle Ricerche, Italy; Maurizio Troiano, Università degli Studi di Napoli Federico II, Italy; Francesco Saverio Marra, Consiglio Nazionale delle Ricerche, Italy; Piero Salatino, Università degli Studi di Napoli Federico II, Italy; Fabio Montagnaro, Università degli Studi di Napoli Federico II, Italy; Roberto Solimene, Consiglio Nazionale delle Ricerche, Italy

During slagging combustion/gasification of solid fuels, particles move toward the film of slag covering the reactor walls, establishing a very complex multiphase particle-wall interaction. This study aims to improve the understanding of the different char-slag micromechanical interaction patterns which occur in the near-wall region, on the basis of the stickiness of both the wall layer and the impinging char particle. A 0.10 m-ID lab-scale cold entrained-flow reactor (length 0.1–0.6 m), optically accessible, and equipped with a nozzle whence molten wax is atomized into a mainstream of air, is used to mimic, at atmospheric conditions, the near-wall fate of char/ash particles in entrained-flow gasifiers. The flow and segregation patterns are observed by means of a high speed CMOS camera. The partitioning of the wax droplets/particles into solid and liquid phases is characterized by their selective collection at the reactor exhaust. The same configuration is simulated, adopting simplifying assumptions, with a granular flow model. Particle-particle collisions are modelled with an Hertzian approach, including torque and cohesion effects. Results illustrate the different structures of near-wall particle layer that establish in the four interaction regimes between particles and confining walls under sticky (molten) or non sticky (solid) conditions.

## PRESSURE DROP FOR GAS-NON-NEWTONIAN LIQUID TWO-PHASE FLOW IN CIRCULAR MICROCHANNEL

Akimaro Kawahara, Kumamoto University, Japan; Michio Sadatomi, Kumamoto University, Japan; Wen Zhe Law, Kumamoto University, Japan; Akifumi Mori, Kumamoto University, Japan; Mohamed H. Mansour, Mansoura University, Egypt

This study investigated pressure drop for gas-non Newtonian liquid two-phase flows in a horizontal circular microchannel. The microchannel was a fused silica capillary tube with 0.25 mm I.D. Polyacrylamide aqueous solutions with different mass concentration, which exhibit pseudo plastic behavior with viscoelasticity, were used as non-Newtonian liquids, while the nitrogen gas was used as the test gas. Pressure drop for both single-phase liquid and two-phase gas-liquid flows were measured with a calibrated differential pressure transducer. For the single-phase liquid flows, the friction factor data have been correlated with a classical laminar flow theory if generalized Reynold number was used. For the two-phase flows, the pressure drop data have been compared with calculations by various correlations, usually used for prediction of the pressure drop in mini/micro channels as well as conventional sized channel. As the comparison results, the pressure drop data have been correlated with separated flow model by using a newly developed equation of two-phase friction factor as well as homogenous flow model with an effective viscosity accounting for non-Newtonian effects. The Lockhart-Martinelli model also predicted well the present data if an appropriate C-value in two-phase friction multiplier was adopted.

## EFFICIENT MODELLING OF BUBBLE SIZE DISTRIBUTION IN A FERMENTATION REACTOR WITH CFD

Thomas Eppinger, CD-adapco, Germany; Ravindra Aglave, CD-adapco, USA; Simon Lo, CD-ADAPCO, UK

Fermentation is a common method for producing bio-products such as alcohols and pharmaceutical compounds or alternative feedstocks. In the primary stage, presence of oxygen drives the metabolism and hence growth of the organic substance. Oxygen is supplied via air in stirred vessel type fermenters. While a certain configuration tested in a lab may result in satisfactory distribution of the dissolved oxygen (DO), large scale fermenters need careful design considerations.

Computational methods (CFD) have been reported in large amount of literature to investigate the flow patterns and consequently the gas hold up in such equipment. However, the air bubble injected vary in size throughout the reactors due to the choice of impellers, rpm, baffles, and other internals such as coils or dip tubes. The size of the air bubbles affects directly the gas hold up and the interfacial area. Therefore it is necessary to account for the varying bubble size distribution due to coalescence and breakup.

In this work, we present the use of a size distribution model to account for the bubble size distribution called S-gamma model in the commercial code STAR-CCM+. This model shows the differences in assuming constant bubble size but is not as computationally expensive as the quadrature (DQMOM/QMOM) type of models.

## EXPERIMENTAL INVESTIGATION OF AN AXIAL OIL-WATER SWIRL SEPARATOR

Robert Mudde, Delft University of Technology, The Netherlands; Harry Hoeijmakers, University of Twente, The Netherlands; Anand Ashok, Delft University of Technology, The Netherlands; Jacco Hospers, University of Twente, The Netherlands

It has been shown that bulk oil-water separation can be achieved using in-line axial swirl elements that produce substantial centrifugal forces. Due to the density difference between oil and water, an oil-rich core forms in the center of the pipe with a water-rich annulus forming on the outside (see Figure 1). In order to achieve separation efficiencies that are required by industry, further knowledge of the swirling flow is required, in particular the upstream influence of the pickup tube on the swirling flow. Using an industrial-scale flow loop at swirl, Reynolds and Weber numbers relevant to industry, three-component velocity fields and wall pressure profiles were measured at several axial locations near the pickup tube. The confined vortex's location, strength and instantaneous behavior were investigated in the single phase water flow. Subsequently, separation efficiencies are measured in the oil-water two-phase flow to assess the performance of the separator. Computations of the swirling flow have been performed by collaborators at the University of Twente using the same geometry and flow conditions, allowing for comparisons to be made between the two studies.

\*We gratefully acknowledge support from the Institute for Sustainable Process Technology ([www.ispt.eu](http://www.ispt.eu)).

## MASS TRANSFER WITH NANOFLUIDS – THE ROLE OF MICROCONVECTION

Efstathios Michaelides, TCU, USA

Mass transfer by diffusion in fluids is a slow transport process. Several experimental studies on the mass transfer with nanofluids (liquid suspensions of nanoparticles) have concluded that the addition of very small amounts (1-2%) of nanoparticles causes the mass transfer coefficient to increase by almost an order of magnitude. However, several other experimental studies, as well as all the analytical work on this subject, concluded that the presence of nanoparticles actually impedes mass diffusion.

This paper examines critically the experimental data and methods for the determination of the diffusion and mass transfer coefficients in nanofluids. It is concluded that the principal mechanism for the observed significant mass transfer enhancement is the microconvection process in fluid-solid suspensions: The Brownian movement of nanoparticles causes an advection and fluid "agitation" in the vicinity of the particles. As the particles move from one position to another, the surrounding fluid is also displaced to accommodate the movement of the particle itself as well as the added mass. If these fluid movements are not suppressed by the instrument used, they give rise to an unsteady fluid microconvection, which in turn significantly contributes to the local mass transfer enhancement.

## PROPERTIES OF FINE BUBBLES AND OF NANO PARTICLES IN PURE WATER

Yoshikatsu Ueda, *Research Institute for Sustainable Humanosphere, Japan*; Yomei Tokuda, *Institute for Chemical Research, Japan*; Naoto Nihei, *Graduate School of Agricultural and Life Science, Japan*; Shoichiro Hamamoto, *Graduate School of Agricultural and Life Science, Japan*; Yuichi Ogawa, *Graduate School of Agriculture, Japan*; Akifumi Sugiyama, *Research Institute for Sustainable Humanosphere, Japan*

Recently, fine particles such as fine bubbles (FB) and nano particles (NP) have been finding applications in various fields. In the FB field, many research articles on industrial and agricultural applications have been published. There are some articles indicating a correlation between the basic properties of FB and their effect, but there is still a need to conduct basic research, because we do not have the techniques to distinguish between a fine bubble and solid particle.

We therefore focused our study on fine particles in water, especially FB. For a theoretical approach, it is important to understand the properties of FB and contaminated nano particles. We tried to measure fine particles with a resonant mass measurement method. We could recognize FB and NP by the resonant frequency.

We also focused on an easy method of measurement of FB concentration by using turbidimetric measurement, where turbidity is an index of fine particles. We have already found correlation between the concentration of bubbles and electrical conductivity. Hence, we also conducted experimental research using standard nano particles.

## BOTTOM BUBBLING CONDITIONS ON MIXING CHARACTERISTICS IN A GAS-STIRRED LADLE

Eunju Jeong, *POSCO, Republic of Korea*; Sang-Beom Lee, *POSCO, Republic of Korea*

Flow Phenomena and mixing characteristics of the molten metal bath have been investigated according to the bottom bubbling gas conditions. The mixing time of the bath was affected from the location of gas bubbling plugs on the bottom of the ladle and numbers (from 1 to 4 plugs). For the same number of bottom bubbling plugs, the bottom bubbling in the center of the ladle showed the shortest mixing time. As increasing the distance between plugs, the mixing time of bath was increased. The location of bottom bubbling plugs contributed to the flow velocity. The plugs located to the close the ladle side wall interrupted the bath flow each other therefore, it led to the increase of mixing time. As increasing the number of bottom bubbling plugs, the mixing time of bath flow was decreased. For the number and location of bottom bubbling plug in the bath, the shortest mixing time was existed. For the case of changing the bottom bubbling flow rate periodically, the mixing time of bath flow was investigated. The periodic flow rate of the bottom bubbling also changed the mixing time in the case of using same bottom plugs. The short-periodic flow rate showed the decrease of mixing time compared with the normal bottom bubbling flow rate. When period of bottom bubbling flow rate got longer, the mixing time converged on the specific value. For the condition of periodic bottom bubbling flow rate, as total bubbling flow rate was increased, the mixing time was decrease, but the decreasing rate of mixing time was increased.

## EXPERIMENTAL CHARACTERIZATION OF INTERCHANNEL MIXING OF MULTIPHASE FLOW THROUGH A NARROW GAP

Simo Makiharju, *University of Michigan, USA*; James W. Gose, *University of Michigan, USA*; John Buchanan, *Naval Nuclear Laboratory, USA*; Alexander Mychkovsky, *Bettis Atomic Power Laboratory, USA*; Kevin Hogan, *Bettis Atomic Power Laboratory, USA*; Kirk Lowe, *Bettis Atomic Power Laboratory, USA*; Steven L. Ceccio, *University of Michigan, USA*

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 127x127 mm channels connected by a 1219 mm long and 229 mm wide 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  $4E+04$  up to  $1E+05$ . The inlet void fraction was varied from 1 to 20%. The gas was injected as nominally monodisperse bubbles with variable diameter  $O(3\text{ mm})$ . The mass transfer of both liquid and gas through the gap was determined from measurements of the mass flow rates of water and air, and tracer concentration taken at channel inlets and outlets. The void fraction, bubble diameter distribution and gas mass flux were determined at the inlets based on mass flow rate measurements of individual phases prior to gas injection, optical probe data 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 dye concentration and liquid mass flow rate after separation of gas. Imaging of fluorescent tracer dye was utilized for select conditions to examine the dynamics of the mixing.

## USING TOMOGRAPHY TECHNIQUE TO ANALYZE THE SUSPENSION OF THE SOLID PARTICLES IN A SLURRY REACTOR

Farhad Ein Mozaffari, *Ryerson University, Canada*; Prakash Mishra, *Ryerson University, Canada*

The main objective of this study was to evaluate the performance of a Maxblend impeller in the dispersion of the solid particles in a slurry reactor as a function of the impeller speed, impeller clearance, particle size, and solid concentration. To achieve this goal, electrical resistance tomography (ERT), a non-intrusive flow visualization technique, was utilized to monitor the distribution of the solid particles inside the reactor. The ERT sensor planes were positioned around the circumference of the mixing tank. To determine the axial solid concentration profile, the average solid concentration was computed for each ERT plane. The axial concentration profile was then used to calculate the degree of homogeneity for the slurry reactor as a function of the operating conditions and design parameters. ERT data were also employed to validate the CFD model developed for solid-liquid mixing operation. The results showed that the degree of homogeneity for the slurry reactor was improved with increasing the impeller speed. However, after reaching the maximum achievable homogeneity, further increase in impeller speed was not beneficial but might be detrimental. The degree of homogeneity achieved by the Maxblend impeller was higher than those for the pitched blade and Rushton impellers.

## MIXING INDUCED BY A HOMOGENEOUS BUBBLE SWARM IN PRESENCE OF EXTERNAL TURBULENCE

Elise Almeras, *University of Twente, The Netherlands*; Varghese Mathai, *University of Twente, The Netherlands*; Detlef Lohse, *University of Twente, The Netherlands*; Chao Sun, *University of Twente, The Netherlands*

We experimentally investigate the mixing of a passive scalar at high Schmidt number in a homogeneous bubble swarm in the presence of external turbulence. The experiments are conducted in the Twente Water Tunnel in which nearly homogeneous and isotropic turbulence (Taylor Reynolds number up to 300) can be generated using an active grid. We use an upward mean liquid flow in the measurement section, and the flow velocity is varied from 10 to 30 cm/s. Three millimeter diameter bubbles are injected from the bottom section of the water tunnel. The gas volume fraction is varied from 0.1% to 3%. Fluorescent dye (fluorescein sodium) is continuously injected from a lower part of the tunnel. Using a high-sensitive camera and an adapted image processing, the horizontal profile of the dye concentration is measured at three different positions from the injection point. A horizontal effective diffusivity is then calculating from the evolution of the variance of these profiles over the vertical direction. We will discuss how the spreading of the dye and the effective diffusivity are affected by the bubble-induced turbulence and the external turbulence of varying intensity that is produced by the active grid.

## MULTIPHASE EQUATIONS SUITABLE FOR THE NUMERICAL SIMULATION OF ICE PRODUCTION IN OCEAN

Vanessa Covello, *Politecnico di Milano, Italy*; Antonella Abba, *Politecnico di Milano, Italy*; Luca Bonaventura, *Politecnico di Milano, Italy*; Lorenzo Valdettaro, *Politecnico di Milano, Italy*

The aim of this work is to improve the mathematical models currently available for the simulation of ice production in turbulent sea water. The multiphase equations presented here presented the ice-seawater mixture as a dense compressible fluid. A low-Mach number asymptotic analysis has been performed to investigate the behaviour of the governing equations in the incompressible limit. The incompressibility condition has then been imposed on the zero-Mach-number equations by projecting the continuous phase equations onto the space of divergence-free vector fields. Moreover, the behaviour of sea water has been modeled by a reduced form of an equation of state that relates density, salinity, temperature and pressure [Brydon et al., JRG-Oceans, 104.C1:1537-1540, 1999], allowing us to go beyond the Boussinesq approximation. The derived system of multiphase equations is therefore suitable for the numerical simulation of ice production when the ice concentration exceeds 10-3.

\* Support from grant of the PNRA project "PANACEA The role of frazil and PANcake ice in the mass and energy budgets of the Antarctic sea ice cover" is gratefully acknowledged.

## PREDICTION OF AEROSOL SAMPLING EFFICIENCY USING A SECTIONAL DROPLET DRIFT FLUX MODEL

Edo Frederix, University of Twente, The Netherlands; Arkadiusz Kuczaj, Philip Morris Products SA, Switzerland; Markus Nordlund, Philip Morris Products SA, Switzerland; Bernard Geurts, University of Twente, The Netherlands

Computational modeling of evolving dense aerosols in complex geometries is a challenging subject of investigation, in particular in the case of multicomponent, compressible mixtures. We developed a Eulerian framework, which is computationally more attainable than a Lagrangian formulation. The aerosol size domain is discretized using the sectional approach, while the physical domain is treated with a finite volume method. To study the influence of non-isokinetic aerosol flow on deposition, we developed a sectional, size-dependent droplet drift flux model, in which droplets are subject to inertial, gravitational and drag forces, embedded in a compressible mixture. The new model has been validated against existing experimental data of aspiration efficiency, in which a difference between the main flow velocity and the sampling flow velocity leads to rotational flow and a corresponding size-dependent droplet drift. This causes a difference between the input size distribution and the measured one. In particular, we pay attention to the correct prediction of the flow and aerosol field, by studying the grid dependence of the solution. Moreover, we quantify how the aspiration efficiency depends on the ratio of main and sampling flow velocity. Comparison is made with physical experiments showing the level of agreement of the Eulerian approach.

## NUMERICAL MODELING OF CONVECTIVE FLOW IN COMPLEX HYDROCARBON LIQUID UNDER RADIO-FREQUENCY HEATING

Liana Kovaleva, Bashkir State University, Russian Federation; Victor Kireev, Mavlutov Institute of Mechanics, Russian Federation; Ayrat Musin, Bashkir State University, Russian Federation

The study of convective heat and mass transfer in multiphase multicomponent media such as natural (heavy oil and bitumen) or manmade (sludge and petroleum products) hydrocarbon systems is associated with a number of problems arising in the oil and gas industry. One of the problem is the fast and effective technique for heat treatment to reduce the viscosity and to increase the fluidity of hydrocarbon systems. The urgency of radio-frequency electromagnetic method is connected with the fact that for these systems, it is important not only to reduce the viscosity by heating, but to destroy stable structures of high-molecular hydrocarbon compounds, which is achieved by the effect of electromagnetic radiation. In the present work the results of numerical modeling of thermal convection in high viscosity heterogeneous hydrocarbon liquid with temperature-dependent physical properties (viscosity, thermal conductivity and others) under radio-frequency electromagnetic field heating are presented. The efficiency of the radio-frequency field impact in comparison to the induction heating is proved. It is shown that radio-frequency electromagnetic field led to an appearance of multi-vertex patterns in the whole volume of the liquid. Thus, radio-frequency electromagnetic heating provides a more homogeneous redistribution of heat and prevents local overheating of the liquid.

\* Support from the Ministry of Education and Science of the Russian Federation (grant #3.1251.2014/K) is gratefully acknowledged.

## PARTICLE IMAGE VELOCIMETRY MEASUREMENTS OF INJECTED PARTICLE PLUMES IN A TWO-DIMENSIONAL VERTICAL CHANNEL

Matthew Yates, University of Nottingham, UK; David Hann, University of Nottingham, UK; Buddhikha Hewakandamby, University of Nottingham, UK; Barry James Azzopardi, University of Nottingham, UK

Turbulent dispersed flows are common in natural and industrial settings but are one of the least understood areas in multiphase systems. To develop models, information required includes the influence of phase coupling, dispersed phase distribution, and statistical descriptions. To quantify some of these parameters, experiments have been conducted at the University of Nottingham multiphase flow lab. Particle Image Velocimetry (PIV) measurements were made of particle injections at the entrance of a two-dimensional vertical channel. Water and fluorescent hydrogel particles were used as the dispersed phase system. Investigations were made for a range of particle injection velocities, dispersed phase fractions, channel Reynolds numbers and for varying distances downstream of the injection point. Velocity PDFs are presented to describe dispersed phase behavior, with an extension to multi-dimensional forms to account for particle pair correlations. Particle distribution data is compared for different downstream distances. Particle collisions at the injection point are analyzed using measurements with high time-resolution. Continuous phase PIV data is used to find local velocity correlations between both phases.

\* Support from Chevron as part of TMF consortium is gratefully acknowledged

## THE IMPACT OF DIFFERENT FINE GRID SIMULATIONS ON THE SUB-GRID MODIFICATION FOR GAS-SOLID DRAG

Simon Schneiderbauer, Johannes Kepler University, Austria; Stefan Pirker, Johannes Kepler University, Austria

Fluidized beds are widely used in a variety of industrially important processes. During the last decades the analysis of the hydrodynamics of fluidized beds through numerical simulations has become increasingly common, where the two-fluid model (TFM) approach has proven to provide fairly good predictions of the hydrodynamics of gas-solid flows. However, due to computational limitations a fully resolved simulation of industrial scale reactors is still unfeasible. It is therefore common to use coarse grids to reduce the demand on computational resources, which inevitably neglects small (unresolved) scales. Therefore, many sub-grid drag modifications have been put forth by academic researchers to account for the effect of small unresolved scales in this case.

This contribution demonstrates that the setup of the fine grid two-fluid simulations, which are commonly used to derive those sub-grid modifications, considerably affects their functional form. In particular, we investigate the impact of the frictional stress model, the superficial gas velocity, the value of the maximum packing ratio, the particle diameter and density as well as the geometrical configuration (full fluidized bed against periodic box) on the sub-grid drag modification. Our results clearly show that the type of fine grid simulation directly changes the functional form of the constitutive relation for the macroscopic drag when using the filtered solids volume fraction and the filtered slip velocity as independent variables.

## AN EXPERIMENTAL AND NUMERICAL VALIDATION STUDY OF PARTICLE LADEN SUPERSONIC FLOWS

Arthur Rudek, Hochschule Darmstadt and Dublin Institute of Technology, Germany; Gerald Russ, Hochschule Darmstadt, Germany; Barry Duignan, Dublin Institute of Technology, Ireland

An experiment utilizing high speed cameras (HSC) in a single-exposure multi-frame setting is presented, which provides validation data for macroscopic particle transport simulations in a convergent divergent nozzle. Solid spherical particles made from Polyoxymethylen (POM) with diameters from 1.5 to 3.0 mm are photographed passing through the transparent test nozzle. The particles are transported by compressed air at 2, 4 and 6 bar gauge pressure. All recordings are made using a calibrated HSC. An intensity-based image postprocessor is used to generate particle diameter and velocity information along the nozzle. The particle tracks are correlated and discussed with respect to possible uncertainties. Furthermore, the experimental outcome is used for the validation of a numerical simulation of the given experimental setup. All simulations are realized by means of the commercial Finite Element (FE) based Finite Volume Method (FVM) solver Ansys CFX 15.0. Initially, the methodology for pure air flow simulation is validated against experimental data from literature. Particle laden simulations are realized by means of Euler-Lagrangian particle tracking. Results obtained from a range of approaches varying the phase coupling method and particles ODE of motion are discussed with reference to the experimental results.

## TURBULENCE ATTENUATION IN PARTICLE-LADEN FLOW

Bert Vreman, AKZO Nobel, The Netherlands

Downward gas-solid flow in smooth and rough vertical channels has been investigated by means of point-particle direct numerical simulation (Vreman 2015, J. Fluid Mech. 773, 103-136). Two-way coupling and inter-particle collisions are included. Simulation results are compared to experimental results from literature. The rough walls are modeled by fixed layers of tiny spherical particles with diameter much smaller than the viscous wall unit. Since the non-uniformity of the mean particle feedback force increases with wall roughness, the effect of the non-uniform part of the mean feedback force is investigated in detail. For both smooth and rough walls, the non-uniform part of the mean feedback force is shown to contribute significantly to the particle-induced turbulence attenuation.

In addition, results of ongoing work on turbulence attenuation in particle-laden flow simulated by particle-resolved direct numerical simulation using an overlapping grid method are presented.

## TURBULENT HEAT FLUXES UNDER THE INFLUENCE OF EVAPORATING DROPLETS

David Richter, University of Notre Dame, USA; Brian Helgans, University of Notre Dame, USA

Direct numerical simulations of an Eulerian air phase are performed which are dynamically and thermodynamically coupled to Lagrangian water droplets dispersed throughout the flow. With these simulations we aim to understand energy transport in turbulent, wall-bounded flows under the influence of a liquid dispersed phase which is undergoing evaporation and condensation, thus providing much-needed insight into a wide variety of industrial and geophysical systems. In particular, we focus on wall-normal fluxes of total enthalpy in turbulent planar Couette flow, and how this energy is partitioned between sensible and latent components with and without the presence of droplets. The simulations demonstrate clearly that droplets undergoing phase change have substantial effects on both sensible and latent heat fluxes in this system, but that this influence is opposite in sign (evaporating droplets cool the surrounding air, condensing droplets heat the surrounding air) and partially cancels. Furthermore by altering the prescribed boundary conditions, we show that droplets can induce energy fluxes which are counter to the background gradients. Finally we show that the ability of droplets to modify thermodynamic fluxes is highly sensitive to their inertia, as indicated by the droplet Stokes number, due to clustering and turbophoresis. These effects, as well as their implications for several geophysical flows, will be discussed in detail.

The authors acknowledge the National Science Foundation for funding under Grant #AGS-1429921

## MODELING TURBULENT LIQUID/GAS PHASE INTERFACE USING A VOF BASED DUAL-SCALE LES SUBGRID MODEL

Marcus Herrmann, Arizona State University, USA

In this paper, a dual-scale modeling approach is presented to describe turbulent two-phase interface dynamics in a Large Eddy Simulation (LES) type spatial filtering context. To close the unclosed terms related to the phase interface arising from filtering the Navier-Stokes equation, a resolved realization of the phase interface dynamics is explicitly filtered. This resolved realization is maintained on a high-resolution over-set mesh using a Refined Local Surface Grid approach (Herrmann, 2008) employing an un-split, geometric, bounded, and conservative Volume-of-Fluid (VOF) method (Owkes & Desjardins, 2014). The required model for the resolved realization of the interface advection velocity includes the effects of sub-filter surface tension, dissipation, and turbulent eddies. Results of the dual-scale model are compared to fully resolved simulations of surface tension dominated flows including the oscillation of sub-filter liquid drops, sub-filter Raleigh-Plateau stable and unstable liquid columns, and the breakup of a viscous liquid ligament. Furthermore, results of the dual-scale model are compared to recent direct numerical simulations of interfaces in homogeneous isotropic turbulence (McCaslin & Desjardins, 2015) at varying Weber numbers.

\* Support from NASA grant NNX15AV06A is gratefully acknowledged.

## ROTATION OF NON-SPHERICAL PARTICLES IN TURBULENCE - EFFECT OF SIZE ON KINEMATIC STATISTICS

Evan Variano, University of California, Berkeley, USA; Nimish Pujara, University of California, Berkeley, USA; Nimish Bordoloi, University of California, Berkeley, USA

We explore the ways in which particle shape affects particle motion, focusing specifically on how particle rotation is divided among the principle axes of non-spherical particles. We focus on idealized axisymmetric particles shaped as rods, discs, and spheroids. Their size, shape, and inertia are chosen so as to explain the physics of aspherical-particle motion that will be relevant for natural particles such as plankton, sediment, or aggregates. We find that shape has only a very weak effect on total angular velocity, but if we analyze rotation in a particle's local frame (i.e. the particle's principle axes of rotation), then particle shape has a strong effect on rotation. In the local frame, rotation is described by two components: tumbling and spinning. We find that rod-shaped particles spin more than they tumble, and we find that disc-shaped particles tumble more than they spin. Such behavior is indicative of how particles respond to the directional influence of vortex tubes in turbulence, and such response has implications for particle motion other than rotation. Understanding particle alignment is relevant for predicting particle-particle collision rates, particle-wall collision rates, and the shear-driven breakup of aggregates. We discuss these briefly in the context of what can be concluded from the rotation data discussed above.

\* Support from Gulf of Mexico Research Initiative and National Science Foundation is gratefully acknowledged.

## INTERACTION BETWEEN A FINITE-SIZE BUOYANT BUBBLE AND TURBULENCE

Aurore Loisy, Université Claude Bernard Lyon 1, France; Aurore Naso, CNRS, France; Peter Spelt, Laboratoire de Mécanique des Fluides et d'Acoustique, France

Bubble dynamics in turbulent flows has been extensively studied using the point-bubble approximation. But when the carrier flow varies on length scales smaller than the bubble size, this approximation is no longer appropriate. Recently the interaction between turbulence and large objects has been examined by a number of investigators. But in most of these studies, only rigid particles were considered.

To contribute filling this gap, we investigate the interaction between a large buoyant bubble and otherwise homogeneous isotropic turbulence using 3D DNS so that all the scales present in the two-phase flow are resolved. The bubble is deformable and its diameter is of the order of Taylor length-scale size.

A significant reduction of the bubble rise velocity compared to its value in still liquid is observed, similarly to what is obtained for point bubbles. Preferential sampling of high vorticity and downward velocity regions is compared to prior results for microbubbles. Turbulence modulation by large bubbles is investigated through Eulerian statistics of the liquid phase conditioned on the presence of a bubble. Results are compared to those obtained for solid particles.

\* Supports from ANR (TEC2), GENCI-CINES (Grant 2015-x20152b6893) and FLMSN are gratefully acknowledged.

## STUDY OF PARTICLE-TURBULENCE INTERACTIONS IN TURBULENT PARTICLE-LADEN FLOW BY INTERFACE-RESOLVED SIMULATIONS

Lian-Ping Wang, University of Delaware, USA; Cheng Peng, University of Delaware, USA

The nature of modulation of the carrier-phase turbulence by finite-size inertial particles depends on many factors including scales and geometric configurations of the carrier-phase flow and particle characteristics such as size, density, mass loading. Finite-size particles introduce both local viscous dissipation and kinetic energy production. In this work we apply the mesoscopic lattice Boltzmann method to conduct interface-resolved simulations of particle-laden homogeneous-isotropic turbulence, and the resulting data are used to examine flow modulation by finite-size solid particles. We will focus on two aspects here. The first is the flow modulation near a finite-size solid particle at very low volume fraction such that only the particle-turbulence interaction is important. The second is the hydrodynamic force and torque acting on the finite-size solid particle under the similar conditions. The size of the solid particle is varied with respect to the flow Kolmogorov scale. The key question here is how to describe and properly formulate the problem of flow modulation and hydrodynamic force when a finite-size particle is involved. We will present flow statistics conditioned on the particle surface, hydrodynamic force, and local flow seen by the solid particle. Our results will be compared to previous studies on the similar problem.

## RELATIVE ROTATIONAL MOTION BETWEEN RIGID FIBERS AND FLUID IN TURBULENT CHANNEL FLOW

Cristian Marchioli, University of Udine, Italy; Lihao Zhao, Norwegian University of Science and Technology, Norway; Helge Andersson, Norwegian University of Science and Technology, Norway

In this study the rotation of small rigid fibers relative to the surrounding fluid in wall-bounded turbulence is examined by means of direct numerical simulations coupled with Lagrangian tracking. Statistics of the relative (fiber-to-fluid) angular velocity, referred to as slip spin in the present study, are evaluated by modeling fibers as prolate spheroidal particles with Stokes number,  $St$ , ranging from 1 to 100 and aspect ratio,  $r$ , ranging from 3 to 50. Results are compared one-to-one with those obtained for spherical particles ( $r=1$ ) to highlight effects due to fiber length. The statistical moments of the slip spin show that differences in the rotation rate of fibers and fluid are influenced by inertia, but depend strongly also on fiber length: Departures from the spherical shape, even when small, are associated with an increase of rotational inertia and prevent fibers from passively following the surrounding fluid. An increase of fiber length, in addition, decouples the rotational dynamics of a fiber from its translational dynamics suggesting that the two motions can be modeled independently only for long enough fibers. We also examine the transfer of rotational energy that occurs from the fluid to the fiber: Fibers appear to orient themselves minimizing the transferred energy, this mechanism being strongly affected by fiber length.

\* Support from COST Action FP1005 "Fiber suspension flow modeling" is gratefully acknowledged.

## COMBINED MEASUREMENTS OF FLOW FIELD AND RIGID, INERTIAL FIBRE ROTATION/TRANSLATION IN NEAR HOMOGENEOUS ISOTROPIC TURBULENCE

Lilach Sabban, Technion - Israel Institute of Technology, Israel; Asaf Cohen, Technion - Israel Institute of Technology, Israel; Rene Van Hout, Technion - Israel Institute of Technology, Israel

Here begins the body of the abstract.

Time resolved, planar particle image velocimetry (TR-PIV, 3kHz) and two-orthogonal view, digital holographic cinematography (2kHz) were used to measure the flow field in the vicinity of inertial fibres as well as the (3D) fibre trajectories/orientation dynamics in near homogeneous isotropic air turbulence (HIT) at dilute number densities. HIT (Taylor scale Reynolds number 144) was generated in the centre of a 403cm<sup>3</sup> cube by eight woofers mounted on each of its corners. Two different batches of nylon fibres having a nominal length of 0.5mm and diameter of 14 and 19µm, respectively, were released from the chamber's top. Fibres had Stokes numbers of order one and results showed that they resided at the periphery of vortices in regions of low vorticity. Although instantaneous fibre Reynolds numbers based on fibre length were below 30, turbulence enhancement occurred for the heaviest fibres (19µm). Results further indicated a weak correlation between the fibre's in-plane orientation angle and that of the extensional strain rate. Similar as for non-inertial fibres at the same aspect ratio, the present results indicate that the fibres have non-random orientations. Furthermore, the normalized, mean square rotation rate of the inertial fibres decreased in comparison to non-inertial fibres.

\* Support from Israel Science Foundation grant 1596/14 is gratefully acknowledged.

## TWO-PHASE FLOW AND TOTAL PRESSURE IN HIGH-VELOCITY FREE-SURFACE CHUTE FLOWS

Gangfu Zhang, The University of Queensland, Australia; Hubert Chanson, The University of Queensland, Australia

Stepped spillways have been used as flood release facilities for several centuries and a key feature is the intense free-surface aeration observed in both prototype and laboratory. Herein the air bubble entrainment and turbulence were investigated in a stepped spillway model, to characterise the interplay between air bubble entrainment and turbulence, and the complicated interactions between mainstream flow and cavity recirculation motion. New experiments were conducted in a large steep stepped chute ( $\theta = 45^\circ$ ,  $h = 0.10$  m,  $W = 0.985$  m). Detailed two-phase flow measurements were conducted for a range of discharges corresponding Reynolds numbers between 3(105 and 9(105). The total pressure, air-water flow and turbulence properties were documented systematically in the mainstream and cavity flows. Energy calculations showed an overall energy dissipation of about 50% regardless of the discharge. Overall the data indicated that the bottom roughness (i.e. stepped profile) was the determining factor on the energy dissipation performance of the stepped structure, as well as on the longitudinal changes in air-water flow properties.

\* Support from grant ARC DP120100481 is gratefully acknowledged.

## MODELLING OF TURBULENCE MODULATION IN BUBBLY FLOWS WITH THE AID OF DIRECT NUMERICAL SIMULATION

Tian Ma, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Thomas Ziegenhein, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Dirk Lucas, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Jochen Frohlich, Technische Universität Dresden, Germany

Several technical flows include a continuous liquid and a dispersed gaseous phase. The turbulence of the liquid phase influences the local distribution of the dispersed phase, bubble coalescence and breakup and other important flow characteristics. Because of the importance of turbulence it is necessary to consider its modification by bubbles. The conservation equation for the turbulent kinetic energy (TKE) of the two-phase flow besides the single-phase-like terms contains an additional interfacial term representing the interfacial energy transfer.

The analysis of the TKE budget from Direct Numerical Simulation data can be used for the development of new two-phase Reynolds-averaged Navier-Stokes (RANS) models. In the present work, Euler-Euler modelling of vertical bubbly channel flow is performed using the Shear Stress Transport model. In this flow, the bubble-induced turbulence (BIT) is dominating, i.e. the interfacial term compensates nearly the total dissipation in the TKE equation. New coefficients for the source terms in the k-equation and the  $\omega$ -equation are derived from DNS data. With these improved source terms in the RANS model, the results show a good agreement with the reference data. Furthermore, a new BIT model dependent on the bubble Reynolds number is proposed and validated in a more complex bi-dispersed bubbly channel flow.

## MEASURING ROTATION AND ALIGNMENT OF ANISOTROPIC PARTICLES IN TURBULENCE

Greg Voth, Wesleyan University, USA

Flows with anisotropic particles in a turbulent fluid appear in many settings including icy clouds, bioreactors, paper manufacturing, and marine plankton. Underlying these complex applications is a foundational problem concerning the translation and rotation of small, rigid, density matched, nonspherical particles at low concentrations in a high Reynolds number turbulent fluid flow. Study of this simple problem has uncovered a surprisingly rich phenomenology in which particles are strongly aligned by the turbulence and display rotations that depend strongly on particle shape. We have experimentally measured the 3D translation and rotation of particles with spheroidal shapes ranging from disks to rods in a turbulent flow between oscillating grids. We have also studied the rotations of particles with chiral ends that show a preferential rotation direction in isotropic turbulence. The rotation and alignment of each of these shapes can be understood as a consequence of the Lagrangian strain experienced by the particles over the past few Kolmogorov times.

\* Support from US NSF grant DMR-1508575 is gratefully acknowledged.

## EXPERIMENTAL STUDY ON ATOMIZATION OF ALLOPHANE WITH MICRO BUBBLES

Yuzo Inoue, University of Tsukuba, Japan; Yoshihiro Arai, University of Tsukuba, Japan; Akiko Kaneko, University of Tsukuba, Japan; Fumio Takemura, National Institute of Advanced Industrial Science and Techno, Japan; Masatoshi Ike, Aptex LLC, Japan; Tetsuya Kanagawa, University of Tsukuba, Japan; Yutaka Abe, University of Tsukuba, Japan

A coagulant treatment is one of the countermeasures for long-term turbid water in dams which worsens the environment of the downstream river. Since allophane is a coagulant which is derived from nature, it is thought that allophane has little burden on the environment. In order to put coagulant treatment into practical use, there are two important points, one is the establishment of the atomization technique for coagulants and the other is effective diffusion technique in closed water. Thus, this study focuses on the atomization technique. We use the venturi type micro bubble generator, which produces a large number of micro bubbles with a diameter of 10 µm – 1mm thanks to the bubble collapse caused by a pressure difference in the diverging section of the tube, as an effective atomization technique of allophane. It is expected that allophane will atomize as in the case of micro bubbles. We throw water, air, and allophane into a venturi tube, and conducted optical measurement of the particle size of allophane both before it enters a venturi tube and after it leaves the tube. Furthermore, we create a map of particle size generated where the parameters are flow conditions, and reveal the flow conditions and patterns, and reveal the flow conditions for an effective atomization.

## A SIMPLIFIED ENERGY DISSIPATION BASED MODEL OF HEAT TRANSFER FOR SUBCOOLED FLOW BOILING

Dariusz Mikielewicz, Gdansk University of Technology, Poland; Jaroslaw Mikielewicz, Institute of Fluid Flow Machinery, Poland

A model is presented based on energetic considerations for subcooled flow boiling heat transfer. The model is the extension of authors own model developed earlier for saturated flow boiling and condensation. In the former one we used the heat transfer coefficient for the liquid single-phase as a reference level, due to the lack of the appropriate model for heat transfer coefficient for the subcooled flow boiling. That issue was a fundamental weakness of the former model. The purpose of present investigation is to fulfill this drawback. Now the reference heat transfer coefficient for the saturated flow boiling in terms of the value taking into account the subcooled flow conditions. The wall heat flux is based on partitioning and constitutes of two principal components, namely the convective heat flux and partial evaporation heat flux of the liquid replacing the detached bubble. Both terms are accordingly modeled. The convective heat flux is regarding vapour bubbles travelling longitudinally and the liquid moving radially – liquid pumping. The results of calculations have been compared with some experimental data from literature showing a satisfactory consistency.



## EXPERIMENTAL ANALYSIS OF CONDENSATION INDUCED WATER HAMMERS IN HORIZONTAL PIPES

Christian Urban, Hamburg University of Technology, Germany; Marko Hoffmann, Hamburg University of Technology, Germany; Michael Schlueter, Hamburg University of Technology, Germany

Instant condensation procedures are regarded as dangerous safety issues in various industrial applications, e.g. start-up processes of steam pipes in chemical plants and the emergency cooling system of nuclear reactors. In this context the so called condensation induced water hammer (CIWH), where a subcooled water stream is brought into contact with saturated steam in a confined space, is one of the most harmful forms of occurrence. In this case saturated steam is brought into contact with a subcooled liquid flow through an adequately large exchange surface area, resulting in a two-phase flow pattern that promotes inclusion of relatively large steam pockets. If these pockets collapse suddenly the surrounding water slugs collide, leading to a huge pressure rise in the system and causes the breakdown of pipes and connected instrumentation. By means of an experimental setup and based on more than 175 experiments a detailed analysis of CIWHs in a horizontal pipe with an angle of inclination of  $1.4^\circ$  was performed. These experimental results enable conclusions regarding the probability of CIWHs with the corresponding maximum pressure value, the exact position of nucleation sites of steam implosion, the derivation of critical parameter combinations, e.g. the degree of subcooling in connection with the injection flow rate as well as the elaboration of a reasonable criteria for the prevention of CIWH.

\*This work was kindly supported by the Federal Ministry of Education and Research (BMBF) under grant number 02NUK011D.

## NUMERICAL SIMULATION OF MULTIPLE DROPLET IMPINGEMENT USING DIFFUSE INTERFACE MODEL

Yosuke Sugitani, Mie University, Japan; Koichi Tsujimoto, Mie University, Japan; Toshihiko Shakouchi, Mie University, Japan; Toshiyuki Ando, Mie University, Japan

Droplet impingement technology is applied to spray printing, spray drying, powder producing and 3D printing. An advancement of these technologies need flow control based on the elucidation of a detailed mechanism of impingement. Until now, although a large number of knowledge about single droplet impingement is acquired, however, multiple droplet impingement is not enough to make clear. In general, although numerical analysis is effective in the elucidation of mechanism of this phenomena, however, to reproduce the detailed flow phenomena of multiple droplets, high computational power is inevitably required. Also in the impingement problems, a numerical scheme should be capable of easily deal with complicated interface topology change such as coalescence, breakup and so on. In the present paper, we adopt DIM (diffuse interface model) as an interface tracking method to well respond to the topological change of interface. In addition the computational code (in-house) is parallelized using GP-GPU (General Purpose Graphics Processing Unit) to conduct a large-scale computation. Using our developed code, single droplet impingements are examined to validate the performance of the DIM. Further multiple droplet impingements are investigated under the various flow-control parameter.

## INFLUENCE OF GAS AND DROPLET PROPERTIES ON DROP-DROP COLLISION OUTCOMES

Ghoku Krishnan, University of Virginia, USA; Eric Loth, University of Virginia, USA

The collision of two spherical droplets in a gas is considered in terms of the five primary phenomenological outcomes: slow coalescence (SC), bounce (B), fast coalescence (FC), reflexive separation (RS), and stretching separation (SS). The boundaries that separate these outcomes were investigated herein in terms of droplet viscosity and surface tension as well as gas pressure and density. Gas effects are not accounted for previous models, but can be important for hydrocarbon drops in pressurized sprays associated with many fuel systems. For slow coalescence/bouncing (SC/B), increasing droplet viscosity and gas pressure were found to increase the probability of a bouncing outcome of the collision. For the B/FC boundary, increasing droplet viscosity and gas density were also found to increase bouncing probability. In both cases, the variations can be explained in terms of the stability of the gas layer that develops between the droplets. Additionally, the Brazier-Smith model for the FC/SS boundary was modified to increase robustness for a wide range of droplet viscosities. In general, the proposed models reasonably predicted collision outcomes for a large variety of gas pressures and densities as well as droplet viscosities and surface tensions.

## PARTICLE PROPERTIES AND PHASE TRANSITION IN AN INDUCTIVELY COUPLED PLASMA SPRAYING

Lijuan Qian, China Jiliang University, China; Jianzhong Lin, China Jiliang University, China

Suspension plasma spray is a promising technique for nano-structured coatings and nano-powder synthesis where nano-particles are injected into the plasma jet with the help of liquid precursors. When the particles fly through the plasma flame, their mass, momentum and energy will dramatically change due to the interaction with the flame. A comprehensive model was developed to investigate the suspension spraying in the radio frequency inductively coupled plasma torch. The model is based on hybrid Eulerian/Lagrangian coordinate system to illustrate the suspension behavior, such as suspension droplets collision, heating and evaporation; nano or agglomerate particles heating, melting and evaporation. Special considerations are directed to the suspension droplets collision, non-continuum effects and the influence of evaporation on heat transfer. After validation with experimental data, the comprehensive particle model is used to predict the trajectory, velocity, temperature and size of the in-flight nano- or agglomerate particles. The effects of the atomization operating conditions and the characteristics of the non-Newtonian liquid on the particle acceleration and heating process are investigated. The parameters that have a significant influence on the spray process are identified.

\* Support by the National Natural Science Foundation of China with Grant No.11372006 is gratefully acknowledged.

## NOVEL MEASUREMENT FOR DISPERSED DROPLETS THROUGH A SINGLE TIP OPTICAL FIBER PROBE WITH POST-SIGNAL PROCESSING

Hiromu Shimizu, Shizuoka University, Japan; Takayuki Saito, Shizuoka University, Japan

An optical fiber probing is one of laser-based techniques to measure dispersed bubbles or droplets in gas-liquid two-phase flows. Although a single-tip optical fiber probe (S-TOP) possesses satisfactory performance in simultaneous measurement of velocity, chord length and number density of a sub-millimeter droplet, a random error occurs due to uncertainty of the S-TOP's penetration position into the droplet. To overcome this, in our past study, we suggested the detection method of the position penetrated by the S-TOP in droplet measurement, using post-signal method.

In this study, we have rapidly improved the signal processing against the post-signal detection, and have verified the accuracy of the S-TOP measurement in a spray flow. The spray flow with a full-conical pattern including spherical and ellipsoidal droplets generated by pressure injection nozzle was employed. Using the glare point technique rapidly improved through full-3D ray-tracing simulation, the reference data for the verification were obtained. As a result, the measurement accuracy of a droplet velocity and chord length through the S-TOP was remarkably improved by using this post-signal processing. Additionally, we demonstrated high performance of both laser-based measurement techniques by calculating the beam propagation in the droplet, based on 3D ray-tracing simulation.

## AN EXPERIMENTAL STUDY OF HEAT TRANSFER MECHANISM IN GRANULAR FLOWS

Shu-San Hsiau, National Central University, Taiwan; Shang-Yu Liu, National Central University, Taiwan; Shih-Hao Chou, National Central University, Taiwan

Heat transport characterization of the granular materials has a considerable importance linked to many industrial applications. There are two basic mechanisms of granular heat transfer: conduction and convection. Granular conduction is transferred through the contact between particles. The thermal energy is conducted by temperature difference between particles. The heat transfer rate is dependent on conductivity of material, contact area and contact radius. Granular convection has two different ways. One is particle exposed to a uniform interstitial fluid, and the heat transfer from the surface is exchanged through the fluid. Another mechanism is similar to forced convection: The particles move and flow due to external excitations such as vibration, gravity, etc. Thermal energy transferred through the mixing between hot & cold particles. In this study we experimentally investigated the heat transfer behaviors of a granular avalanche flow in a quasi-2D rotating drum. The drum circular wall was subjected in a heat input condition and the thermal energy could be transferred into the granular media. The experimental instruments include the thermal couples and a thermal imaging system. The mechanisms of heat transfer phenomena inside granular flow will be carefully studied.

\* Support from grant MOST-103-2221-E-008-042-MY3 from Ministry of Science and Technology of Taiwan is gratefully acknowledged.

## FLUID-GRANULAR MEDIA MULTI-PHASE MODELLING USING THE SMOOTHED PARTICLE HYDRODYNAMICS METHOD

Kamil Szwec, *The Szewalski Institute of Fluid-Flow Machinery, PAF, Poland*

Problems involving interactions of fluid and granular media phases are active research in the fields of geo-mechanics and natural hazard management. Particular attention is paid to understand the processes and learn how to predict the results of interaction between water waves and seabed (formation of tsunamis, transport of sediments). The present grid-based approaches suffer from many issues related to the Eulerian origin. On the other hand, the Distinct Element Method, the approach commonly used in modelling of granular media, requires coupling with other CFD approaches. Hence, There is a need for a new, accurate and efficient approach. In recent decade, many so-called meshless methods have been developed. One of the most mature and commonly used approaches is Smoothed Particle Hydrodynamics. The SPH method is a particle-based Lagrangian method for fluid-flow simulations. In this work, the fundamental concepts of this approach are first briefly recalled. Then, the ability to accurately model interactions of fluid-flow and granular materials are studied. For this purpose sets of numerical calculations (2D and 3D) of fundamental problems of water and granular media collapses are performed. In order to improve the numerical efficiency, the Graphics Processing Unit implementation is discussed.

\* This research has been funded by National Science Centre, Poland via grant Opus 6 no DEC-2013/11/B/ST8/03818.

## COMPUTATIONAL MODELING OF DROPLET COALESCENCE IN COMPLEX GEOMETRIES

Sheng Wang, *Cornell University, USA; Gilou Agbaglah, Cornell University, USA; Olivier Desjardins, Cornell University, USA*

A liquid-gas coalescer is a non-woven fibrous sheet, usually composed of arrays of cylindrical rods, which is used to remove small liquid droplets from gas process streams. The coalescence process in the presence of rods is dominated by surface tension and contact line motion at fairly low Reynolds number. The coalescence efficiency is closely linked to the drop size distribution downstream of the coalescer. Here we present a conservative, robust, and efficient computational framework for modeling this flow. Our simulation approach combines an accurate conservative level set method to capture the interface position, an immersed boundary method to represent the solid boundary and a macroscopic moving contact line model. The performance of the proposed approach is assessed through several simulations. The steady states of a drop on a flat plane and on a circular cylinder without gravity are used to validate the equilibrium contact angle. The contact line dynamics is validated through the migration of a drop on an inclined plane. The conservation errors of mass and momentum are shown to remain small for all the simulations. The framework is then applied in a 3D simulation of a droplet laden flow through a collection of cylindrical rods. The statistics of drop size is calculated to assess the performance of the coalescer model.

## PREDICTION OF AIR RELEASE IN HYDRAULIC RESERVOIRS USING LAGRANGIAN PARTICLE TRACKING

Marco Longhitano, *RWTH Aachen University, Germany; Hubertus Murrenhoff, RWTH Aachen University, Germany*

The presence of an air phase in a hydraulic fluid is inevitable. Entrained air forms inside the hydraulic components during gas-cavitation and strongly affects the performance of the hydraulic system. The hydraulic reservoir performs the function of releasing the accumulated air to the surroundings. The air bubbles escape from the free surface of the fluid. Techniques to measure the air release efficiency of the hydraulic reservoirs are expensive. For this reason, in recent years, multiphase CFD models have been discussed in the literature to simulate the air release in hydraulic reservoirs. These include Euler-Euler and Euler-Lagrange strategies. In Euler-Euler strategy both the air and the fluid phase are treated as interpenetrating and continuous phases. On the other hand, the Euler-Lagrange approach tracks Lagrangian air bubbles within an Eulerian phase. In this paper, a comparison of these two methods is presented utilising the Open Source tool OpenFOAM. In-house drag correlations have been implemented in the code to simulate the air behaviour in mineral oils. The simulations have been conducted varying the oil flow rate and the air load for different geometries. The simulation results are expressed in terms of an air release efficiency and validated with experimental results.

## SATURATION LENGTH SCALE ON WALL PRESSURE DISTRIBUTION DURING A GRANULAR DISCHARGE FROM A NARROW SILO

Mingjer Jiang, *National Taiwan University, Taiwan; Fuling Yang, National Taiwan University, Taiwan*

This work exploits serial load cells to measure the depth profile of lateral pressure  $p(t,z)$  during granular discharge from a rectangular silo of flat bottom and different widths  $L$  with  $t$  and  $z$  denoting time and depth from free surface. As a reference, static profile is measured to follow Janssens' model,  $p(0,z) = (\phi_0 g z [1 - \exp(-z/\lambda)])$  with initial volume fraction,  $\phi_0$ , material intrinsic density,  $\rho$ , and a characteristic length  $\lambda$ . Right after the opening of silo, distinctive variations of  $p(t,z)$  is found. In the shallow layer with  $z < \lambda$ ,  $p(t,z)$  remains linear but at a magnitude slightly higher than the static  $p(0,z)$ , possibly due to dilatancy effect upon descending of the layer. In deeper bed with  $z > \lambda$ ,  $p(t,z)$  drops greatly from the static saturated value at a degree growing with  $z$  which may results from locally lower  $\phi$  in the core flowing zone. Such a two-layer dynamic pressure distribution suggests distinctive nature of internal force chain network around  $z = \lambda$  which speculation is confirmed by the distinctive frequency spectrum of the load cell signals. Finally, very different  $p(t,z)$  is observed for discharge from a very wide silo.

## NUMERICAL INVESTIGATION OF AN AXIAL OIL-WATER SWIRL SEPARATOR

Jacco Hospers, *University of Twente, The Netherlands; Harry Hoeijmakers, University of Twente, The Netherlands*

Within the scope of an industry funded project, an in-line, axial, liquid-liquid swirl separator has been investigated. In particular, the effect of the core vortex stability has been studied, leading to an optimized design of the in-line swirl separator. A numerical model to investigate the behavior of the confined vortex and its influence on the separation efficiency is presented. Several geometric configurations and operational parameters have been investigated in order to arrive at an optimal configuration. These numerical investigations, performed at the University of Twente, are validated using experiments performed in an industrial-scale test facility, at the Delft University of Technology. Where possible, similarities and discrepancies between the numerical and experimental results are discussed.

\* Project is funded by ispt (the Institute for Sustainable Process Technology), which is gratefully acknowledged.  
<http://www.ispt.eu>

## UNCERTAINTY QUANTIFICATION OF THE TWO-FLUID MODEL AND FLOW PATTERN TRANSITION BOUNDARIES PREDICTIONS FOR THE STRATIFIED FLOW REGIME BY MONTECARLO SIMULATIONS AND POLYNOMIAL CHAOS EXPANSIONS.

Davide Picchi, *Università degli Studi di Brescia, Italy; Pietro Poesio, Università degli Studi di Brescia, Italy*

The prediction of uncertainties is a growing interest in flow assurance industrial applications, but only few works have been presented on this topic. In this work, uncertainty quantification and a sensitivity analysis are performed on the steady and fully developed two-fluid model for the stratified flow regime. The effect of several variables on pressure drop and holdup predictions is investigated, such as fluid flow rates, geometry (the inclination angle and the pipe diameter), and fluid properties (density and viscosity); the case of a non-Newtonian power-law fluid behavior is presented. Flow pattern transition boundary predictions from the stratified flow regime obtained by the linear interfacial stability and the well-posedness analyses are also considered in the uncertainty analysis. Two approaches are used for this purpose: Monte Carlo simulations and the polynomial chaos expansions. The results obtained by the simulations are discussed and a sensitivity analysis is carried out on models predictions to understand the contribution of each input parameter.

## FLOW REGIMES AND INSTABILITIES IN A T-MIXER OPERATING WITH A BINARY MIXTURE

Lorenzo Siconolfi, University of Pisa, Italy; Chiara Galletti, Università di Pisa, Italy; Simone Camarri, University of Pisa, Italy; Maria Vittoria Salvetti, University of Pisa, Italy

Efficient mixing in small devices is a key target in many processes. Among micro-devices, passive T-shaped micro-mixers are widely used. For this reason T-mixers have been studied in the literature and its working flow regimes have been identified, as well as the instabilities leading to the onset of these flow regimes. However, in most of the available theoretical studies it is assumed that only one working fluid is used, i.e. that the same fluid at the same thermodynamic conditions is entering the two inlet conduits of the mixer. Conversely, the practical use of micro-devices often involves the mixing of two different fluids or of the same fluid at different thermodynamic conditions. In this case the scenario of possible flow regimes can be substantially different from that observed for a single working fluid. To investigate this aspect, the present work aims at identifying the flow regimes in a T-mixer when water at two different temperatures is entering the mixer, the viscosity and density depending on temperature. The effect of the temperature difference on the flow regimes in a 3D T-mixer is investigated by DNS and stability analysis, and the results are compared to the case in which a single working fluid is employed.

\* Support from University of Pisa within PRA2015 is acknowledged.

## THE CHARACTERISTIC SCALES OF THE CAPILLARY INSTABILITY

Christophe Dumouchel, Normandie Université, France; Wojciek Aniszewski, Normandie Université, France; Thibaut Menard, Normandie Université, France; Trung-Thanh Vu, Normandie Université, France

Being free of any parietal constraints, a liquid jet in a gaseous environment deforms and breaks up into liquid fragments that, to their turn, may deform and breakup into smaller fragments. This process, called atomization process, continues until the liquid system has become a flow of stable spherical drops, i.e., a spray. During this process, the liquid-gas interface continuously varies with time. However, the mechanisms responsible for this depend on the size of the liquid fragment considered. A multi-scale approach is therefore required to investigate atomization processes.

A multi-scale description tool is described in the present work and applied to a liquid thread subject to a capillary instability. The behavior of the liquid thread is obtained from DNS using the code ARCHER. The parameter of the study is the wavenumber of the initial perturbation that triggers the instability. The simulations are performed in 3D allowing the multi-scale description and analysis to be performed in 2-D (as it would be done from experimental results) and in 3-D, which has never been done so far. Among other, the analysis will investigate the dynamic of characteristic scales revealed from the 2-D and the 3-D descriptions.

\* Thanks to the French National Research Agency (ANR) through the program Investissements d'Avenir (ANR-10 LABX-09-01), LABEX EMC3.

## EXPERIMENTAL INVESTIGATION OF INTERFACIAL WAVES INSTABILITIES IN STRATIFIED FLOW

Paula Stofer Cordeiro De Farias, Pontificia Universidade Católica do Rio de Janeiro, Brazil; Igor Braga De Paula, Pontificia Universidade Católica do Rio de Janeiro, Brazil; Anis Awal Ayati, University of Oslo, Norway; Luis Fernando Alzugar Alzevedo, Pontificia Universidade Católica do Rio de Janeiro, Brazil

Evolution of interfacial waves in stratified gas-liquid flow is investigated experimentally in a horizontal pipe. An oscillating plate controlled by a wave form generator artificially introduces the waves in the gas-liquid interface. The evolution of the excited waves is captured using two digital video cameras synchronized with a laser and the wave generator, enabling the application of ensemble averaging techniques. A transverse system allows measurements in several axial positions. A specially developed image processing algorithm is employed to automatically detect the position of the air-water interface in each image frame. Wave celerity, wavelength, skewness, amplification rates and spectral content of the fluctuations are obtained for different experimental parameters such as mixture velocities, wave amplitude and wave frequency. In order to obtain velocity profiles and measure interfacial shear stress, Particle Image Velocimetry technique (PIV) is performed both in water and gas phases. Results are analyzed in light of linear stability theory based on existing two-phase flow models. Some degree of agreement is observed for wave statistics such as wave celerity and wavelength.

## FARADAY INSTABILITY IN THREE FLUID MECHANICALLY OSCILLATED SYSTEMS

Kevin Ward, University of Florida, USA; Farzam Zoueshtiagh, University of Lille 1, France; Ranga Narayanan, University of Florida, USA

When stacked multi-fluid systems are periodically accelerated in a direction normal to the fluid interfaces, structured flow patterns on the interfaces will develop as a result of resonance between the imposed oscillation frequency and the natural frequency of the system. This resonance leads to deflection and growth of the interfaces once system parameters exceed a critical threshold, a phenomenon known as Faraday instability. In this work, we present theoretical calculations for the critical thresholds in three fluid mechanically forced systems, as well as the design and methodology for three fluid Faraday experiments. Experimental measurements of the thresholds and mode pictures will be compared against those predicted by linear stability analysis. Three fluid systems offer underlying physics distinct from traditional systems due to the coupling of the interfaces, and the effects of the third fluid on the Faraday thresholds may offer insight into the experimentally observed bifurcation dependence on forcing frequency found in traditional studies.

Support by NSF 0968313, CASIS NNH11CD70A, NSF EAPSI 1514711, NSF DGE-1315138, FSGC NNX15A110H, and a Chateaubriand Fellowship is gratefully acknowledged.

## EXPERIMENTAL STUDY ON FRAGMENTATION OF A MOLTEN JET INTO WATER

Miloud Hadj-Achour, University of Lorraine, France; Nicolas Rimbert, University of Lorraine, France; Sebastian Castrillon-Escobar, University of Lorraine, France; Michel Gradeck, University of Lorraine, France; Renaud Meignen, IRSN, France

In the event of a hypothetical core disruptive accident in nuclear power plants, the molten core may flow out the reactor vessel and interact with the cold water. The evolution of the accident is strongly affected by the fragmentation of the jet of molten metallic fuel due to its interaction with water (i.e. this situation is known as fuel coolant interaction, FCI). The present work focuses on the fragmentation of a molten jet into water. A device was designed and built for this purpose. Visual measurements were obtained with a high speed camera (phantom V70) and measurements of the frequencies associated with the molten-water interfacial instabilities are used to study the underlying physical mechanisms involved in the primary breakup of the molten jet. The role of the secondary liquid break-up and solidification of the molten jet is also investigated. The final size distributions of fragments of the solidified molten material obtained by a particle size analyzer (AS 200) will be compared with DNS simulation using the open-source software Gerris, it's expected that the turbulence and solidification play an important role on the final size of fragments.

\* This work was done under the research program on nuclear safety and radioprotection (RSNR) grant n° ANR-10-RSNR-01. The authors would like to also acknowledge support from EU COST MP 1305 "Flowing Matter"

## GRANULAR MIXING INSIDE ROTATING CYLINDERS: PRELIMINARY RESULTS

Gaetano De Monaco, Università di Napoli "Federico II", Italy; Davide Lombardi, Università di Napoli "Federico II", Italy; Francesco Greco, Istituto di Ricerche sulla Combustione, IRC-CNR, Italy; Pier Luca Maffettone, Università di Napoli Federico II, Italy

Blending two granular materials is a relevant problem in many industrial contexts. Nevertheless, granular mixing is still poorly understood and represents an exciting cutting edge topic in scientific research. Rotating cylinders, i.e. horizontal axially-rotating cylinders partially filled with granular materials, represent a paradigm problem for studying granular mixing.

The mixing behavior is influenced by difference in density or dimension of the grains, as well as by geometrical and flow factors, like the filling degree of the cylinder and the flow regime imposed to the materials.

In this work, we use a continuum mechanics approach for the description of the mixing of a binary mixture of grains, which undergoes a continuous flow near the free surface (rolling flow), by means of finite volumes numerical simulations. The materials are identical in their physical properties, but the two phases are marked in order to follow their blending. Some preliminary results will be discussed.

## INVESTIGATION ON THE PREDICTION OF DROP SIZE DISTRIBUTION IN SURFACTANT ADDED OIL-WATER EMULSION

Nannan Liu, China University of Petroleum, Beijing, China; Wei Wang, China University of Petroleum Beijing, China; Yunya Tian, China University of Petroleum, Beijing, China; Jiacheng Han, China University of Petroleum, Beijing, China; Meng Zhang, China University of Petroleum, Beijing, China; Jing Gong, China University of Petroleum Beijing, China

In this work, both experimental measurement and population balance equations about effects of surfactant concentration on drop size distributions in oil-water system were performed. Experiments were carried out by utilizing Focused Beam Reflectance Measurement (FBRM) with various concentrations of water soluble surfactant (Tween20) in stirred tank equipping four-bladed impeller with constant temperature, rotating speed and fixed total mixture volume. Employing a backward transform algorithm, chord length distribution (CLD) of dispersed droplets gained from FBRM was numerically converted to drop size distribution (DSD). Coupling several breakage and coalescence models proposed in literatures, population balance equations (PBE) with two region configuration was used to predict drop size distribution in surfactant dispersed system. Experiment data indicated that surfactant drastically influenced both drop size and counts, which was attributed to the change of interfacial properties. Comparison between PBE results against experimental data suggested that reasonable predication can be gained at dilute and dense surfactant concentration (after Critical Micelle Concentration, CMC), while for medium concentration large deviation still exist.

\* Support from the Beijing youth talent program funding (YETP0679) is gratefully acknowledged.

## PREDICTION OF INTERFACE LEVEL HEIGHT OF STRATIFIED LIQUID-LIQUID FLOW USING ARTIFICIAL NEURAL NETWORK

Mohamed Awad, Mansoura University, Egypt; Sadra Azizi, Islamic Azad University, Iran; Ebrahim Ahmadloo, Islamic Azad University, Iran; Roberto Ibarra, Imperial College London, UK; Ivan Zadrzil, Imperial College London, UK; Christos N. Markides, Imperial College London, UK

In this study, artificial neural network (ANN) was used to predict the interface level height (ILH) of two liquid phases flowing in a horizontal pipe. A three-layer feed-forward back-propagation (FFBP) neural network was constructed and trained with experimental data of two different liquid-liquid flow systems reported by Morgan et al. (2013) and Ibarra et al. (2015). The all studied flow patterns were stratified flow (stratified smooth and stratified wavy with or without droplets at interface). The input parameters of the ANN model were superficial velocity of phases, pipe diameter, the ratio of the lighter phase density to the heavier phase density ( $\rho_l/\rho_h$ ) and the ratio of the lighter phase viscosity to the heavier phase viscosity ( $\mu_l/\mu_h$ ), while the interface level height (ILH) of phases was its output. The Levenberg-Marquardt (LM) algorithm, the hyperbolic tangent sigmoid and the linear activation functions were used for training and developing the ANN. Optimal configuration of the ANN model was determined using minimizing the mean absolute percent error (MAPE) and mean square errors (MSE) between experimental and predicted ILH data by the ANN model. The results showed that the optimal configuration was a network with five neurons in hidden layer that was highly accurate in predicting the interface level. MAPE and correlation coefficient (R) between the experimental and predicted values were determined as 1.8% and 0.9962 for training, and 1.52% and 0.9996 for testing data sets.

## TOWARDS GENERAL PURPOSE LES MODEL OF ATOMIZATION

Nicolas Hecht, CNRS UMR 6614 - CORIA, France; Francois-Xavier Demoulin, Université de Rouen, France; Julien Reveillon, Université de Rouen, France

Various approaches are used to simulate two-phase flows. Usually, interface tracking methods are used to simulate the primary atomization while a dispersed method such as the Lagrangian particle-tracking approach may be used to model the final spray. Despite progresses in numerical methods and computer performance, complete simulation of atomization and spray remains inaccessible for many applications. A possibility is to extend the ELSA approach to the well-established LES framework combined with an interface density equation for subgrid scales. A postulated transport equation of the interface density is used to describe the subgrid spray formation ranging from interface wrinkling, ligaments, and sheets up to the droplets. In this work we incorporate the LES ELSA method developed by Chesnel et al. into a general purpose solver OpenFOAM. The method has been adapted for unstructured mesh to address applications. The interface capturing method is based on the transport of a liquid volume fraction,  $\alpha_l$ . To sustain the sharp gradient of  $\alpha_l$  an additional compressive term is added to the equation. LES formulation induced a subgrid term  $\text{SGS}\alpha_l$  that is incompatible with the numerical procedure used to capture the interface. The advantage of the compressive method is the possibility to switch it off when  $\text{SGS}\alpha_l$  become important, i.e. when the interface fluctuations become important at subgrid scale. To improve ELSA method a coupling with Lagrangian particle tracking will be initialized. Lagrangian particles are created when liquid structure become droplet. Particle diameter and number are determined via the liquid volume fraction and the interface density. The Eulerian equations are linked to the Lagrangian phase through the liquid turbulent diffusion flux closure. The aim of this paper is to study the LES atomization method and make comparison of the liquid dispersion around the axes with a

## SIMULATION OF THE DYNAMICS COMPLEX INFLUENCE ON HYDROCARBON MIXTURES

Aidar Assylbekuly, International Kazakh-Turkish University, H.A.Yassawe, Kazakhstan; Dauren Zhakebayev, al-Farabi Kazakh National University, Kazakhstan; Ualikhan Abdibekov, Kazakh National University named after al-Farabi, Kazakhstan

This paper deals with mechanical activation and acoustic impact on the hydrocarbon mixtures. Such action is organized by a special disc rotating at the same speed, but in opposite directions, inside the cylindrical area. Acoustic effects achieved by the rotation of the disks leads to a periodic emergence of opened regions, when the disks position is coincide, for the leaking the liquid; while the mechanical action determined by the size and speed of rotation of the disks. At the initial time a certain amount of liquid is injected, followed by a rotation of the disks inside the area, leading to vortex shedding. The numerical modelling of the mechanical activation and acoustic impact on the hydrocarbon mixture is solved based on the averaged Navier-Stokes equation, the continuity equation and equation for the concentration of components of a hydrocarbon mixture with chemical reactions in a cylindrical coordinate system. The simulation results showed that at a certain speed of the disk the vortex formation is observed, leading to change the concentration distribution inside the observed area. With increasing disks rotation speed also increases the number of the small scale eddies. Thus, implementation of the developed mathematical models for complex effects on the hydrocarbon mixture allows studying the effects of the process at the setting condition of various speeds rotation of disks, and also find the optimal speed, in order to be able to catch the vortex formation. It should be noted that the too slow, on an equal basis with fast rotation of disks, has no influence on hydrocarbon mix, described above.

## TOWARDS LARGE-EDDY SIMULATION OF LIQUID ATOMIZATION: A-PRIORI SUBGRID ANALYSIS

Sebastian Ketterl, Universität der Bundeswehr München, Germany; Markus Klein, Universität der Bundeswehr München, Germany

This work contributes towards Large-Eddy Simulation (LES) of primary atomization of liquids. Whereas LES of single-phase flows is already widely used in CFD, LES of two-phase interfacial flows remains a largely unexplored area since successful application of LES to two-phase flow problems should account for the unresolved complex interaction between turbulence and interface dynamics. The presence of the interface causes additional terms to appear in the LES formalism unknown from single-phase flow. Within this work, a-priori analysis is applied to direct numerical simulation (DNS) data of the liquid breakup process in order to evaluate the influence of two-phase LES subgrid contributions. A three dimensional DNS investigation of a spatially developing plane Diesel jet exhausting in gaseous atmosphere is conducted, with focus on the process leading to liquid atomization. Flow and interface statistics in the near nozzle region of the liquid Diesel jet which are very difficult to obtain experimentally will be presented. An explicit filtering of the DNS field allows an estimation of the order of magnitude of the LES subgrid contributions and conclusions on their relative importance are derived. In order to include the dynamics of small unresolved scales in LES computations, closure models for the most influencing two-phase subgrid contributions are proposed and evaluated by a-priori analysis.

## TWO-PHASE FLOWS IN SINUSOIDALLY CONSTRICTED CHANNELS USING MOVING MESH MOVING BOUNDARY TECHNIQUE

Gustavo Anjos, State University of Rio de Janeiro, Brazil; Gustavo Oliveira, Federal University of Paraíba, Brazil; Jose Pontes, State University of Rio de Janeiro, Brazil; Norberto Mangiacavacchi, State University of Rio de Janeiro, Brazil; John Thome, École Polytechnique Fédérale de Lausanne, Switzerland

Bubbles and drops dynamics through capillaries of variable cross-section still remains of considerable importance in two-phase flows through porous media. Recovery of oil by chemical flooding, biological processes and crude oil transportation in through pipelines are examples of industry-related applications. We seek to study numerically the effects of the surface tension, bubble dynamics and channel geometry for two-phase flows in sinusoidally constricted channels using a moving boundary domain scheme. This scheme dramatically shortens the domain length by moving the computational boundary nodes periodically according to the flow field or bubble centroid's velocity. The set of equations is written in a generalized form, the Arbitrary Lagrangian-Eulerian (ALE) description, which combines the best aspects of both Lagrangian and Eulerian framework. The two-phase interface position moves according to the flow field and it is explicitly described by a set of interconnected nodes, segments and elements which ensures a sharp representation of the front, not requiring any additional equation of motion. The new methodology proposed to simulate two-phase flows in sinusoidally constricted channels is compared to experimental results found in the literature showing good accuracy to describe interfacial forces and bubble dynamics in different complex geometries with moving boundaries.

\* Support from CNPq, CAPES and FAPERJ is gratefully acknowledged.

## LARGE EDDY SIMULATION OF GAS-PARTICLE TURBULENT OPPOSED JET FLOW

Dan Wu, Huazhong University of Science and Technology, China; Jing Li, Huazhong University of Science and Technology, China; Zhaohui Liu, Xi'an Jiaotong University, China; Chuguang Zheng, Huazhong University of Science and Technology, China

Large eddy simulation based on high order finite difference method has been implemented to investigate the gas-particle behavior in particle-laden turbulent opposed jet flow. A six-order finite difference scheme is applied for spatial discretization and corresponding six-order filter is used to filter the numerical error at high wave number. Subgrid stress is modeled by Smagorinsky model. Particles are traced using Lagrange method and four-way couplings are considered. The deterministic hard-sphere collision model is used to deal with the inter-particle collisions. The gas-particle opposed-jets flow are frequently encountered in chemical processes such as coal combustion and gasification, material grinding and drying. The difficulties in this type of flow is the complicated flow structures and massive inter-particle collisions in the impinging zone. In this paper, the coherent structures is captured by high-accuracy LES and the interaction between the instable stagnation plane and particles are carefully investigated. Particles inertial and volume fraction are found to have large influence on particle concentration profile. Moderate inertial particle penetrates repeatedly in the impinging zone; while the particle collisions under large particle volume fraction interrupt the penetration phenomenon and shorten the particle residence time, which leads the maximum particle concentration in the impinging zone.

\* Support from National Natural Science Foundation of China (Grant Nos. 51306065, 51276076, 51390494) are gratefully acknowledged

## STUDY ON SEAWEED REMOVAL EFFICIENCY FROM THE WASTEWATER BY SWIRL FLOW SEPARATOR

Hiroaki Tsubone, National Institute of Technology, Ariake College, Japan; Masaya Shimomura, National Institute of Technology, Ariake College, Japan; Syuichi Takesue, National Institute of Technology, Ariake College, Japan

In the seaweed manufacturing, much wastewater including fine seaweeds has been discharged at each process of storage, mincing, shaping, drying, washing and so on. The wastewater has been drained to the canals, river and sea because no draining standard for the seaweed wastewater has been decided. The color of wastewater change to red due to the pigment included in the seaweed cells. In addition, the seaweeds settled in the canals have become putrid and given off a bad smell. In this study, the purpose is to propose and spread a compact, cheap, maintenance free separator to remove seaweed from wastewater released from seaweed processing factories. At first, as an effective separation method of the seaweed from the wastewater, a swirl flow separator was proposed.

In this paper, the characteristics of removal efficiency of seaweed from wastewater, water percentage of removed seaweed and mass flow rate fraction of removed seaweed for total wastewater on mean flow velocity at the inlet pipe connected to the swirl flow generator are experimentally investigated. In addition, the effect of the parameters on swirl intensity is clarified. The swirl flow generator was a vertical circular pipe of 40 mm i.d. The separation section was a circular stainless steel mesh having same i.d. of the swirl flow generator and 120 mm in length. In the experiment, the range of mean velocity at the inlet pipe having 13 mm i.d. of the wastewater was 1.2 to 3.0 m/s. The detail experimental results and discussions are presented in this paper.

## UNSTEADY THREE-DIMENSIONAL NUMERICAL SIMULATIONS OF THE METHANE COMBUSTION IN DENSE FLUIDIZED BED

Ziad Hamidouche, Institut de Mécanique des Fluides de Toulouse (IMFT), France; Enrica Masi, INP Toulouse, France; Pascal Fede, INP Toulouse, France; Olivier Simonin, INP Toulouse, France; Renaud Ansart, LGC, France; Mehdi Hemati, Laboratoire de Génie Chimique de Toulouse (LGC), France

Natural gas combustion in dense fluidized beds containing inert particles involves complex physical mechanisms related to the bed hydrodynamic coupling with the gaseous combustion. In their experiments, Dounit et al. (Powder Technology 120, 49-54, 2001, Chemical Engineering Journal 140, 457-465, 2008) reported that for bed temperatures lower than a critical value ( $< 850^{\circ}\text{C}$ ), almost all the combustion takes place above the bed surface. In the present study, detailed unsteady 3D CFD simulations of such experiments have been performed using NEPTUNE\_CFD code (Konan et al., 7th International Conference on Multiphase Flow, Tampa, 2010), which is based on a two-fluid approach to compute both the gas and the solid phases. Time-averaged quantities are computed and compared with the available experimental measurements. The numerical results (gas temperature and gaseous-species molar fractions in and above the bed) are found to be very sensitive to the choice of the models for combustion kinetics, thermal radiation and heat exchanges between the phases and the reactor walls. In this study several models have been tested and their results discussed. Also the question of the mesh refinement was investigated. Numerical results showed that the mesh size may strongly affect the numerical predictions of such reactive dense gas-particle flows.

## LARGE EDDY SIMULATIONS OF PARTICULATE FLOW INSIDE A DIFFERENTIALLY HEATED CAVITY AT RAYLEIGH NUMBERS UP TO $10^{10}$

Abdel Dehbi, Paul Scherrer Institute, Switzerland; Jarmo Kalilainen, Paul Scherrer Institute, Switzerland

A series of well-resolved Large Eddy Simulations (LES) have been performed to investigate turbulent air flow and particle depletion rates inside a parallelepiped differentially heated cavity at Rayleigh numbers up to  $10^{10}$ . The mean flow field and turbulent statistics are first validated against recent experimental data conducted at Rayleigh number  $10^9$ . Subsequently, Lagrangian particle tracking coupled to the LES is used to predict the removal rates of dilute particles with diameters 1.4 microns and 3.5 microns. Both flow field and particle decay rates are in very good agreement with experimental data. The Euler/Lagrange simulations are extended to Rayleigh numbers up to  $10^{10}$ . While the particle concentration in the cavity decays exponentially with time, the decay rate constants are quite different from those predicted by the traditional "stirred settling" model for micron-sized particles. The latter are removed considerably faster than predicted by the "stirred settling" model. Only when particles are larger than 5 microns, does the "stirred settling" model yield results similar to the LES predictions. The LES investigations demonstrate that the stirred settling model is too simplistic to capture the transport of the smaller particles entrained in cavities where turbulent naturally driven flows prevail.

## SWIRLING FLOW CURVATURE SENSITIVE TURBULENCE MODELLING OF A GAS-SOLID CYCLONE

Yaser Alahmadi, The University of Sheffield, UK; Andrzej Nowakowski, University of Sheffield, UK

Owing to complex geometry, the turbulent flow in cyclones gives rise to a very broad range of spatial and turbulent scales which affect the swirling flow pattern. Reynolds stress model (RSM) and Large Eddy Simulation (LES) become preferred approaches for high swirling flows. However their obvious drawback is high computational cost and impracticality for complex design and performance study.

It is widely known that popular eddy viscosity models based on the Boussinesq assumption are insensitive to streamline curvature and system rotation due to the coordinate invariance. However because of their simplicity and stability they are still a preferred option in engineering simulations. The present work proposes a new sensitized eddy viscosity turbulence model to account for a strong streamline curvature. The numerical predictions are comprehensively validated using experimental results and are further compared with other available numerical techniques to simulate turbulent flow in cyclones. The numerical techniques used to compare the result with the proposed model are characterized by the application of different curvature models and are applied for cases in which rotation and curvature are known to have profound influence. Additionally, performance study are performed to assess the influence of the used method on the calculation of the particle separation efficiency in cyclones.

## NUMERICAL SIMULATION OF PROJECTILE LAUNCHED AT THE UNDERWATER USING GRANULAR PROPELLANTS

Kei Asano, Keio University, Japan; Akiko Matsuo, Keio University, Japan

The projectile launched at the underwater including combustion of granular propellants was numerically investigated. The 2D axisymmetric simulations were conducted by the newly developed code, which includes solid-gas reactive flow solver for the combustion chamber and liquid-gas cavitating flow solver for the underwater. For the propellants combustion, Vieille's formula was utilized to examine the burning velocity. The projectile accelerator has a 5.56 mm caliber and the 468 mm launch tube in length. The predicted maximum pressure in the chamber increase twofold from normal launched at atmosphere and has a good agreement with experiments. In addition, cavitation appeared around the muzzle the same as the experimental visualization. However, the projectile velocity has a 30% error, and the rate of pressure rise is overestimated. Simulations of w/ and w/o cavitation were compared to examine the cavitation effects for acceleration process, and then there is no difference of the pressure profile and the muzzle velocity. These indicated that cavitation is negligible to estimate projectile velocity. Therefore, it is supposed that the error of combustion process is a dominant reason of the numerical error. Since granular propellants behave differently from normal launching at atmosphere in underwater launched, constants of Vieille's formula should be modified to fit underwater launched.

## DIRECT NUMERICAL SIMULATION OF COMBUSTION NOISE OF A SPRAY FLAME

Abhishek Pillai, Kyoto University, Japan; Tomoaki Kitano, Kyoto University, Japan; Ryoichi Kurose, Kyoto University, Japan; Satoru Komori, Kyoto University, Japan

Combustion is a major source of core engine noise in gas turbines. In this study, a Direct Numerical Simulation (DNS) of a turbulent ethanol/air spray flame is performed using the Eulerian-Lagrangian approach. The Navier-Stokes equations solved using the Eulerian procedure describe the carrier gas phase, while the motion of the evaporating droplets is tracked by a Lagrangian approach. To account for droplet vaporization, a non-equilibrium Langmuir-Knudsen evaporation model is adopted. Combustion model for the evaporated ethanol is based on the Arrhenius formation for a two-step global reaction mechanism. Spatial-derivation of the nonlinear terms in the governing equations for the gas phase is approximated by a fourth-order low dispersive and low dissipative finite-difference scheme. Time integration is performed using a low-storage 4th-order six-stage Runge-Kutta algorithm and non-reflective boundary conditions are applied at the limits of the computational domain. The simulation results are compared with experimental data for the gas temperature as well as mean and fluctuating droplet velocities. Moreover, by analyzing the sound intensity levels, Overall Sound Pressure Level (OASPL), directivity and spectral content of noise in the near field of the spray flame, the combustion noise phenomenon in the spray flame is scrutinized and understood.

## MODULATION OF TURBULENCE BY DISPERSED SOLID PARTICLES IN A SPATIALLY DEVELOPING FLAT PLATE BOUNDARY LAYER

Dong Li, Zhejiang University, China; Kun Luo, Zhejiang University, China; Jianren Fan, Zhejiang University, China

Direct numerical simulations of particle-laden flows in a spatially developing flat plate boundary layer have been performed to investigate the effect of inertial particles on turbulence modulation, using the Eulerian-Lagrangian point-particle approach with two-way coupling. The present results show that the addition of particles increases the mean streamwise velocity, which, in turn, leads to the reduction in the boundary layer integral parameters and the increase in the skin-friction drag. The streamwise turbulence intensity is slightly enhanced in the close vicinity of the wall whereas damped in the outer layer. However, the Reynolds stress and the turbulence intensities in the wall-normal and spanwise directions are substantially attenuated, and the level of attenuation increases monotonically with both particle Stokes number and mass loading. The exchange of kinetic energy between particles and fluid indicates that the particle-fluid interactions cause an extra energy dissipation, which plays a crucial role in the turbulence modulation.

\* Support from grant 51136006 is gratefully acknowledged.

## DIRECT NUMERICAL SIMULATION MODEL INTERCOMPARISON FOR TURBULENT COLLISION STATISTICS OF INERTIAL PARTICLES

Bogdan Rosa, Institute of Meteorology and Water Management, Poland; Ryo Onishi, Earth Simulator Center, Japan; Lian-Ping Wang, University of Delaware, USA

In recent years, direct numerical simulations (DNS) have become an important tool for generating rigorous data for multiphase flows. DNS are particularly convenient for simulating complex microscopic atmospheric processes such as turbulent collision of cloud droplets relevant to cloud formation and precipitation development. This study focuses on small but finite-inertial particles carried in atmospheric turbulent flows. Turbulence-induced collisions of such inertial particles are relevant to many applications in engineering, geophysical and astrophysical flows. One serious problem is that simulation results are not always consistent and in some cases they may be different from one another because of the different numerical methods and algorithms used. In this study we perform, for the first time, a detailed comparison of the two independent DNS codes for turbulent collisions of inertial particles. The first code solves the Navier-Stokes equations using a finite difference approach and is based on a 3D domain decomposition. The second uses a pseudo-spectral method and is based on a 2D domain decomposition. The two codes have been developed independently in different groups but both have a deterministic forcing scheme to obtain statistically stationary turbulence. A number of simulations are performed using the same mesh size, viscosity, and forcing method. We then compare kinematic and dynamic collision statistics of inertial particles, relevant to collision growth of cloud droplets in turbulent clouds.

## EFFECTS OF PERMEABILITY-DEPENDENT POROUS ELECTRODE COMPRESSED RATIO ON THE PERFORMANCE OF VANADIUM REDOX FLOW BATTERY

Qiong Wang, Xi'an Jiaotong University, China; Zhiguo Qu, Xi'an Jiaotong University, China

The purpose of the present study is to investigate both experimentally and numerically the effect of electrolyte diffusion layer (EDL) porosity non-uniformity on the performance of vanadium redox flow battery (VRFB) due to clamping force. We experimentally demonstrate the electrical contact resistance and determine the permeability of the carbon felt under different electrode compression ratio. In the numerical simulations, a three-dimensional model of the same geometry as the test VRFB with interdigitated flow channel design is proposed to study the hydrodynamics and the distributions of current density and species concentration. Numerical results obtained by using porous electrode with uniformly distributed properties are compared with that by using electrode with non-uniform properties. It is found that although the overall cell performance is similar, local distributions from both models are significantly different. The distributions of electrolyte velocity, pressure drop, current density and species concentration are found to be highly oscillating in nature between the local rib and channel locations. Electrolyte flow rate and compression ratio are proved to be the critical factors affecting flow distribution and cell performance. Furthermore, the higher the compression ratio, the better is the cell performance and larger is the fluctuation amplitude.

## ON THE EMULATION OF BUBBLE-SWARM-INDUCED TURBULENCE WITH FLEXIBLE PARTICLE GRIDS

Katharina Haase, University of the Bundeswehr Muenchen, Germany; Ulf Kuck, University of Bremen, Germany; Jorg Thoming, University of Bremen, Germany; Christian Kaehler, University of the Bundeswehr Muenchen, Germany

The behavior of an individual bubble moving in a bubble swarm is of great interest to understand the effect of bubble-induced agitation on heat and mass transfer. However, detailed measurements of the bubble's motion, its shape and the generated wake are difficult to conduct due to the limited optical access in a bubble swarm. Therefore, a counter flow channel was built that allows for the emulation of typical bubble swarm turbulence as well as the investigation of a single bubble moving within this turbulent flow field. One method to imitate the hydrodynamic behavior of a bubble swarm is the usage of plastic particles organized in a flexible grid. With planar PIV (particle image velocimetry) the homogeneity, isotropy and turbulence level and with EDM (electrodiffusion measurement, University of Bremen) the energy spectra are measured and compared to turbulence levels in real two-phase, buoyancy driven flows. Two particle diameters and volume fractions of 5 and 10 mm respectively 3.87% and 9.8% are investigated. First PIV measurement conducted with the 10 mm particles and 9.8% volume fraction show a homogeneous turbulence distribution and the energy spectrum reveals the typical -3 slope known to be present in bubbly swarm turbulence.

\*This project is funded within the DFG SPP 1740.

## MODELLING TURBULENCE IN LARGE SCALE BUBBLE PLUMES

Jan Erik Olsen, SINTEF, Norway; Stein Tore Johansen, SINTEF, Norway; Paal Skjetne, , Norway

Large scale bubble plumes are found in large reactors (e.g. steel ladles) or in oceans and lakes. Turbulence is a governing mechanism for their behavior. Turbulent dispersion causes the bubble plume to expand horizontally as it rises upwards. Due to this the bubble plume is shaped like a cone. An Eulerian-Lagrangian modelling concept has been developed in which the bubbles are tracked as Lagrangian particles. Their dispersion due to turbulence is accounted for by a random walk model which is linked to the turbulence in the surrounding liquid. An accurate and reliable model needs to capture the turbulence properly. Traditionally the standard k-epsilon model has been applied to model the turbulence. It is numerically affordable and robust, but for some transient flows it does not capture the turbulence properly. LES models are more suited at modelling transient flows and turbulence. However, the computational cost might be discouraging. An alternative model is the VLES-model (very large eddy scale) which resolves the largest eddy scales at an affordable cost. The models are compared against each other and against experimental results.

# ROTATION OF A NEUTRALLY BUOYANT SPHEROID IN A SIMPLE SHEAR AT SMALL REYNOLDS NUMBER

Jonas Einarsson, University of Gothenburg, Sweden; Fabien Candelier, University of Aix Marseille, France; Fredrik Lundell, KTH, Sweden; Jean-Regis Angilella, University of Caen, France; Bernhard Mehlig, University of Gothenburg, Sweden

We derive an effective equation of motion for the angular dynamics of a neutrally buoyant spheroid suspended in a simple shear flow. The equation is valid for arbitrary particle aspect ratios and to linear order in the shear Reynolds number. It allows us to determine the stabilities of the log-rolling and tumbling orbits of the spheroid at infinitesimal shear Reynolds numbers. For prolate spheroids we find stable tumbling in the shear plane, log-rolling is unstable. For oblate spheroids, by contrast, log-rolling is stable and tumbling is unstable provided that the particle is not too disk-like (moderate asphericity). For very flat oblate spheroids both log-rolling and tumbling are stable, separated by an unstable limit cycle. For nearly spherical particles our results correct an earlier prediction by Saffman (1956). In the slender-body limit our results are consistent with earlier results of Subramanian & Koch (2005). The effective equation of motion gives rise to a number of transitions between different types of angular motion as the aspect ratio and the Reynolds number are varied. We discuss these transitions for finite but small Reynolds numbers.

J Einarsson, F Candelier, F Lundell, JR Angilella, B Mehlig, Physical Review E 91 (2015) 041002 and Phys. Fluids 27 (2015) 063301

F Candelier, J Einarsson, F Lundell, B Mehlig, JR Angilella, Physical Review E 91 (2015) 053023

# INVESTIGATION OF DRAG FORCE ON FIBRES OF BONDED SPHERICAL ELEMENTS USING A COUPLED CFD-DEM APPROACH

Anna Lyhne Jensen, Aalborg University, Denmark; Henrik Soerensen, Aalborg University, Denmark; Lasse Rosendahl, Aalborg University, Denmark; Per Adamsen, Grundfos, Denmark; Flemming Lykholt Ustrup, Grundfos, Denmark

Clogging in waste water pumps leads to expensive downtime of pumping systems and risk of pollution. Simulation of clogging effects is a complex task, which has potential to minimize the risk of clogging in new waste water pump designs by significantly facilitating the design process. Clogging is often caused by textiles and similar objects in the waste water. Therefore, simulation of clogging effects requires simulation of flexible objects and the interaction between these objects and fluid. This can be accomplished by a coupled CFD-DEM approach. Using this approach, the flexible objects are not resolved by the fluid mesh, and thus fluid forces must be modelled. In this study a fibre is modelled as a chain of bonded spherical elements. The free stream drag equation is initially used and the sum of the drag forces on the fibre is compared to the drag forces on a fully resolved rigid cylinder, with the same length and diameter as the fibre. Furthermore, the dependence of the shape of the flexible fibre on the total drag force is investigated and a new formulation of drag on a flexible fibre is tested.

\* Support from Grundfos is gratefully acknowledged.

# UNCERTAINTY QUANTIFICATION OF INJECTED DROPLET SIZE IN MONO-DISPERSE EULERIAN SIMULATIONS

Thea Lancien, CNRS, CentraleSupélec, Université Paris-Saclay, France; Nicolas Dumont, CNRS, CentraleSupélec, Université Paris-Saclay, France; Olivier Gicquel, CNRS, CentraleSupélec, Université Paris-Saclay, France; Ronan Vicquelin, CNRS, CentraleSupélec, Université Paris-Saclay, France

Large-eddy simulations (LES) of a laboratory-scale two-phase burner are considered by describing the dispersed liquid phase with a mono-disperse Eulerian approach. In such conditions, neglecting polydispersion of the injected spray makes the choice in the size of the injected droplet critical. The impact of such a key parameter upon the numerical results is carefully assessed through uncertainty quantification. Using Polynomial Chaos Expansion and Clenshaw-Curtis cubature rule, several LES are performed for different injected droplet size to obtain a response surface of the droplet size at any point in the computational domain as a function of the injected one. Post-treatment of the response surface gives access to the precise impact of the chosen injected droplet size on the results and to a first estimate of the actual polydispersed number density function (NDF). This information obtained from different mono-dispersed simulations enables to answer a couple of practical questions in such two-phase flow simulations: If only one simulation is to be carried out for a larger case, which value of the injected droplet size is the best? Given NDF experiment data at one point of interest but without any information at the injection point, how to set the injected NDF in a simulation that accounts for polydispersion?

\* Support from grant ANR-CE23-0009-01 from ANR (Agence Nationale de la Recherche) and 2015-2b0164 from GENCI (Grand Equipement National de Calcul Intensif) is gratefully acknowledged.

# DNS OF TURBULENT CHANNEL FLOW LADEN WITH A DENSE SUSPENSION OF NEUTRALLY BUOYANT FINITE-SIZE SPHEROIDS

Mehdi Niazi Ardekani, KTH Royal Institute of Technology, Sweden; Pedro Costa, Delft University of Technology, The Netherlands; Wim-Paul Breugem, Delft University of Technology, The Netherlands; Luca Brandt, KTH Royal Institute of Technology, Sweden

Suspensions of solid particles in a viscous liquid are of scientific and technological interest in a wide range of applications. In recent years only a few studies have been devoted to investigating the behavior of dense suspensions in the turbulent/inertial regime with the majority of these analyses are limited to monodisperse rigid neutrally-buoyant spheres. The present study aims therefore to fill this gap by investigating the behavior of non-spherical particles in the turbulent/inertial regime. For this purpose, direct numerical simulations (DNS) have been performed in a channel flow with bulk Reynolds number  $Re_b = 5600$ . In this study the mean particle diameter  $D_p^+$  is about 20 viscous wall units with the volume fraction varying between  $\Phi = 0$  to 0.1. A direct-forcing immersed boundary method (IBM) was used to account for the dispersed phase, combined with collision and lubrication models for particle-particle and particle-wall interactions. Preliminary results reveal a significant migration towards the channel centerline of the oblate spheroids; the additional stresses due to the particles are evident at volume fractions lower than for spherical particles. Detailed statistics of the fluid and particle phase will be presented at the conference.

# SUSPENSIONS OF INERTIAL SPHEROIDAL PARTICLES IN TURBULENT VERTICAL CHANNEL FLOW

Helge Andersson, Norwegian University of Science and Technology, Norway; Niranjana Reddy Challabotla, Norwegian University of Science and Technology, Norway; Lihao Zhao, Norwegian University of Science and Technology, Norway

The dynamic behavior of inertial spheroidal particles suspended in a turbulent vertical channel flow has been investigated by means of Eulerian-Lagrangian direct numerical simulations (DNSs). One-way coupled simulations at frictional Reynolds number 180 were performed for axisymmetric particles with aspect ratios ranging from 0.01 (flat discs) to 50 (long rods) and Stokes numbers ( $St$ ) from 1 to 100. For each particle class three different gravity configurations were considered: no-gravity, gravity acting in the flow direction (i.e. downflow) and gravity opposing the bulk flow (i.e. upflow). We examined the statistics of particle concentration, translational and rotational motions as well as the spheroids' orientation in view of the effects of gravity, particle shape, and particle inertia. The results show that the overall effect of gravity is negligible on the statistics of the least inertial spheroids ( $St=1$ ), whereas significant effects are seen for heavier particles. Comparisons with the no-gravity case showed that gravity plays an important role on the distribution of the non-spherical particles across the vertical channel. The gravity effects are sometimes, but not always, opposite in downflow and upflow. This behavior can be explained by consideration of the drift velocity of the spheroids in the three gravity configurations.

# TWO-SIZE MOMENT EULERIAN MULTI-FLUID METHOD DESCRIBING THE STATISTICAL TRAJECTORY CROSSING: MODELING AND NUMERICAL SCHEME

Matthieu Boileau, CNRS - CentraleSupélec, France; Valentin Dupif, ONERA, France; Frederique Laurent, EM2C, CentraleSupélec, France; Julien Lagarde, CNRS - CentraleSupélec, France; Marc Massot, CentraleSupélec, EM2C, France

High fidelity modeling and simulation at relatively low cost of turbulent dispersed disperse two-phase flows, where a discrete particulate condensed phase is transported by a carrier gas, is still a major challenge for many applications. Eulerian approaches are well suited for high performance computations of such flows. Recently, some hybrid Eulerian methods between that combine the multi-fluid method - one where the size is discretized - and the moment ones methods were developed. On the one side, in order to capture efficiently the size-polydispersion, two . It used two moments was used on each interval of the size discretization. In On the other side, the Anisotropic Gaussian velocity closure werehas been introduced as a relevant minimal model to describe velocity dispersion occurring when the particles from the disperse phase have a significant inertia compared to the time scales of the flow, and leading to when particle trajectory crossings then crossing occurs. The purpose of this contribution is to develop a model able to describe size and velocity dispersion, coupling the two-size moment Eulerian multi-fluid method and the anisotropic velocity closure. Adapted numerical schemes based on a relaxation method are also provided. This new model is then evaluated on various test cases relevant to solid propulsion and two-phase combustion.

This new model is then evaluated on relevant test cases.

## A COMPARATIVE STUDY OF THE SURFACE COMPRESSION AND INTERFACE RECONSTRUCTION ALGORITHMS FOR THE VOF APPROACH

Paolo Cifani, University of Twente, The Netherlands; Bernard Geurts, University of Twente, The Netherlands; Hans Kuerten, Eindhoven University of Technology, The Netherlands; Cees Van Der Geld, Eindhoven University of Technology, The Netherlands; Wiktor Michalek, Eindhoven University of Technology, The Netherlands; Giel Priems, Eindhoven University of Technology, The Netherlands

A comparative analysis between a compressive scheme and an interface reconstruction technique is carried out for the volume of fluid approach (VOF). Positive properties of the compression method are its simple implementation, applicability on unstructured meshes, mass conservation and boundedness, but this method results in artificial smearing out of sharp interfaces and a loss of accuracy. The interface reconstruction method, on the other hand, gives an accurate representation of the interface which is kept sharp by construction, at the expense of higher computational time. The platform we work with is OpenFOAM, where a new library was implemented that performs the advection of a color function by means of a Piecewise Linear Interface Calculation (PLIC) together with a splitting advection algorithm. The two methods are compared for several 2D and 3D advection test cases with prescribed velocity field and then applied to the simulation of a rising bubble, including surface tension and a mass density ratio between the two phases of 1000. The computational model is used to simulate a two-phase turbulent bubbly channel flow at  $Re = 180$ . Mean liquid and gas velocity, as well as kinetic energy of the liquid phase are computed for different gas volume fractions and compared with the corresponding single-phase channel flow.

## DYNAMICS OF DOUBLE EMULSION DROPLETS IN SHEAR AND CHANNEL FLOWS

Sangkyu (calvin) Kim, Purdue University, USA; Sadegh Dabiri, Purdue University, USA

Double emulsion drops or compound drops have been of great interest for potential applications in many different fields such as materials processing, pharmaceutical engineering, and microfluidics applications. The usage of double emulsion droplets as chemical delivery systems would be of significant importance, provided that the associated stability, temporal response, and release (breaking) mechanisms are well understood. Using finite volume method with front-tracking, we simulate the dynamics of a compound drop in a simple shear flow and in a channel flow and explore the stable configurations and limit cycles of a compound drop in these flows. It is found that for a compound drop in a simple shear flow, depending on the initial location of the inner drop relative to the outer drop, either a stable equilibrium is reached or a limit cycle is reached where inner drop cycles on the inner periphery of the outer drop in the plane of the shear. The compound drop in the channel flow shows the lateral migration for both outer drop and inner drop that is similar to but not exactly the same as what has been observed for solid particles migration.

\* This work is supported by the United States National Science Foundation Grant No. CBET-1335913.

## NUMERICAL INVESTIGATION OF COMPRESSIBILITY EFFECTS IN THE DROPLET LADEN SUPERSONIC SHEAR LAYER

Zhaoxin Ren, Tsinghua University, China; Bing Wang, Tsinghua University, China; Huiqiang Zhang, Tsinghua University, China

In this paper, a numerical simulation is performed to investigate the compressibility effects on the dispersion and evaporation of fuel droplets. A three-dimensional spatially-developing supersonic shear layer flow laden with evaporating liquid fuel (n-decane) droplets is conducted and the multiphase flow is described by a hybrid Eulerian-Lagrangian approach. The gas-phase is governed by the compressible Navier-Stokes equations together with species transport equations and the convective flux terms of which are obtained by a hybrid-WENO-Compact scheme. The droplets are treated as point mass and tracked individually in the Lagrangian framework. Two-way coupling interaction is considered through the exchange of mass, momentum, and energy between the carrier-gas fluid and the liquid-fuel spray. The results indicate that the large scale mixing and the growth rates are reduced with increasing compressibility of the shear layer. The PDF of the droplet size and instantaneous droplet distribution reveal a poly-disperse system in the turbulent flow. Compressibility effects on droplets dynamics, evaporation and droplet size distribution are detected and commented. In addition, shocklets in highly turbulent compressible shear layers affect the evaporation process and the unsteady fuel-oxidizer mixing.

## ISOADVECTOR: A VOF ALGORITHM FOR SOLVING THE SHARP INTERFACE ADVECTION PROBLEM ON ARBITRARY MESHES

Johan Roenby, DHI, Denmark; Henrik Bredmose, Technical University of Denmark, Denmark; Hrvoje Jasak, University of Zagreb, Croatia

A central task in numerical solution of two-phase flows is the advection of a sharp interface through a computational mesh. We have developed a new VOF algorithm that works on general polyhedral meshes: At the start of a time step we identify surface cells as those with volume fraction between  $\epsilon$  and  $1 - \epsilon$ , where typically  $\epsilon \sim 10^{-8}$ . For such cells we find the internal isosurface, or *isoface*, cutting it into two parts containing each fluid, say, water and air, respectively. Using different isovalues for different cells ensures that cells are cut into the correct volume fractions. We then model the motion of this isoface *within* the time step  $\Delta t$  by interpolating the velocity field to the isoface centre. This we use to get on all the *downwind surface cell faces* an estimated motion of the *face-interface intersection line*. Assuming constant total fluid flux on such a face during  $\Delta t$ , we analytically integrate the evolving "wetted" face area, to obtain an approximation of the water volume crossing the polygonal face during  $\Delta t$ . In the talk we will demonstrate the excellent performance of the method with benchmark cases on both 2D and 3D structured and unstructured meshes: Volume conservation is near exact; the interface is kept sharp to within the cell size; boundedness is also good and can be improved by an additional gentle redistribution step; shape preservation is very good, especially compared to standard algebraic VOF algorithms, to which the method is similar in terms of speed.

\* Support from The Danish Council for Independent Research (Grant-ID: DFF - 1337-00118) is gratefully acknowledged.

## DROP BREAKUP MODELS IN SPRAY SIMULATIONS

Timea Kekesi, Royal Institute of Technology (KTH), Sweden; Gustav Amberg, KTH, Sweden; Lisa Pahl Wittberg, Royal Institute of Technology (KTH), Sweden

The deformation and breakup of liquid drops have been studied numerically in uniform flows and flows with various shear rates. Fragmentation of fuel drops plays an essential role in combustion, influencing the number and size of droplets, and thereby the rate of mixing and the efficiency of the process. This in turn is a crucial factor from an energy- and pollution point of view, and has been a problem of longstanding interest. Weber number 12 has generally been used as the critical value for the onset of breakup of single drops, and breakup mode regimes were defined as a function of Weber number. These definitions are established for simple flows and do not take density and viscosity ratios into account. In this project existing atomization models applied for fuel sprays are extended by accounting for density and viscosity ratios in addition to the Reynolds and Weber numbers already present in current models. Furthermore, the effect of local velocity gradients of the flow on the breakup dynamics is investigated and included in the breakup models. Simulations with a coupled VOF-LPT method are performed employing the new breakup model.

\* Support from the Swedish Research Council (Vetenskapsrådet) through the Linné Flow Centre is gratefully acknowledged.

## LEVITATION OF WATER DROPLET INSIDE FERROFLUID: COMPUTATIONAL SIMULATION AND EXPERIMENTAL DEMONSTRATION

Chamkor Singh, Indian Institute of Technology Kharagpur, India; Arup Kumar Das, Indian Institute of Technology Roorkee, India, India; Prasanta Kumar Das, Indian Institute of Technology Kharagpur, India

Ferrofluid (FF) is a colloidal suspension of surfactant coated magnetic nanoparticles (MNPs, characteristic size  $\sim 10$  nm) inside a suitable carrier liquid. A spatially non-uniform magnetic field (H) can cause a pressure (p) redistribution inside the ferrofluid, pressure being higher at higher field regions. This fact leads to a net unbalanced force on a non-magnetic object placed inside the ferrofluid. As a result the object can be levitated, even defying gravity. This unique hydrodynamics has been simulated using a computational two-phase fluid dynamics based on coupled flow-maxwell equations. Particular attention has been given to study the effect of drop diameter and the field strength on the drop dynamics. Parallely, we investigate the levitation dynamics of a gravity stabilized water droplet (diameter  $\sim 1$  mm) immersed in a ferrofluid inside a Hele-Shaw cell (wall gap  $\sim 1$  mm) under a spatially non-uniform H. The phenomenon is captured by a digital camera with a high frame rate (5000 fps) for a set of repetitive experiments. Simulation results compares with the experiments reasonably well. Further elaboration of the study will be presented in the full paper.

\* Support from CSIR, India is gratefully acknowledged.



## DEVELOPMENT OF CLOSURE MODELS FOR UNRESOLVED PARTICLE SIMULATIONS OF HEAT AND MASS TRANSFER IN DENSE PARTICLE BEDS USING CPPPO: A LIBRARY FOR FAST PARALLEL DATA FILTERING

Federico Municchi, TU Graz, Austria; Stefan Radl, TU Graz, Austria; Christoph Goniva, DCS computing, Austria

Most multiphase flows are characterized by a wide range of length scales at which processes of interest take place. Hence, closure models need to be used to account for small-scale phenomena, e.g., heat transfer from the particle surface to the surrounding fluid. We present a new opensource library for data filtering and processing named CPPPO (Compilation of fluid-Particle Post-Processing routines) that aims on developing such closure models. This library has been designed as a universal tool for developing closure models in the context of single- and multiphase flows. CPPPO features a novel, highly efficient algorithm for on-the-fly parallel data filtering, and can be easily connected to any existing flow simulator with a minimum amount of coding. Furthermore, we will use CPPPO to derive a closure model for heat and mass transfer (from DNS data) to be used in unresolved CFD-DEM simulations. The new closure model accounts for the filter size and the local flow environment experienced by each particle in addition to the classically-used dimensionless groups.

\* Support from grant FP7 604656 of the European Commission is gratefully acknowledged.

## FLUIDIZING PULVERISED COAL WITH A VERY WIDE SIZE DISTRIBUTION

Barry James Azzopardi, University of Nottingham, UK; Xia Li, University of Nottingham, UK; Josep Escrig Escrig, University of Nottingham, UK; David Hann, University of Nottingham, UK

The majority of research into fluidization is carried out with mono-sized particles. Yet in reality, there is always a range of particle sizes present. Pulverised coal with a very wide range of particle sizes (0-800 $\mu$ m) has been fluidized using nitrogen in a 127 mm diameter bed. Both aerated and bubbly flows were observed. The solids fraction, and its variation in time and cross-section have been measured using Electrical Capacitance Tomography (ECT). The time series of cross-sectionally averaged solids fraction have been analysed to extract frequency information. Though individual bubble can be inferred from the cross-sectionally averaged time series, their clear identification requires knowledge of the cross-sectional distribution of solids fraction. An iterative reconstruction approach has been used to provide such information. The sizes and position of bubbles and their trends with gas velocity and bed depth have been quantified. The results are used to test published equations. In addition, bubble velocities have been extracted from the time series from two ECT planes at different axial position on the column. Their magnitude and trends with gas velocity are compared with published data.

## UNCERTAINTIES AND FLUCTUATIONS IN THE UPSCALING OF DENSE INERTIAL PARTICLE FLOWS

Matteo Icardi, University of Warwick, UK

Eulerian models for multiphase dispersed flows are commonly derived by means of ensemble (or spatial) averaging. They are therefore based on closures and correlations defined by averaging on different particle configurations. However, averaged momentum exchange terms (e.g., drag) are explicitly known only for the dilute (isolated spheres) and dense (Ergun) limits, where the equation can be derived analytically. Explicit closures are instead not generally known for arbitrary particle density and particle shapes and we have to rely on approximate correlation laws. This is typically one of the main sources of errors and variability in the averaged models. It is well known, in fact, that the overall results are often very sensitive to the correlation chosen and to the closure approximations for fluctuating quantities, strongly limiting the predictive capability of the models. In this work the forces acting on random arrays of spheres or more complex particles have been studied with a novel statistical approach based on multilevel Monte Carlo. Fully resolved DNS Simulations are used to resolve the flow around static spheres and both the numerical and statistical error are controlled accurately. Ensemble mean and fluctuations (with respect to the single particle or the volumetric ensemble) of the forces can be characterized to derive new statistically robust closures (e.g., drag, lift, dispersion correlation laws) or higher order models for dense poly-dispersed flows in the quasi-static approximation. We test our approach for laminar flows, showing the computational advantages of the method and its potential.

## STUDY ON THE COMPRESSION, FLOW AND PERMEATION OF FINE AND ULTRA-FINE, COHESIVE AND COMPRESSIBLE POWDERS

Abbas Kamranian Mamani, Chair of Mechanical Process Engineering, Germany; Andreas Bueck, Otto von Guericke University, Germany; Sergiy Antonyuk, Kaiserslautern University of Technology, Germany; Dominique Thevenin, Otto von Guericke University, Germany; Jurgen Tomas, Chair of Mechanical Process Engineering, Germany

Fine and cohesive powders are ubiquitous in manufacturing of many products. However, unlike liquids and gases, the understanding of the powder properties and behavior is poor. Knowledge about fluidization, compressibility and permeability of powders can give pivotal information about their properties and help to specify their flow behavior. The fluidization behavior of ultrafine cohesive powders (Geldart C) is identified by cracks and channels leading to serious non-uniformity in the bed. The fine powders (Geldart A) are recognized as more homogeneous fluidization behavior. In this paper the fluidization behavior and the affected material and process parameters (e.g. particle size and density, bed height, superficial velocity and the bed pressure drop), the compression and permeation and the affected parameters (e.g. applied pressure, bed height and bulk density) and the re-fluidization (after compression, permeation or both) and the history dependent behavior of fine ( $d < 100 \mu$ m) and ultrafine ( $d < 10 \mu$ m) powders are thoroughly discussed and compared. The model materials used in this study are two milled products with a rhombohedral crystalline structure, i.e. an ultra-fine CALCITE MX-10 and a fine CALCIT MVT-100, extracted from CALCIT FW-270 with some classification processes.

\* Support from grant DFG-GRK 1554 "Micro-Macro Interactions in Structured Media and Particulate Systems" is gratefully acknowledged.

## TOWARDS HIGH ORDER ACCURACY CAVITATION SIMULATION WITH THE DISCONTINUOUS GALERKIN SPECTRAL ELEMENT METHOD

Fabian Hempert, Robert Bosch GmbH, Germany; Sebastian Boblest, University of Stuttgart, Germany; Malte Hoffmann, University of Stuttgart, Germany; Philipp Offenhaeuser, University of Stuttgart, Germany; Uwe Iben, Robert Bosch GmbH, Germany; Claus-Dieter Munz, University of Stuttgart, Germany

Component failure due to cavitation erosion is a pressing concern for the hydraulic component development. To account for cavitation in early stages of the development process a simulation framework is necessary, which represents the interrelationship of the cavitation and the flow field. Here we propose a high-order accuracy discontinuous Galerkin spectral elements method in combination with a tabulated equation of state for V-oil 1404. A full energetic thermodynamic equilibrium model was applied for the two-phase flow. To gain more details of the changes of the thermodynamic properties during cavitation a single bubble collapse was analyzed in detail. Further, we examined a micro throttle flow at various operating conditions and achieved good agreement with experimental results. It should be noted, that the present investigation was capable of representing the flow in more detail than conventionally applied simulation frameworks. Most notably, we obtained a detailed representation of small scale phenomena and fluctuations within the vapor region. The results suggest, that with the present methodology it is possible to simulate the complex thermodynamic effects of cavitation in an efficient manner and to obtain further insights in the interrelationships between cavitation and the flow field.

\* Support from the Bundesministerium für Bildung und Forschung (BMBF) and the Gauss Centre for Supercomputing (GCS) is gratefully acknowledged.

## EVALUATION OF VOID FRACTION AROUND A TUBE IN TWO-PHASE FLOW ACROSS HORIZONTAL TUBE BUNDLE

Hideki Murakawa, Kobe University, Japan; Misaki Baba, Kobe University, Japan; Katsumi Sugimoto, Kobe University, Japan; Nobuyuki Takenaka, Kobe University, Japan

Measurements of void fraction distribution around the tubes were performed in two-phase flow across a horizontal tube bundle by using X-ray radiography. The test section was vertical duct with  $90 \times 90 \text{ mm}^2$  having the in-line tube bundle. The in-line tube bundle had three vertical columns and half tubes were on the side walls with eight horizontal rows. Diameter of the tubes was 15 mm, and the pitch to diameter ratio was 1.5. The experiments were carried out at superficial gas velocity, JG, of  $0.03 \sim 0.43 \text{ m/s}$ , superficial liquid velocity, JL, of  $0.2 \text{ m/s}$ , under atmospheric condition. Under bubbly flow condition, the void fraction in the vertical gap in the bundle was higher than that in the horizontal gap. Furthermore, the void fraction was lower in the downstream of the tube than in the upstream of the tube. Transition from bubbly flow to intermittent flow could be predicted by comparison of void fraction and/or bubbles' passing time between the maximum gap and the minimum gap.

## CFD-INFORMED UNIFIED CLOSURE RELATION FOR THE RISE VELOCITY OF TAYLOR BUBBLES IN PIPES WITH LIQUID FLOW

Enrique Lizarraga-Garcia, MIT, USA; Jacopo Buongiorno, Massachusetts Institute of Technology, USA; Eissa Al Safran, Kuwait University, Kuwait; Djamel Lakehal, ASCOMP AG, Switzerland

Two-phase slug flow commonly occurs in gas and oil production systems. Current predictive methods are based on the mechanistic modeling approach, which requires the use of closure relations to complement the conservation equations to predict integral flow parameters such as liquid holdup (or void fraction) and pressure gradient. Taylor bubble velocity in slug flow, which is one of these closure relations, has been found to significantly affect the calculation of these parameters. In this work, Computational Fluid Dynamics (CFD) with Level-Set as the Interface Tracking Method (ITM) are employed to simulate the motion of Taylor bubbles in slug flow, for which the CMFD code TransAT( is used. A large numerical database with stagnant and flowing liquid for various Reynolds numbers is generated from which a unified Taylor bubble velocity correlation in flowing liquids ( $10 < Re < 1E+03$ ) for a wide range of fluid properties and pipe diameters ( $1E-06 < Mo < 5E+03$ ,  $10 < Eo < 700$ ) is developed. Furthermore, it is important to predict under which conditions the lubricating thin film above the Taylor bubble in inclined pipes is present. An analytical model predicting the gravity-induced drainage of the film, and a criterion to avoid film breakup are presented; the model has been validated experimentally.

## STABILITY OF STRATIFIED TWO-PHASE FLOWS IN INCLINED CHANNELS

Ilya Barmak, Tel Aviv University, Israel; Alexander Gelfgat, Tel Aviv University, Israel; Amos Ullmann, Tel Aviv University, Israel; Neima Brauner, Tel Aviv University, Israel

Linear stability of stratified two-phase flows in horizontal and inclined channels to arbitrary wave number disturbances is studied.

The problem is reduced to Orr-Sommerfeld equations for the stream function disturbance, defined in each sublayer and coupled via boundary conditions that account also for possible interface deformation and capillary forces. Upon applying the Chebyshev collocation method, the equations and interface boundary conditions are reduced to the generalized eigenvalue problems. These are solved by standard means of numerical linear algebra for the entire spectrum of eigenvalues and associated eigenvectors. The most unstable perturbations described by the leading eigenvectors are reported and their patterns are discussed, which allows us to make some additional conclusions on the nature of instability. The validity of the numerical solution for horizontal channel flow has been confirmed through comparison with analytical solution obtained by Kushnir et al. (2014) and with the numerical results of Tilley et al. (1994). The results are summarized in the form of stability maps to show physical conditions at which a stratified-smooth pattern can exist. It is found that for gas-liquid and liquid-liquid systems the stratified flow with smooth interface can be stable only in confined zone of relatively low flow rates, which is in agreement with experiments, but is not predicted by long wave analysis. The existence of multiple stable solutions in concurrent and countercurrent inclined flows is carefully examined and discussed.

## INVESTIGATION OF PARALLEL CHANNEL INSTABILITIES WITH VAPOR BACKFLOW USING AN IMPROVED DYNAMIC MICROCHANNEL EVAPORATOR MODEL

Tom Saenen, ÉCOLE POLYTECHNIQUE FÉDÉRALE DE LAUSANNE, Switzerland; John Thome, École Polytechnique Fédérale de Lausanne, Switzerland

Microchannel evaporators are envisaged to efficiently cool next generation high performance electronic devices. However, since their inception, many microchannel evaporator designs have been plagued by parallel channel instabilities which deteriorate their performance and accelerate the onset of dryout and critical heat flux. Recently, microchannel evaporators are designed with inlet restrictions to stabilize the flow at the cost of additional pressure losses. The design of the inlet restrictions is still an open question in two-phase microchannel flow research. To tackle this problem, a new fully dynamic numerical model of a microchannel evaporator has recently been developed. With the help of this model the stabilizing effects of inlet restrictions can be investigated in further detail.

In this paper, the model is further expanded to better simulate the flow distribution in the in- and outlet manifolds or plenums of the microchannel evaporator. Furthermore, a more accurate model is used to incorporate the pressure losses of the inlet restrictions. These improvements allow the model to more precisely simulate unstable cases with vapor backflow into the inlet manifold which is often encountered during parallel channel instabilities.

Finally, new simulation results of parallel channel instabilities with vapor backflow and the stabilizing effects of the inlet restrictions will be presented for a wide variety of realistic operating conditions.

## NUMERICAL ANALYSIS OF FLOW CHARACTERISTICS INSIDE HYDRAULIC COUPLING AT VARIOUS OPERATIONAL CONDITIONS

Minhee Kwak, Sogang University, Republic of Korea; Nahmkeon Hur, Sogang University, Republic of Korea; Choon-Soo Chang, VS ENG, Republic of Korea; Nam-Won Kang, VS ENG, Republic of Korea

Hydraulic couplings are devices serve for torque transmission between separated shafts. Hence, in industrial applications the transmitted torque of hydraulic coupling is controlled by the amount of working fluid inside the system. The purpose of this paper is to investigate three-dimensional simulation of a water driver at different speed ratios and with various amount of working fluid. The two wheels are assumed to rotate at different speed ratios. In order to accurately simulate the physical phenomenon inside the wheels, unsteady turbulent flow is simulated using sliding mesh technique together with the Realizable k- $\epsilon$  turbulence model. In addition, the Volume of Fluid (VOF) method is employed to simulate two-phase mixture of air and water inside the water drive. The results are presented and discussed in terms of phase distribution, velocity components, flow patterns, circulating mass flow rate at interface between two wheels and transmitted torque, which are crucial factors for understanding the flow characteristics of hydraulic coupling in various operating conditions. The results shown that the circulation mass flow and torque rapidly decrease at special speed ratio from case to case, which is a result of formation (or increase) of the unwet surfaces over the wheels increase at the higher speed ratios.

## THREE-DIMENSIONAL STUDY OF DISTURBANCE WAVES FORMATION IN DOWNWARD FLOW OF GAS-SHEARED LIQUID FILM

Mikhail Cherdantsev, Institute of Thermophysics SB RAS, Russian Federation; Sergey Isaenkov, Institute of Thermophysics SB RAS, Russian Federation; Andrey Cherdantsev, Kutateladze Institute of Thermophysics, Russian Federation; Dmitriy Markovich, Kutateladze Institute of Thermophysics, SB RAS, Russian Federation; Sergey Alekseenko, Institute of Thermophysics SB RAS, Russian Federation

Disturbance waves strongly affect pressure drop as well as heat and mass transfer in annular flow. They carry the major fraction of liquid and are necessary for entrainment of liquid from film surface. To properly model the disturbance waves, experimental information on formation and spatiotemporal behaviour of disturbance waves is highly desired, since the mechanisms of these processes are still unclear due to high complexity of the problem. Recently, brightness-based LIF technique was applied to study formation and initial stages of development of disturbance waves in one longitudinal section of a vertical pipe. It was found that the disturbance waves are created due to coalescence of high-frequency Kelvin-Helmholtz (KH) waves. In the present work film thickness measurements were resolved along both longitudinal and transverse coordinates to study the spatiotemporal evolution of film thickness in a vertical rectangular duct. It was observed that the KH waves are initially 2D, but they are promptly fragmented into transversely localised ripples. Further downstream large groups of such ripples coalesce and form a wide 3D plateau of locally thicker film, which evolves into a disturbance wave.

\* Work supported by Russian Foundation for Basic Research (Proj. 15-58-10059\_KO-a).

## INSTABILITIES IN STRATIFIED TWO THERMOVISCIOUS LIQUIDS FLOW IN A PLANE CHANNEL

Victor Kireev, Mavlutov Institute of Mechanics, Russian Federation; Adelina Nizamova, Mavlutov Institute of Mechanics, Russian Federation; Said Urmancheev, Mavlutov Institute of Mechanics, Russian Federation

The present paper is concerned with the numerical analysis on the stability of plane-parallel flow of stratified two immiscible viscous liquids with temperature-dependent viscosities in non-uniform temperature field. The aims of this work is to understand whether the variation of viscosity with temperature influences on the onset and the evolution of Kelvin-Helmholtz instability on the interface between two thermoviscous liquids as well as to explain the potential mechanisms for such phenomenon.

The system of ordinary differential equations for perturbation amplitudes of stream functions and temperature has been developed. In the simple case of one-layer plane Poiseuille isothermal flow, this system of ODE can be reduced to the classical Orr-Sommerfeld equation. The spectra of eigenvalues for stratified flows with different temperature dependences of viscosity have been studied numerically. The considerable differences between the spectra of eigenvalues for the flow of thermoviscous liquid and liquid with constant viscosity are discovered. It is shown that the temperature dependence on viscosity considerably affects on the stability of stratified two-fluid flows.

The results of the work will contribute to the theoretical explanation of the recent experimental data that show there exist concurrent laminar-turbulent regimes for stratified two-fluid flows.

\* Support from the Russian Foundation for Basic Researches (grant #14-01-97034) is gratefully acknowledged.

#### CFD SIMULATION OF GROWING NANOPARTICLES IN A FSP FLAME USING DIRECT QUADRATURE METHOD OF MOMENTS

Pedro Bianchi, University of Blumenau, Brazil; Dirceu Noriler, University of Blumenau (FURB), Brazil; Henry França Meier, Regional University of Blumenau (FURB), Brazil; Udo Fritsching, Foundation Institute of Material Science, Univ. Bremen, Germany; Florian Meierhofer, University of Bremen, Germany

Although relatively new, the Flame Spray Pyrolysis (FSP) process is as well known technic for nanoparticle production. One of its advantages is that most of the elements on the periodic table can be converted into nanoparticles through their oxides and that may often be found as low cost precursors. This process may be divided in well-defined steps made possible due to the enormous amount of thermal energy released by the combustion of the fuel-precursor mixture in the reaction zone. The main objective of the present work is to apply CFD to investigate the nanoparticle development and growth inside the FSP reactor through the use of bivariate population balances solved with the direct quadrature method of moments (DQMOM). In the mathematical model, the mass, energy and momentum balances for the gaseous phase are represented by an Eulerian approach, which is coupled to a Lagrangian tracking of the evaporating fuel-precursor droplets. Satisfactory results were reached with the proposed models, given the complexity of the multiphase and multiscale problem. Results show that, even though nucleation starts at the region closest to the injection nozzle (~8 mm HAB), the actual growth of the nanoparticle only begins further up the flame where most of the droplets have already been evaporated and the thermal energy is available for the other growth related steps (~40 mm HAB). Particle diameter showed satisfactory agreement with experimental data with similar deviation for different concentrations of precursor.

#### EXPERIMENT AND COMPUTATIONAL SIMULATIONS OF LIQUID-LIQUID INTERFACIAL INSTABILITIES UPON FLOW DISPLACEMENT IN MICROCHANNELS

Yu Lu, University of Birmingham, UK; Mark Simmons, University of Birmingham, UK

Microfluidics have great potential in various applications and offer the opportunities of studying transport phenomena at very small length scales. The cleaning of a micro-scale channel is an essential aspect of any microfluidic device but has received little attention. The mechanism of cleaning studied is the displacement of one fluid by another, with particular focus on the interfacial instabilities of liquid-liquid two phase flow. These have been studied both experimentally and computationally. The experiment consists of the injection of one fluid into the microchannel that is pre-filled with another fluid. Micro-Particle Image Velocimetry ( $\mu$ -PIV) and shadowgraphy have been used to examine flow displacement in semi-circular or circular microchannels with diameters of 205  $\mu$ m and 200  $\mu$ m respectively. Miscible and immiscible fluid pairs with various viscosity ratios are studied. Different interfacial instabilities are observed and characterised in flow regime maps. Displacement efficiency, residual liquid film thickness, velocity fields and the effect of wall conditions such as wall wettability are also examined. The finite volume Ansys Fluent CFD package is used to simulate the flow phenomena observed and compared with velocity fields obtained from the  $\mu$ -PIV experiments.

\* Support from EPSRC MEMPHIS project grant is gratefully acknowledged.

#### MOFFAT VORTICES IN THE VICINITY OF A CONTACT LINE

Mijail Febres Soría, Institut de Mécanique des Fluides de Toulouse (IMFT), France; Dominique Legendre, Institut de Mécanique des Fluides de Toulouse, France

We consider a series of numerical simulations for the motion of a contact line formed by two viscous fluids and a solid surface in order to observe a sequence of eddies (Moffat Vortices) in its vicinity as proposed by Kirkinis and Davis (2014). This sequence of eddies is due to motion of the contact line and co-moves with it, in opposition to the stationary case (Moffatt 1964) where the sequence is due to an arbitrary distant motion of the fluid. Different contact angles, viscosity ratios and slip lengths are tested using the VOF model and the Navier slip boundary condition on the solid surface inside the in-house code JADIM. Moffat vortices existence is confirmed in the reference frame that moves with the contact line, however, vortices size and shape are observed to be affected by the slip length used.

\* Support from FINCYT under contract 099-FINCYT-BDE-2014 is gratefully acknowledged by M. F. The simulations were performed at CALMIP supercalculator.

#### STUDY OF NATURAL CONVECTION IN AN ENCLOSURE WITH WATER AND $\alpha$ - $\text{Al}_2\text{O}_3$ /WATER NANOFLUID FOR DIFFERENT ASPECT RATIOS USING STATISTICAL ANALYSIS

Rajesh Chowdhury, Indian Institute of Technology Roorkee, India; Sudhakar Subudhi, Indian Institute of Technology Roorkee, India

Parameters such as geometry and the tilt angle of the enclosure, heating surface and aspect ratio affect the natural convection. In the present study, the effect of aspect ratio in the range of 0.5 to 2.5 on the natural convection in the water and  $\text{Al}_2\text{O}_3$ /water nanofluid is investigated using statistical analysis in terms of histogram, skewness and flatness. A relation between the Nusselt number and the Rayleigh number is examined in a vertical square enclosure. The different combinations of aspect ratio and constant heat flux condition in the range of 5 – 25 kW/m<sup>2</sup> are applied. The outcomes demonstrated the dependence of the Rayleigh number on the aspect ratio for different heat flux conditions. The variation in the temperature of the fluid layer along with the height of the enclosure is observed. For the aspect ratio of 0.5, a little enhancement was observed in the heat transfer in nanofluid at lower heat flux than that of the distilled water. But for the aspect ratios of 1.0 and 2.5, the deterioration is observed at lower Rayleigh number and an enhancement in natural convection is observed at the higher Rayleigh number.

#### CFD SIMULATION OF WATER REMOVAL FROM GAS FLOW CHANNELS OF PEMFC USING WETTABILITY GRADIENT

Shadman Khan, Indian Institute of Technology Roorkee, India; Sumana Ghosh, Indian Institute of Technology Roorkee, India

One of the most potent alternative with extensive range of application is fuel cell. Among different types of fuel cells, Proton Exchange Membrane Fuel Cell (PEMFC) is most attractive because of its normal temperature operating condition. However a well hydrated membrane and saturated gas leads to the formation of water droplets inside the channel. This phenomenon results in hindrance in mass transport by blocking the gas diffusion layer (GDL). Few studies report the influence of static contact angle of channel surface on water removal. However, none of these works deal with a gradient in contact angle to remove water droplets. In the present work, a computational fluid dynamic simulation has been performed to investigate the behaviour of water in a rectangular straight gas flow channel of 20 mm length and 1mm<sup>2</sup> cross sectional area. Finite volume based numerical solver Ansys15 has been used for discretization of the governing equation.

#### WET GAS FLOW REGIMES: EXPERIMENTS AND TRANSITION MODEL BASED ON DIMENSIONLESS NUMBERS

Dennis Van Patten, DNV GL, The Netherlands; Rob Hagmeijer, University Twente, The Netherlands

Oil and gas operators rely on accurate data on multiphase flow regimes in flow measurement and transportation. A lot of data on flow regime maps is readily available in the literature; however, this data has been validated mainly against experiments on small scale and low pressure test facilities. Scaling rules are used to transfer this data to actual field conditions. The extensive set of independent variables in multiphase flow makes it difficult to verify this transfer or to exactly mimic the field conditions in a laboratory. Recently, a large testing campaign has been executed at the high pressure and large scale test facility at DNV GL in Groningen. The tests were operated in the so-called wet gas flow regime (gas volume fraction >95%) in horizontal orientation and utilized ultrasonic measurement devices to generate experimental data on liquid hold-up and the gas flow profile. Moreover, optical pipe spools capable of handling the high pressures were used to visualize the flow regimes and determination of the hold-up. The typical flow regimes encountered are (wavy) stratified, dispersed/annular and fully dispersed flow. A flow regime transition model based on dimensionless parameters has been derived and was verified with the high pressure experiments. The developed transition model can be used to validate existing flow regime maps and their extension to more realistic field conditions. The experiments demonstrated that for relative low liquid volume fractions (~5%) the liquid hold-up can reach values of 30%. Also the influence of the liquid on the flow profile of the gas was investigated.

## VAPORIZATION AND EXPANSION OF MOLTEN FUEL IN A SODIUM FAST REACTOR: A COMBINED NON-EQUILIBRIUM THERMODYNAMICS AND DIMENSIONAL ANALYSIS METHODOLOGY

Xavier Manchon, CEA Cadarache, France; Frederic Bertrand, CEA Cadarache, France; Nathalie Marie, CEA Cadarache, France; Michel Lance, LMFA, France; Damien Schmitt, EDF R&D, France

In order to improve safety and design of Sodium-cooled Fast nuclear Reactors (SFR), the Commissariat à l'Énergie Atomique (CEA) is working on hypothetical severe accident scenarios. These scenarios include a nuclear power excursion that might lead to the melting down of nuclear fuel, followed by its quick vaporization in the liquid sodium coolant. The mechanical energy released by the fuel vapor expansion will lead to the reactor vessel mechanical loading. This paper presents a methodology relying on Non-Equilibrium Thermodynamics (NET) and Dimensional Analysis (DA) which allows identifying the driving physical phenomena that should be taken into account in the modelling of the vapor expansion. The entropy production of this multiphase system is derived from discontinuous NET equations. From this entropy production, driving phenomena that should be investigated are identified. Dimensional Analysis is used to keep only the main governing phenomena. This method is used to assess the validity of the physical models of "COMBUS", an analytical CEA software.

\*This work is supported by the R4G program at CEA and by the R&D division of Électricité de France (EDF), which are gratefully acknowledged.

## EXPERIMENTAL STUDY OF PARTICLES BEHAVIOR IN TORNADO-LIKE GAS-SOLID FLOW

Mikhail Protasov, Joint Institute for High Temperatures, Russian Federation; Nikolay Vasiliev, Joint Institute for High Temperatures, Russian Federation; Aleksei Varaksin, Joint Institute for High Temperatures, Russian Federation

Tornadoes occur frequently in North America and other parts of the world. They destroy everything on its path and lead to the human fatalities. In our opinion, further progress in studying the problems associated with generation, stability and breaking of vortex structures with a view to explaining the tornado phenomenon will be based on the concept of its two-phase nature (the presence of droplets, solid particles and different subjects). The aim of this paper is to study the solid particles behavior in nonstationary air vortices. The simple setup used in this study allows to make the controlled heating of the aluminum plate top surface and to generate wall-free nonstationary concentrated vortex structures (called here as tornado-like vortices) over the plate due to the unstable air stratification. For the disperse phase behavior investigation the magnesium low-inertial (6  $\mu\text{m}$ ) and high-inertial (50  $\mu\text{m}$ ) particles were used. The process of the particles involving in tornado-like vortices and the specialties of their motion were studied by use of video camera and PIV technique. The role of centrifugal effect was found to be critical for particle flow visualization and accuracy of carrier gas velocity measurements. The dimensionless criterion (Stokes number) was applied, which characterized the inertia and centrifugal effect of solid particles during their motion in tornado-like vortices of different intensity.

\* This work was supported by the Russian Science Foundation (agreement no. 14-19-00453).

## SIMULATION OF A LIQUID-SOLID FLUIDIZED BED CRYSTALLIZER USING CFD-DEM

Kristin Kerst, University of Magdeburg "Otto von Guericke", Germany; Luis Medeiros De Souza, University of Magdeburg "Otto von Guericke", Germany; Andreas Seidel-Morgenstern, Max Planck Institute for Dynamics of Complex Technical Systems, Germany; Gábor Janiga, University of Magdeburg "Otto von Guericke", Germany

In this study, an innovative reactor concept developed for the continuous separation of enantiomers is investigated numerically. A racemic mixture is separated by selective crystallization in the reactor. For this purpose, the crystals are dissolved in a solution that flows through two parallel fluidized bed reactors. Small starter crystals are added continuously, and grown crystals (with the desired final diameter) are also discharged continuously from the fluidized bed. Locating precisely the injection point of starter crystals in the crystallizer is of central importance to reach the targeted Crystal Size Distribution (CSD). For this purpose, the hydrodynamics in the crystallizer must first be closely examined. Corresponding simulations deliver valuable information that are difficult or even impossible to measure experimentally with sufficient accuracy. For this investigation of a crystal-laden flow, coupled CFD-DEM simulations (DEM: Discrete Element Method) that take into account all physical interactions occurring between particles and fluid have been implemented. Due to the very high number of particles, these simulations require a very high computational effort, yet still manageable using large parallel computers. Simulations were carried out using the coupled open-source software CFDEMcoupling, connecting OpenFOAM with LIGGGHTS. After validating the simulation results with own experimental measurements, CFD-DEM results are used for process understanding and improvement.

## CONDENSATION IN SUPERSONIC STEAM EJECTORS: COMPARISON OF THEORETICAL AND NUMERICAL MODELS

Federico Mazzelli, University of Florence, Italy; Dmitry Brezgin, Ural Federal University, Russian Federation; Ilya Murmanskii, Ural Federal University, Russian Federation; Zhelonkin Nikolay, Ural Federal University, Russian Federation; Adriano Milazzo, University of Florence, Italy

Supersonic ejectors are commonly used for many applications in the field of refrigeration. In many cases, condensation or evaporation occur, either within the motive or the suction flow. A very important case of condensing ejector is the steam ejector, that can be used for suction of non-condensable gases in steam plants or in heat-powered chillers.

Design techniques for ejectors are still a matter of discussion. Ideal gas model are usually employed to easily obtain a coordinated set of dimensions. However, in the case of steam ejectors, the ideal gas behavior is far from being physically consistent. In order to produce a more refined design, real fluid behavior can be included in a thermodynamic, 1D model of the ejector by postulating equilibrium conditions. Other design techniques are based on interpolations of experimental data and, although empirical, can provide suitable sizing of the main ejector dimensions. Finally, a still higher level of modeling is multiphase CFD. Wet steam models are available in some commercial CFD codes and they claim to represent the non-equilibrium behavior of the flow. However, some doubts may be raised about the reliability of this approach in highly supersonic flows. All these levels of analysis are overviewed and compared in this paper.

## PARTICLE-LADEN FLOW IN MODELS OF ASTHMATIC AIRWAYS

Pantelis Koullapis, University of Cyprus, Cyprus; Stavros Kassinos, University of Cyprus, Cyprus

In Computational Fluid Dynamics (CFD) studies, the airways of the human upper respiratory tract are usually presented as smooth pipes. In reality, they may have complex cross sections, especially in the case of respiratory diseases such as asthma. Airflow obstruction, which is the main characteristic of asthma and causes difficulties in breathing function, results from the narrowing of the airways (referred as bronchoconstriction). Previous studies have revealed that particle deposition in asthmatic airways is considerably higher than in healthy airways, but none of these studies account for the morphological differences between normal and asthmatic airways. Specifically, during bronchoconstriction the airways may develop mucosal folding, in which the airways develop a number of folds instead of uniform constriction. The influence of airway narrowing and mucosal fold formation on the airflow and on particle deposition in such geometries under turbulent flow conditions has not been addressed in the past. The objective of this study is to give insight for this problem in simplified models of such geometries and identify differences as a function of several parameters, i.e. constriction ratio, Reynolds and Stokes number, using Large Eddy Simulations.

\* The present study was funded by the European Union 7th framework program HEXACOMM FP7/2007-2013 under grant agreement No 315760.

## THERMODYNAMICALLY CONSISTENT TWO-WAY COUPLED MODELLING OF GAS-PARTICLE FLOWS WITH PARTICLE TRAJECTORY CROSSING USING EULERIAN MOMENT METHODS

Marc Massot, CentraleSupélec, EM2C, France; Valentin Dupif, ONERA, France; Joël Dupays, ONERA, France; Frederique Laurent, EM2C, CentraleSupélec, France

Polydisperse gas-particle and gas droplet flows are encountered in a large number of two-phase flow applications, such as solid propulsion, two-phase combustion in aeronautical engines and automotive engines. In such applications, the mass loading of particles or droplets are usually high enough such that two-way coupling between the phases is occurring. Such phenomena can usually be described by a fluid-kinetic model, where the disperse phase is described at a mesoscopic level by a kinetic type of equation. Starting from there Eulerian moment methods can be used to design a mathematically well-posed system of equations describing low inertia particles. In such cases, consistent thermodynamics can be identified. For larger inertia particles experiencing particle trajectory crossing (PTC) high order moment methods can be used to describe properly PTC. However, in the framework of existing Eulerian models, only one-way coupling has been studied and the extension to a thermodynamically consistent two-way coupling requires a special care. In this work, we introduce a new framework to cope with such a new extension and make the link with another class of two-fluid models used for separate phase modeling, where thermodynamics consistency has been included since for example the early work of Baer and Nunziato. Relevant and accurate numerical schemes are also identified.

\* Support from ONERA for the PhD grant of V. Dupif as well as from EM2C Laboratory are gratefully acknowledged.

## TRANSITION BETWEEN ROBUST EQUILIBRIUM DESIGN POINTS FOR STEADY STATE PROCESSES OF PULVERIZED COAL COMBUSTION IN DOWNERS

Tal Eluk, Ben Gurion University of the Negev, Israel; Avi Levy, Ben Gurion University of the Negev, Israel; Efim Korytnyi, Bergmann Campus, Ben-Gurion University of the Negev, Israel; Tali Bar Kohany, Ben Gurion University of the Negev, Israel

Pulverized coal combustion inside a downer type test furnace was simulated using computational fluid dynamics (CFD). The two phase hydrodynamics was modeled using Euler-Lagrange approach applied by CFD-DPM coupling of heat, mass and momentum. Based on good agreement between the model predictions and the experimental results (Temperature and flue gases) the model was used for studying the various parameters affecting the combustion in downer reactors and process optimization. Specific design was found to achieve strong stability of combustion structure. The peak temperature location along the reactor was observed after altering some operating parameters such as inlet temperature and wall heat fluxes. While those parameters affected the reactor's behavior at a specific range, when the transition to the robust combustion structure mentioned above was made, it remained stable to a wide range of operating conditions. Similar effects were detected when dealing with changes in material properties that can be described as devolatilization rates and heterogeneous reaction rates. When design consideration of a steady state processes is considered, the transition between robust equilibrium design points is of special interest. The cause to this phenomenon is suggested to be the flow patterns near the reactor entrance region, dominated by inlet velocity and swirl, the mass and momentum transfer to the continuous phase and the heat generated by the exothermic reactions.

## EXPERIMENTAL UNCERTAINTY RECONCILIATION IN DROP TUBE CHARACTERIZATION OF REACTING STREAMS OF PARTICLES

Chiara Galletti, Università di Pisa, Italy; Gianluca Caposciutti, Università di Pisa, Italy; Salvatore Iavarone, Université Libre de Bruxelles, Belgium; Alessandro Parente, Université Libre de Bruxelles, Belgium; Leonardo Tognotti, Università di Pisa, Italy

One promising technology for carbon capture and sequestration (CCS) is oxy-fuel combustion, in which solid fuel (coal/biomass) combustion takes place in a mixture of oxygen and recycled flue gases, instead of air. The development of this technology requires further advancements, mainly aimed at providing high-fidelity simulation tools, as those based on Computational Fluid Dynamics, to be used for the design of practical burners and furnaces in power generation plants. So far reliable solid fuel combustion models are available only for conventional combustion, mainly because the lack of knowledge on the combustion process (e.g. devolatilization, char oxidation) in an atmosphere that is different from air. Indeed, even if some experimental data are available on oxy-fuel conditions, most of them have been generally taken in lab-scale reactors, and are hardly applicable to real furnaces.

The present work wants to derive heterogeneous kinetics in oxy-fuel combustion from experiments performed in a pilot-scale drop tube reactor, which ensures heating rates and temperatures similar to the industrial ones. A new procedure, based on both experimental data analysis and reactor modeling, is proposed to derive heterogeneous kinetics and to estimate the different sources of uncertainties. A proper treatment of the cloud of reacting solid particles was found to be fundamental for the determination of kinetics.

## MEASUREMENTS OF VOID FRACTION, PRESSURE DROP AND HEAT TRANSFER IN HORIZONTAL AND DOWNWARD INCLINED GAS-LIQUID STRATIFIED FLOW

Swanand Bhagwat, Oklahoma State University, USA; Afshin Ghajar, Oklahoma State University, USA

Two phase flow literature reports several studies dedicated to the characterization of stratified flow in horizontal pipelines while very little is known about the behavior of this flow pattern and its effect on two phase flow variables in downward pipe inclinations. This study aims at measurements of void fraction, frictional pressure drop and heat transfer in air-water stratified flow using a 12.5 mm pipe oriented in eight different downward inclinations ranging from 0 to 90 degrees. Depending on the pipe orientation, the stratified flow pattern is generated by varying gas and liquid flow rates that corresponds to superficial liquid (2000 to 10000) and superficial gas (180 to 8000) Reynolds numbers. A change in pipe orientation is found to significantly affect the extent of stratified flow on a flow regime map as well the magnitude of measured two phase flow variables. In general, void fraction, pressure drop and heat transfer coefficient is strongly affected by the change in liquid flow rate and remains virtually unchanged with increase in gas flow rate at a constant liquid flow rate. Additionally the data available in literature is used to study the effect of pipe diameter on measured two phase flow variables in stratified flow.

## PHYSICAL AND COMBUSTION CHARACTERIZATION OF BIOMASS PARTICLES PULVERIZED BY DIFFERENT MILLING PROCESSES FOR SUSPENSION FIRING IN UTILITY BOILERS

Chungen Yin, Aalborg University, Denmark; Maryam Momeni, Aalborg University, Denmark; Soren Knudsen Kaer, Aalborg University, Denmark

This paper presents a comprehensive study of biomass particle combustion under realistic conditions as encountered in suspension-fired utility boilers. Wood pellets are pulverized by hammer mills and roller mills, respectively, to prepare solid biomass feedstock. The actual particle density is measured by ethanol displacement method. The particle size and shape distributions are measured by a CAMSIZER analyzer based on dynamical digital image processing, and it is found that roller mills produce quite larger biomass particles than hammer mills do. Combustion tests of the milled biomass particles are performed in a single particle combustion reactor rig under conditions as close to those in suspension-fired utility boilers as possible. A charge-coupled device camera is used to record the whole combustion process. The ignition, devolatilization and burnout times of different biomass particles under different combustion conditions are analyzed. A one-dimensional transient model, which not only properly accounts for the particle surface-ambient flow interaction but appropriately addresses the key intra-particle sub-processes, is also used to simulate single biomass particle combustion. The numerical predictions show a good agreement with the experimental results.

\* Financial support from DONG Energy is gratefully acknowledged.

## THE INFLUENCE OF REACTOR ENCLOSURE AND GAS ENTRAINMENT ON FLAME SPRAYS WITH APPLICATION IN NANOPARTICLE SYNTHESIS

Florian Meierhofer, University of Bremen, Germany; Lizoel Buss, IWT Foundation Institute of Material Science, Germany; Dirceu Noriler, University of Blumenau (FURB), Brazil; Henry França Meier, Regional University of Blumenau (FURB), Brazil; Udo Fritsching, Foundation Institute of Material Science, Univ. Bremen, Germany

Flame Spray Pyrolysis (FSP) has become a well-established technique to synthesize metal oxide nanoparticles with desired material properties for applications in catalysis, optoelectronics and energy storage. In such reactors a metal-organic precursor is dissolved in liquid fuel, dispersed by oxygen in fine droplets and simultaneously ignited by a pilot flame forming a turbulent spray flame. Applying a tube-enclosure to the reactor is an effective way for controlling the fuel-oxygen ratio in order to produce metal particles as well as particles with various coatings. However, gas recirculation is one drawback of such enclosures, causing undesired particle deposition and elevated temperatures in the process downstream. A user defined function for nanoparticle growth is implemented in commercial CFD software to analyze combustion and particle growth of open and enclosed reactor geometries. The model is further validated with experimental temperature and particle size measurements. It is found that by supplying additional amounts of co-flowing gas as "artificial entrainment" into the enclosure-tube reduces the vortex formation and results in similar temperature time histories as found for open reactor settings.

\* Support from grant DFG (Project ID: FR 912 / 33-1) as part of the BRAGECRIM framework and CAPES/DAAD (Process ID: BEX 12369/12-8) is gratefully acknowledged.

## STUDY OF A TURBULENCE DAMPING PROCEDURE NEAR THE INTERFACE OF GAS-LIQUID STRATIFIED FLOW

Vinesh H. Gada, CD-adapco, India; Pranav Punde, CD-adapco, India; Mohit P. Tandon, CD-adapco, India; Roman Vikulov, CD-adapco, Russian Federation

Additional modeling to account for the interfacial turbulence is needed when simulating turbulent multiphase flows. Specifically for gas-liquid stratified flow, boundary layer formation in the vicinity of large scale interface leads to flow laminarization. The interface behaves as a moving wall for the gas phase. To model this phenomena, turbulence in the region close to the interface needs to be damped. In the present work, the role of one such interface turbulence damping procedure for the k-omega turbulence model implemented in STAR-CCM+ within the Euler-Euler methodology is studied for the case of gas-liquid for stratified flow in a channel as well as in a pipe. In this turbulence damping procedure, the specific dissipation rate is estimated using the low Re treatment. This value is then imposed as a Dirichlet condition by casting it into a source term for specific dissipation rate equation. The resulting source terms needs a model calibration factor to control the magnitude of damping. For the present problem, grid-sensitivity as well as sensitivity of model calibration factor are also studied. The results in the form of pressure-drop, fully-developed velocity as well as shear-stress profiles are presented. Traditionally, the turbulence damping procedure has been used for both the phases but in the present work, using the Euler-Euler method, the role of turbulence damping procedure in individual phases is also studied and comparisons against the single-field formulation methodology based Volume-of-Fluid method are also presented.

## MODELING INTER-PHASE MOMENTUM COUPLING VIA THE EXACT REGULARIZED POINT PARTICLE METHOD

Paolo Gualtieri, *Sapienza Università di Roma, Italy*; Francesco Battista, *Sapienza Università di Roma, Italy*; Carlo Massimo Casciola, *Sapienza Università di Roma, Italy*

This contribution presents a new inter-phase momentum coupling model for particle-laden flows in the two way coupling regime, dubbed the Exact Regularized Point Particle (ERPP) method. The approach overcomes the intrinsic difficulties which arise in some circumstances in available approaches like, e.g., the Particle In Cell (PIC) method introduced by Crowe and coworkers since 1977. In the ERPP approach the actual equations for the suspension formed by the fluid endowed with the particles are exploited in an asymptotic form for small particles. This allows a splitting strategy to advance the solution during one time step. The equation for the background flow is advanced in sequence with the equation for the correction field due to the disturbance flow produced by the particles which has been rigorously shown to obey the unsteady Stokes flow. Here we challenge the ERPP in the context of a particle-laden homogeneous shear flow documenting the turbulence modulation operated by the disperse phase for different values of the particle Stokes number and mass loadings. Velocity variances are modified by the inter-phase momentum coupling: turbulent fluctuations are attenuated at increasing mass loading. The alteration is selective in the sense that the different velocity components are modified by the particle back-reaction in a different way. The stream-wise velocity variance is almost unchanged while a substantial attenuation is found for the velocity variances laying in the cross-flow plane and for the Reynolds shear stresses.

## LONG FLEXIBLE FIBERS IN TURBULENT FLOWS

Mostafa Sulaiman, *Institut de Mecanique des Fluides de Toulouse, France*; Eric Climent, *Université de Toulouse, IMFT, CNRS, France*; Blaise Delmotte, *Institut de Mecanique des Fluides de Toulouse, France*; Pascal Fede, *INP Toulouse, France*; Franck Plouraboue, *Institut de Mecanique des Fluides de Toulouse, France*; Gautier Verhille, *Institut de Recherche sur les Phenomenes Hors Equilibre, France*

We studied numerically the transport, deformation and buckling events of an isolated elastic fiber in both the Taylor-Green vortex and Homogeneous Isotropic Turbulent flows.

The fiber is modelled by an assembly of spherical beads. The contact between beads enforces inextensibility of the filament while bending and torsional stresses are accounted for in the Gear Bead Model. In the cellular Taylor-Green flow, the buckling probability is a function of the Sperm number which is a balance between the compression rate of the flow and the elastic response of the filament. The shapes of the filament and its ability to buckle have been successfully validated through comparisons with experiments.

Homogeneous isotropic turbulence is achieved by means of direct numerical simulations of forced 3D Navier-Stokes equations to sustain turbulent kinetic energy. The transport and deformation of long fibers were studied for various flow and fiber material conditions. Two regimes have been identified depending on the ratio of the fiber length and the persistence length which is based on the ratio of the turbulent forcing to the flexibility. The numerical results we obtained are also in a good agreement with existing experimental data. Statistics will be presented and analyzed.

## NUMERICAL SIMULATION OF FLEXIBLE FIBER CONCENTRATION DISTRIBUTION IN A TWO-DIMENSIONAL ANALYTICAL FLOW FIELD FOR DIFFERENT STOKES NUMBERS

Manuel Martinez, *Universitat Rovira i Virgili, Spain*; Jordi Pallares, *Universitat Rovira i Virgili, Spain*; Anton Vernet, *Universitat Rovira i Virgili, Spain*

The present work studies numerically the concentration distributions of flexible fibers in an analytical flow field. Fibers are modeled as a set of rigid cylinders, resulting in a continuous flexible fiber. Forces are applied to the center of mass of each cylinder, thus determining cylinder motion and fiber deformation using a fourth-order Runge-Kutta method to solve the system of ordinary differential equations. The numerical code is validated against experimental data available in the literature. The analytical field is two-dimensional and corresponds to the Arnold-Beltrami-Childress (ABC) flows. Simulations are performed for a wide range of Stokes numbers (St), which relates the characteristic time of the fibers to the characteristic time of the flow. For St below 1, fibers are expected to follow the fluid streamlines, while for St above 1, fibers are expected to have their own inertia. Previous studies with spherical particles from other authors show that there is a preferential concentration of particles in regions of low vorticity, with a maximum preferential concentration for St values around 1. In the present study, it is shown that preferential concentration of fibers for the studied ABC flow has a maximum for St values around 0.3.

\*Support from grant CTQ2013-46799-C2-1-P is gratefully acknowledged.

## DNS OF EVAPORATING DROPLETS IN DECAYING ISOTROPIC TURBULENCE

Michael Dodd, *University of Washington, Seattle, USA*; Antonino Ferrante, *University of Washington, Seattle, USA*

We have performed direct numerical simulation (DNS) of decaying isotropic turbulence laden with thousands of evaporating droplets of Taylor length scale size. The objective of this study is to explain the physical mechanisms occurring in evaporating droplet-laden homogeneous turbulence. To this end, we fully resolve the process of momentum, heat, and mass transfer between the droplets and the carrier fluid. The simulations are performed on a  $1024^3$  grid to resolve each droplet by 32 grids points per diameter with initial Taylor length-scale Reynolds number  $Re=83$ . We show the effects of varying the Weber number on the mean Nusselt number and Sherwood number of the droplets, and on the turbulence kinetic energy budget of the carrier fluid.

\* This work was supported by the National Science Foundation (NSF) CAREER Award, Grant No. ACI-1054591.

## NUMERICAL SIMULATION AND VALIDATION OF ELONGATED FIBRES DISPERSION AND TRANSPORT

Santiago Lain, *Universidad Autónoma de Occidente, Colombia*; Brian Quintero, *Universidad Autonoma de Occidente, Colombia*; Martin Sommerfeld, *Martin-Luther-University Halle-Wittenberg, Germany*

The handling of powders through transportation by a fluid is an important process in numerous industrial and technical areas. These flows are mainly confined flows as for example in hydraulic or pneumatic conveying and in fluidized beds. In most numerical calculations conducted for such particle-laden flow systems, where particles are treated as point masses, it is assumed that they are spherical. However, in practice they are generally non-spherical. Unfortunately, the fluid forces acting on non-spherical particles for larger particle Reynolds numbers are often unknown. Only limited results on correlations for the resistance coefficients (drag, lift and torque) of regularly shaped non-spherical particles in dependence of orientation are available from DNS. With such information, Euler/Lagrange calculations of non-spherical particle laden flows may be conducted by solving for the angular orientation of the particles in the flow and using correlations for the orientation-dependent resistance coefficients. Knowing the particle orientation the wall collision process may be also calculated by solving the impulse equations in connection with Coulomb's law of friction, whose generalised 3D expressions including a novel impact point detection approach are presented. In this work a free-jet laden with rod-like particles is considered (aspect ratio 13). Particles are tracked in the pipe jet flow configuration of Capone et al. (2015). In this configuration the behaviour of the rods inside the pipe determines the particle and fluid conditions at the emerging jet. Numerical computations are compared versus experimental data showing a reasonable agreement.

## DEVIATORIC STRESSES IN SUSPENSIONS OF FLEXIBLE OR CURVED FIBERS

Stefan Lindstrom, *Linköping University, Sweden*; Jelena Andric, *Chalmers University of Technology, Sweden*; Srdjan Sasic, *Chalmers University of Technology, Sweden*; Hakan Nilsson, *Chalmers University of Technology, Sweden*

We employ particle-level simulations to study the rheology of suspensions with rigid or flexible fibers in a Newtonian fluid subject to simple shear flow. The fibers are modeled as chains of rigid cylindrical segments, and the fluid phase is described by the incompressible, three-dimensional Navier-Stokes equations. The two-way coupling between the phases is taken into account. A novel approximate method is proposed for computing the deviatoric stresses in fiber suspensions from the hydrodynamic forces and torques acting on the fiber segments. This makes it possible to conduct a parametric study of the impact of the realistic fiber features, such as flexural rigidity and irregular fiber shapes, on experimentally accessible rheological properties. We demonstrate that when a critical shear rate is exceeded, straight flexible fibers undergo a buckling transition, leading to the development of finite first and second normal stress differences and a reduction of the specific viscosity. On the other hand, the specific viscosity of suspensions of rigid, curved fibers increases with the curvature of the fiber. The knowledge acquired in this study is valuable input for the experimental characterization of fiber morphology and mechanics through rheology.

## BASIC EQUATIONS OF SEPARATION PROCESSES IN SUSPENSIONS WITH GRAVITY, CENTRIFUGAL AND CORIOLIS FORCES

Wilhelm Schneider, TU Wien, Austria

Neglecting inertial and viscosity effects in the bulk flow is a common assumption in the analysis of separation processes in suspensions under the action of gravity or centrifugal and Coriolis forces. While there is a number of examples of particular solutions, the general form of the basic equations for three space dimensions, together with the appropriate boundary and initial conditions, is still uncertain and, with regard to certain aspects, even controversial. An essential point is a proper choice of the variables. Here it is proposed to introduce the mass density of the mixture, the mean mass velocity of the mixture and the total volume flux density as the set of independent variables. After some manipulations, a complete set of basic equations is obtained. It consists of two continuity equations, a generalized drift-flux relation, and two linearly independent components of a vector equation describing the total body force as irrotational. Concerning boundary conditions at solid walls, one has to ascertain whether the total body force at the wall points into the suspension or out of it. In the former case, a thin boundary layer of clear liquid is formed at the wall, whereas in the latter case a thin sediment layer may either stick at the wall or slide along it. Each of those three possibilities leads to a particular boundary condition for the bulk flow in terms of the independent variables.

\* Financial support from AIC Androsch International Management Consulting GmbH is gratefully acknowledged.

## SIMULATIONS TO EXPLORE THE STABILITY OF NANOPARTICLE SUSPENSIONS UNDER HYDRODYNAMIC SHEAR

Minh Vo, The University of Oklahoma, USA; Dimitrios Papavassiliou, The University of Oklahoma, USA

Carbon nanotubes (CNTs) in suspensions tend to form bundles and aggregate. Commercial polymers [such as poly(vinylpyrrolidone), PVP], have been used to stabilize CNT suspensions experimentally.[1] In the present work, we use dissipative particle dynamics (DPD) simulations to determine the configuration of the polymer-grafted CNT suspended particles and the stability of the resulting suspensions under shear. The DPD parameters are determined by comparing literature results with the calculated system properties. [2] The particles are then placed into a Couette flow channel and the shear rate is increased. It is found that there are three zones of shear rates: the zone where the particle is stable, the zone where the polymer starts to detach from the CNT surface, and the zone where the polymer is free in solution and the steric stabilization for the suspension fails. In addition, the effect of the particle curvature on stability is examined by using graphene sheets and particles with a spherical shape.

### References

1. M.J. Kadhun, D.P. Swatske, J.H. Harwell, B. Shiao, D.E. Resasco, *Energy & Fuels*, 27, 6518-6527, 2013.
2. M.D. Vo, D.V. Papavassiliou, *Molecular Simulation*, in print, DOI: 10.1080/08927022.2015.1089989, 2015.

\* Support from grant XSEDE (CTS090017) is gratefully acknowledged.

## SEDIMENTATION OF DILUTE SUSPENSIONS OF SPHERES AT FINITE REYNOLDS NUMBERS

Laurence Bergougnoux, Aix-Marseille University, CNRS, France; Elisabeth Guazzelli, Aix-Marseille University, CNRS, France

The sedimentation dynamics of a dilute suspension of non-Brownian spheres is experimentally examined at particle Reynolds numbers ranging between 0.00001 and 0.1, i.e. extending from the Stokes regime to the small-but-finite inertial regime. While the long-time velocity fluctuations are independent of the Reynolds number in the Stokes regime, it is seen headed for decrease above a particle Reynolds number of 0.0001. The microstructure of the suspension is also seen to evolve with increasing Reynolds number and to depart from random positioning of particles.

## A NUMERICAL STUDY FOR SUSPENSION PARTICLES MIXED WITH DROP FLOWS

Ming Chen, Institute of High Performance Computing, Singapore; Baili Zhang, Institute of High Performance Computing, Singapore; Jing Lou, Institute of High Performance Computing, Singapore

In this study, we simulate a complex flow problem of suspension solid particles and dispersed liquid drops. A combined approach of lattice Boltzmann method and the immersed boundary method was employed for this three phase flow simulation. A pseudo-potential model is used to model the two-phase flow of primary liquid and drops, while the particle suspension model is developed to account for adhesive forces between the suspension particles and interaction with other two liquid phases. In this way, the two phase fluids flow and solid particles are fully coupled. This hybrid method is verified with a few cases studies, such as particle sedimentation in a shallow box, a liquid drop impact on the surface and a lid-driven cavity flow with particles and drops. This new method is further used to analyze the effect of drop size, drop-solid contact angles and drop volume fraction on flow behavior and particle distribution in the flow.

## APPLICATION OF A LEVEL-SET /VOLUME-OF-FLUID METHOD AND AN IMMersed BOUNDARY METHOD TO THE ATOMIZATION OF A LIQUID JET EJECTED FROM A COMPLEX-SHAPED INJECTOR

Trung-Thanh Vu, Normandie Université, France; Thibaut Menard, Normandie Université, France; Christophe Dumouchel, Normandie Université, France

A CLSVOF method is coupled with an immersed boundary method by adopting a relevant work recently published. Via a fluid fraction function over cell face, a modified continuity equation is obtained from the non-penetrating boundary condition on solid frontier. The continuity equation is thus pertinent even in cut-cells. A fractional step approach is used. The incompressibility and non-penetration conditions provide an equation for the pressure in the projection step. In particular, the resulting linear system is demonstrated symmetric positive definite allowing efficient solutions. We implement specific treatments to advect the liquid-gas interface inside the cut-cells. The method is applied to simulate liquid ejection from a reservoir down to external gas medium. The computational domain comprises a reservoir connected to a cylindrical injector and a surrounding gas volume. Therefore, impacts of the internal flow on the breakup can be identified. By varying the nozzle aspect ratio, we expect to observe the influence of the velocity profile on the jet behaviour.

## DEVELOPMENT OF A MULTI-PHASE VOF DYNAMIC SOLVER IN THE OPENFOAM® TECHNOLOGY: AN APPLICATION TO THE SIMULATION OF THE OPENING AND CLOSURE EVENTS IN HIGH PRESSURE GDI INJECTORS

Federico Piscaglia, Politecnico di Milano, Italy; Andrea Montorfano, Politecnico di Milano, Italy; Jerome Helie, Continental Automotive France, France; Francois-Xavier Demoulin, Université de Rouen, France

The simulation of the primary liquid atomization in industrial geometries of atomizers for Gasoline Direct Injection (GDI) requires to accurately resolve the gas/liquid interface and time-resolved turbulence modeling. A dynamic multiphase volume-of-fluid (VOF) dynamic solver has been implemented in the OpenFOAM technology to simulate flow cavitation during needle opening and closure events. The solver is based on the extension of an already existing multiphase two-fluid model; it makes use of ad-hoc implemented cavitation sub-models and, with respect to the solver available in the official distribution of OpenFOAM, it includes some novel features: it is coupled to an advanced fully-automatic parallel mesh motion solver supporting topological changes of the mesh, where dynamic addition and removal of cell layers is performed to simulate the prescribed motion of the injector needle; also, needle opening and closure events are simulated by performing dynamic detach/attach of one mesh region into multiple regions. In the fluid-dynamic solver, a pressure-correction equation to enforce continuity after topological changes in the pressure-velocity coupling algorithm of the segregated solver and to favor the solver convergence without compromising the accuracy of the solution. The resulting solver supports cavitation sub-models as well as LES and hybrid RANS/LES models for turbulence. Code validation is performed against standard test cases available from the literature and against experiments.



## A LES-DEM-VOF METHOD FOR TURBULENT THREE PHASE FLOWS

Gabriele Pozzetti, University of Luxembourg, Luxembourg; Bernhard Peters, University of Luxembourg, Luxembourg

In this work a robust Computational Fluid Dynamic (CFD) - Discrete Element Method (DEM) coupling that can predict free-surface, turbulent flows is presented. A correct prediction of multiphase turbulent flows should ideally be able to capture the discrete dynamics of a dispersed phase (solid particles), and at the same time to take into account the evolution of possible fluid-dynamic instabilities. In this optic a CFD-DEM approach seems promising as it is able to combine the well developed CFD techniques for the study of free-surface flows with the accuracy of the Discrete Particle Method (DPM). A key point of the CFD-DEM method is the coupling between the discrete and the continuous phases. In particular the volume replacement between phases, and the interaction between the discrete phase and the continuous interface must be taken into account in order to perform accurate three phase simulations.

In this work two different approaches to simulate the volume replacement between phases are presented and compared within a four way coupling with a Large Eddy Simulation (LES)-Volume Of Fluid (VOF) solver. The four-way coupled equations for the solid and the fluids are then presented, and some test cases provided in order to evaluate the accuracy of the new solver.

Particular emphasis is posed to study the effects of the coupling on the interface dynamics and stability. The continuous two-phase solver used for the simulations is based on the OpenFoam® architecture, while the discrete phase solver is built on the XDEM code.

## COALESCENCE AND BREAK-UP OF LARGE, DEFORMABLE DROPLETS WITH DIFFERENT VISCOSITIES IN A TURBULENT CHANNEL FLOW

Alessio Roccon, University of Udine, Italy; Francesco Zonta, University of Udine, Italy; Alfredo Soldati, University of Udine, Italy

The dynamics of large, deformable droplets with different viscosities, 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 approach). 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 (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. The droplet deformability alters also the production of Turbulent Kinetic Energy (TKE) inside the droplet and in the proximity of the droplet interface.

\* CINECA 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 Friuli Venezia Giulia under grant PAR FSC 2007/2013 is gratefully acknowledged.

## HIGH FIDELITY SIMULATION OF IMPACT OF HIGH TEMPERATURE AND PRESSURE ON LIQUID JET IN CROSSFLOW ATOMIZATION

Marios Soteriou, United Technologies Research Center, USA; Xiaoyi Li, United Technologies Research Center, USA

Atomization of liquid fuel jets by cross-flowing air is critical to the performance of many aerospace combustors. Recent advances in numerical methods and increases in computational power have enabled the first principles, high fidelity simulation of this phenomenon. In the recent past we demonstrated the first such simulation that was comprehensively validated against experimental data obtained at atmospheric conditions. At combustor operating conditions, however, both pressure and temperature are significantly elevated. In this work we perform a detailed study of the impact of raising these properties on the atomization physics. The resulting study is one of investigating the impact of the liquid-gas density and viscosity ratios, and the evaporation rate. Simulations where these parameters are independently varied are also performed to isolate the impact of each parameter. Results indicate a significant modification of the atomization process at high pressure and temperature. Both, the mechanism of liquid breakup as well and the characteristics of formed droplets are altered under these conditions. These changes are primarily attributed to the impact of the reduced density ratio due to the increased gas pressure.

## 3D, GPU-ACCELERATED AND MPI-PARALLEL SIMULATIONS OF TWO-FLUID FLOWS INTERACTING WITH MOVING RIGID BODIES – APPLICATION IN RENEWABLE ENERGY SYSTEMS

Mehdi Raessi, University of Massachusetts - Dartmouth, USA; Ashish Pathak, University of Massachusetts - Dartmouth, USA

A three-dimensional and fully Eulerian approach to capturing the interaction between a flow of two immiscible fluids and moving rigid bodies is presented. The solid bodies can have arbitrarily complex geometry and can pierce the fluid-fluid interface, forming contact lines. The three-phase interfaces are resolved and reconstructed by using a novel methodology within the volume-of-fluid context. The fluid-structure interaction (FSI) is captured using the fictitious domain method. The Eulerian approach significantly simplifies numerical resolution of the kinematics of rigid bodies of complex geometry and with six degrees of freedom. The methodology was developed in a Message Passing Interface (MPI) parallel framework accelerated with Graphics Processing Units (GPUs). The computationally intensive solution of the pressure Poisson equation is ported to GPUs, while the remaining calculations are performed on CPUs. The performance and accuracy of the methodology is assessed using an array of test cases, focusing on the flow solver and the FSI in surface-piercing configurations. Finally, an application of the proposed methodology in simulations of the ocean wave energy converters is presented.

\*Support from NSF-CBET 1236462 and 1336232 grants is gratefully acknowledged.

## EFFECTS OF INTERFACIAL TENSION AND VISCOSITY ON DROPLETS FORMATION BY BREAKING WAVES IMPINGING ON AN OIL SLICK

Cheng Li, Johns Hopkins University, USA; Joseph Katz, Johns Hopkins University, USA

Oil spilled at sea forms thin films, which are broken up into droplets by waves. This study investigates the effects of water/oil interfacial tension and oil viscosity on the droplet size distributions (DSD) under breaking wave. Experiments are performed in a wave tank using oils with viscosity varying from 9.4 to 306.5 cst, along with MC252 surrogate oil premixed with the dispersant Corexit-9500A at varying dispersant to oil ratios (DOR). The breakup to droplets is visualized using high speed imaging, illustrating the generation of multiple oil-containing structures during successive splash-up cycles. Multi-resolution DSD measurements have been performed using digital holography at 11.1 and 1.1  $\mu\text{m}/\text{pixel}$ , and by fluorescence microscopy at 0.2  $\mu\text{m}/\text{pixel}$ . Results shows shortly after breakup the DSDs are bimodal. The primary mode corresponding to 1-20  $\mu\text{m}$  droplets is sustained for longer periods whereas the secondary mode decays in seconds. Increasing the DOR reduces the size corresponding to both modes, with the primary one falling below 10  $\mu\text{m}$ , and with a substantial fraction falling in the submicron range. There is roughly an order of magnitude increase in the concentration of small droplets ( $<10 \mu\text{m}$ ) as the DOR increases from 0 to 1:500 and then to 1:100 and 1:25.

\* Support from Gulf of Mexico Research Initiative is gratefully acknowledged.

## RECIRCULATION, ENTRAINMENT AND CLUSTER FORMATION IN BOUNDED SPRAYS

Ajloscha Lampa, University Bremen, Germany; Soren Sander, Foundation Institute of Material Science Bremen, Germany; Daniel Schwenck, University of Bremen, Germany; Udo Fritsching, Foundation Institute of Material Science, Univ. Bremen, Germany

Numerical simulations of a bounded spray flow are carried out on different scales, using numerical approaches like Unsteady-Reynolds-Averaged-Navier-Stokes models (URANS) and Large-Eddy-Simulation (LES) models with lagrangian particle tracking for the droplet motion. The history of the droplets/particles in a gas atmosphere is tracked with respect to their environment to estimate particle cluster formation as well as droplet heat and mass transfer kinetics. The local droplet concentration inhomogeneities are identified and quantified with a box-counting algorithm (Garncarek's method). The spray structures in- and outside the central spray cone flow like the recirculation and entrainment zones of gas in between the spray and the chamber wall can be predicted as well as large scale turbulent structures within the spray cone is. The application of perturbations in the gas flow in unsteady spray simulations and its influence on the dispersion of the droplets/particles will be depicted. In the focus of the investigations is the spatial distribution of droplets within the spray and at the spray cone edge. Single droplet clusters have been tracked within the spray cone flow and the interaction of the turbulent motion characteristics of the gas phase and the droplets within the cluster have been analyzed. The numerical results will be compared with 2D-PIV experiments in a lab scale process under isothermal conditions.

Support of this project from the German Research Foundation (DFG) within SPP Process-Spray is acknowledged.



## NUMERICAL ANALYSIS OF DENSE FLUIDIZATION: A BENCHMARK FOR PARTICLE RESOLVED SIMULATIONS OF LIQUID-SOLID AND GAS-SOLID FLUIDIZED BEDS

Amir Esteghamatian, Ecole Centrale de Lyon, France; Manuel Bernard, IFP Energies nouvelles (IFPEN), France; Abdelkader Hammouti, IFP Energies nouvelles (IFPEN), France; Anthony Wachs, University of British Columbia, Canada; Markus Uhlmann, Karlsruhe Institute of Technology (IfH), Germany; Michel Lance, LMFA, France

Detailed data describing the hydrodynamics of a dense fluidized bed of mono-disperse spherical particles is provided. Two test cases of 2000 particles in a bi-periodic domain have been chosen to represent smooth and bubbly liquid-solid and gas-solid fluidizations. The particle-resolved simulations are performed with our multiphase numerical tool based on a Distributed Lagrange Multiplier/Fictitious Domain (DLM/FD) method. The fluid solver is coupled to a Discrete Element Method (DEM) combined with a soft-sphere contact model to solve the Newton-Euler equations with collisions for the particles in a Lagrangian framework. The accuracy of the numerical results are meticulously examined by quantifying the error and assessing the space convergence of the computed solution. A comprehensive study of statistical properties of particle motion is performed in both Eulerian and Lagrangian manners. In addition, the relative influence of hydrodynamic versus collision forces in driving the particle motion is evaluated. A particular attention is also given to the correlation between the hydrodynamic forces and the locally averaged variables. These type of data can provide closure laws for numerical methods at larger scales, where the detailed information at the particle level is not available. Furthermore, present benchmark dataset allows us to adjust the space resolution required for a given error tolerance in the simulation of different flow regimes.

## A NEW DRAG CORRELATION FROM PARTICLE-RESOLVED DIRECT NUMERICAL SIMULATION OF A GAS-FLUIDIZED BED

Junhua Tan, Institute for Thermal Power Engineering, China; Kun Luo, Zhejiang University, China; Jianren Fan, Zhejiang University, China

Although the solid motion in a fluidized bed is dynamic and chaotic, the prevailing drag correlations applied in fluidized bed are derived from the experimental or simulation studies of the flow past fixed or static assemblies of solid particles. Since in the particle-resolved direct numerical simulation (PR-DNS) the hydrodynamic force on a particle is known at any moment without using any drag closures, a laboratory-scale gas-fluidized bed with 9240 spheres was simulated in the present work by means of PR-DNS based on the multi-direct forcing and the immersed boundary method, which is aim to study the drag force in the fluidized bed. The comparison of the "true" drag force in PR-DNS and the "modeled" force evaluated in the framework of an unresolved discrete element model (DEM) using traditional drag correlations shows that the average DEM-type force is about 20-30 % smaller than the DNS-type value. Though the existing drag correlations can capture the variation trend of the drag force with porosity, the values are underestimated. Based on the present DNS database, a new drag correlation is proposed, which is a good fit with the studied case. This correlation is so far the first expression for the drag force in monodisperse dynamic fluidized bed.

\* Support from the Major Program of the National Natural Science Foundations of China (Grant Nos. 51390491, 51390493) is gratefully acknowledged.

## INSTABILITY DUE TO INTERFACIAL TENSION IN STRATIFIED AND CORE-ANNULAR FLOW

Oscar Mauricio Hernandez Rodriguez, University of São Paulo, Brazil; Douglas Martins Rocha, University of São Paulo, São Carlos School of Engineering, Brazil

The particular case of liquid-liquid flow is commonly encountered in the petroleum industry, where a number of applications involve oil-water flow. However, most of the available information has to do with flow in pipes of low viscosity fluids. When it comes to flows of viscous fluids in annular ducts the data are scanty. A general transition criterion has been recently proposed and interfacial tension is considered. A new destabilizing term arises, which is a function of the cross-section curvature of the interface. It is well accepted that interfacial tension favors the stable condition. Nevertheless, the analysis of the new interfacial-tension term shows that it can actually destabilize the basic flow pattern. Such an effect seems to be more important in flows of viscous fluids and in annular-duct flow geometry. The effect of interfacial tension is analyzed and the new flow-pattern transition boundaries are compared with experimental data. The rigorous evaluation of the effects of fluid viscosity and interfacial tension allows the extension of transition models based solely on data of pipe flow of low viscosity fluids.

\*The authors would like to thank STATOIL and ANP for the financial support and CNPq for the research grant.

## NUMERICAL SIMULATION OF PERIODIC CIRCULATING FLUIDIZED BED OF BINARY MIXTURE OF PARTICLES – BUDGET ANALYSIS

Solene Chevrier, INP Toulouse, France; Pascal Fede, INP Toulouse, France; Olivier Simonin, INP Toulouse, France

Since several years the effect of the mesh size has been shown to may have an effect on the prediction of circulating fluidized bed. Typically the cell-size must be of the order of few particle diameters to predict accurately the dynamical behavior of a fluidized bed. Then the Euler-Euler numerical simulations of industrial processes are performed with grids too coarse to allow the prediction of the local segregation effects. A modelling filtered approach is developed where the unknown terms, called subgrid contributions, have to be modelled. Highly resolved simulations consist of Euler-Euler simulations with mesh refinement up to reach a converged solution. Then spatial filters can be applied in order to measure each subgrid contributions appearing in the theoretical filtered approach. Such kind of numerical simulation is very expensive and is restricted to very simple configuration. In the present study, highly resolved simulations are performed for investigating the subgrid contributions in case of binary mixture. A budget analysis is carried out in order to understand and model the effect of subgrid contribution in the global behavior of a periodic circulating fluidized bed.

## DRAG FORCE MODELLING IN DILUTE TO DENSE PARTICLE-LADEN FLOWS WITH MONO-DISPERSE OR BINARY MIXTURE OF SOLID PARTICLES

Olivier Simonin, INP Toulouse, France; Solene Chevrier, INP Toulouse, France; Francois Audard, IMFT, France; Pascal Fede, INP Toulouse, France

Fluid-particle momentum transfer modelling is a key issue for the simulation of gas-solid fluidized beds. In the literature many different empirical correlations can be found for the prediction of the drag force but these correlations are generally not satisfactory for all the flow conditions found in fluidized bed: dilute and dense regime, low and high particle Reynolds number values, mono- or poly-disperse solid mixture. Up to now, in dense particulate flows, the validation of such correlations were performed only by comparison with experiments using mean pressure drop, fluidization or settling velocity measurements and analytical solutions in some limit cases (Stokes flow). Nowadays, the development of numerical techniques, like Lattice Boltzmann Method (LBM), allows to perform Direct Numerical Simulation (DNS) of the flow across dilute and dense particle arrays. Such simulations lead to compute directly the forces acting on the particles which may be used to validate or develop drag force correlations. In the present paper, we show that the simple drag correlation proposed by Gobin et al. (2001), and already used extensively for circulating and dense fluidized bed simulation, is in very good agreement with the LBM results of Tenneti et al. (2011). An extension of the correlation is also proposed for poly-disperse mixture based on the LBM results of Beestra et al. (2007).

## A NUMERICAL INVESTIGATION OF FLOW BOILING INSTABILITY IN HORIZONTAL TUBE

Yufei Wang, Zhejiang University, China; Liang Zhang, Institute of thermal science and power systems, China; Zi-Tao Yu, Institute of thermal science and power systems, China

To better understand the instabilities of gas-liquid stratified flow boiling, a numerical work has been carried out to simulate two-phase flow boiling in a horizontal tube under a relative low heat flux condition. The two phase flow phenomenon was modeled by the Volume of fluid method. The phase-change process was modeled using a numerical code for which appropriate source terms have been added in the mass and energy equations to take into account mass and energy transfer during flow boiling. In order to get more accurate results, the effect of heat conduction of tube has been taken into account using a conjugate heat transfer method. The influences of heat flux, inlet subcooling, and mass flow rate on flow boiling instability were analyzed by the oscillation characteristics of void fraction at outlet of tube. Simultaneously, temperature and void fraction fields have been calculated and compared for various conditions. The CFD predicted results were compared with experimental conclusions and a good agreement was observed. It was also concluded that CFD is a useful tool to simulate the complex phenomenon in flow boiling.

\* Support from the National Natural Science Foundation of China (NSFC) under Grant No. 51406177, the China Postdoctoral Science Foundation (CPSF) under Grant Nos. 2014M551734 and 2015T80613 is gratefully acknowledged.

#### TIME-DOMAIN STUDY ON THE EFFECT OF CHANNEL LENGTH AND CHANNEL NUMBER ON DENSITY-WAVE INSTABILITY OF SUPERCRITICAL FLUID FLOWS IN SINGLE CHANNEL AND PARALLEL MULTIPLE-CHANNELS SYSTEM

Jialun Liu, Xi'an Jiaotong University, China; Hui-Xiong Li, Xi'an Jiaotong University, China; Liangxing Li, Xi'an Jiaotong University, China; Qian Zhang, Xi'an Jiaotong University, China

In this paper, the effect of channel length and channel number on the density-wave instability of supercritical fluid flows in single channel and parallel multiple-channel system is investigated by building a mathematical model using the time-domain method. The new model is developed, on one hand, to take into account the changes in fluid thermo-physical properties along the channels at supercritical conditions, and on the other hand, to study the effect of channel length and channel numbers on the density-wave instability of fluids in single channel and parallel multiple-channel system. The numerical results for both the single channel and parallel multiple-channel system are consistent, and show that when the total heat load on each channel is kept unchanged, the oscillation amplitude of the density wave decreases, and the oscillation period increases, with the increase of channel length, indicating the increases in system stability. However, when the heat flux on each channel is kept unchanged, the oscillation amplitude of the density wave and the oscillation period always increase with the increase of channel length. It is also showed that the stability of the parallel multiple-channel system will decrease with the increase in channel numbers at supercritical pressures.

\* Support from grant No. 2015CB251502 is gratefully acknowledged.

#### DROPLET CONDENSATION/EVAPORATION IN A TURBULENT FREE SHEAR FLOW: MIXING-INDUCED SUPERSATURATION AND INERTIAL DROPLET DYNAMICS AT THE TURBULENT/NON-TURBULENT INTERFACE

Ryan Keedy, Sandia National Laboratories, USA; Alberto Aliseda, University of Washington, USA

We study the evolution of liquid droplets in a turbulent round jet with a strong gradients of water vapor concentration and temperature. These conditions are representative of many industrial and environmental processes, such as cloud-edge mixing in cumulus clouds. The non-linearity in Antoine's law leads to strong supersaturation resulting from the mixing of a high temperature, vapor-saturated stream with another low temperature, dry stream. The inhomogeneity in the instantaneous partial pressure and vapor pressure induces very fast condensation on existing aerosols, leading to spectral broadening of the droplet size distribution. Two models are developed to quantify this behavior. The first one is a simple "stirred-reactor" element in the radial and axial directions that extend the analysis from Strum & Toor 1992, but provides detailed spatial distribution of the axisymmetric time-average droplet sizes along the jet. The second combines an Eulerian description of the carrier turbulent flow with Lagrangian particle elements to compute the evolution of the droplets under the instantaneous supersaturation resulting from the local temperature and vapor pressure. Experiments in a canonical turbulent jet show excellent agreement with the droplet size distribution for the averaged model. A fast one-way coupling code combining finite volume for the carrier gas flow and a Monte-Carlo method for the liquid droplet/vapor concentration PDF is optimized to take advantage of GPU computing, with a significant speed up in this type of particle-based processes.

#### A PRESSURE DISCONTINUOUS CONTROL VOLUME FINITE ELEMENT METHOD FOR MULTIPHASE POROUS MEDIA FLOW WITH DYNAMIC MESH OPTIMIZATION AND PHASE CHANGE

Pablo Salinas, Imperial College London, UK; Dimitrios Pavlidis, Imperial College London, UK; Zhihua Xie, Imperial College London, UK; Christopher Pain, Imperial College London, UK; Matthew Jackson, Imperial College London, UK

We present a new, high-order, control-volume-finite-element (CVFE) method with discontinuous representation for pressure and velocity to simulate multiphase flow of water and hydrocarbons in heterogeneous porous media with mass exchange between the hydrocarbon phases. Time is discretized using an adaptive, fully implicit method. Heterogeneous geologic features are represented as volumes bounded by surfaces. Our approach conserves mass and does not require the use of CVs that span domain boundaries. We demonstrate that the approach accurately preserves sharp saturation changes associated with high aspect ratio geologic domains, allowing efficient simulation of flow in highly heterogeneous models. Computational efficiency is increased by use of dynamic mesh optimization. The use of implicit time integration allows the method to efficiently converge using highly anisotropic meshes without reducing the time-step. Mass exchange is described using a simple 'black-oil' model in which the hydrocarbon phase contains two components (gas and liquid oil) that exchange mass depending on the pressure and/or temperature; this approach is often used to model hydrocarbons with simple phase behaviour and is implemented here for the first time in a CVFE framework with dynamic mesh optimization.

#### VISCOUS VORTEX RECONNECTION IN THE CAVITATING SWIRL FLOW

Mikhail Tsoy, Institute of Thermophysics SB RAS, Russian Federation; Sergey Shtork, Institute of Thermophysics SB RAS, Russian Federation; Sergey Alekseenko, Institute of Thermophysics SB RAS, Russian Federation; Pavel Kuibin, Institute of Thermophysics SB RAS, Russian Federation

We present the results of an experimental study of vortex reconnection process in a real viscous fluid. This experimental work demonstrated for the first time the vortex reconnection processes on the spiral vortex tube formed in a swirl flow in a conical diffuser with sufficiently large swirl parameter values. The result of reconnection is formation of vortex ring while preserving the basic spiral vortex tube. On the original spiral in the reconnection zone, the left-handed Kelvin wave, running up the vortex tube, is generated consistently. A number of topological features of vortex reconnection, such as asymmetry of the process near the ring and spiral tube, formation of two bridges and two threads, as well as formation of external bridges, not associated with the vortex reconnection process, were revealed.

\* This research was supported by the Russian Science Foundation, grant No. 14-29-00093.

#### MULTIPLE HOLDUP PROBLEM IN DEVELOPING LAMINAR-LAMINAR STRATIFIED FLOW IN AN INCLINED CHANNEL

Daniel Thibault, University of Toulouse - INSA - UPS - INP, France; Alain Line, University of Toulouse, France; Roel Belt, Total E&P, France

In this study, the stratified gas-liquid or liquid-liquid pattern is investigated for a laminar-laminar flow in an inclined channel. For some operating conditions (superficial velocities of the phases, channel height, inclination) multiple steady-state solutions for the holdup of the system can be observed. This issue is treated here through a first order ODE model simulating a stationary but developing flow from a non-equilibrium inlet condition to the final downstream solution. Results are analyzed and discussed regarding an approach by minimization of a potential function derived from the catastrophe theory. A good consistency is shown between the conclusions provided by the two methods. Since a supercritical flow condition is set at the inlet and required all along the pipe for the validity of the results, the possibility of hydraulic shock during the longitudinal evolution of the system is also investigated.

#### STATISTICAL MODELLING OF TURBULENCE INTERFACE INTERACTIONS WITH THE CONSERVATIVE LEVEL-SET METHOD

Tomasz Waclawczyk, Warsaw University of Technology, Poland

Based on studies in Waclawczyk M. & Waclawczyk T., Int. J. Heat Fluid Flow 52 (2015) in Waclawczyk T., J. Comput. Phys 299 (2015) we have shown that after conditional averaging of the transport equation for the phase indicator function and conservative closure of the counter gradient diffusive term the general form of the interface evolution equations is obtained, uniting features of the VOF and standard LS methods. Namely, the transport and re-initialization equations of the regularized Heaviside function are equivalent to the transport and re-initialization equations of the signed distance function localized at the interface by the Dirac's delta. The two parameters present in the in the new model equations are the variance in the logistic distribution  $\varepsilon_h = CL$  ( $L$  – is the length scale) and the relaxation velocity  $C$ . For  $\varepsilon_h \rightarrow 0$  and  $C = const.$  sharp representation of the interface is recovered, when  $\varepsilon_h \neq 0$  the new formulation is similar to the standard LS method introduced by Sussman et al., 114 (1994). In the present work we address extension of this new method to the case when  $\varepsilon_h$  and  $C$  are not constant. This allows us to model the evolution of the intermittency region defined as a domain where the interface may be found with non-zero probability, see Brocchini and Peregrine, J. Fluid Mech. 449 (2001). Herein, test cases are presented where we consider different approaches to the solution of the new model equations allowing to account for the existence of the sharp interface and intermittency region in the single computational domain, the coupling of the CLS method with the turbulence models is also addressed.

## NUMERICAL SIMULATION OF BUBBLY FLOWS IN AN AERATION TANK WITH BIOCHEMICAL REACTIONS

Noor Ul Huda Khateeb, University of Tokyo, Japan; Shu Takagi, University of Tokyo, Japan; Xiaobo Gong, Shanghai Jiao Tong University, China; Yoichiro Matsumoto, RIKEN, Japan

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 both aerobic and anaerobic reactions in this bubbly flow based on situation of water and demand. This could be represented as a batch reactor for purification of water. 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.

[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.

## MODELLING OF SLURRY STAGED BUBBLE COLUMN FOR CO<sub>2</sub> CAPTURE IN CARBONATES SOLUTION WITH ENZYME CATALYSIS

Antonio Marzocchella, Università degli Studi di Napoli Federico II, Italy; Maria Elena Russo, Consiglio Nazionale delle Ricerche, Italy; Piero Bareschino, Università degli Studi del Sannio, Italy; Giuseppe Olivieri, Università degli Studi di Napoli Federico II, Italy; Riccardo Chirone, Consiglio Nazionale delle Ricerche, Italy; Piero Salatino, Università degli Studi di Napoli Federico II, Italy  
Carbon dioxide capture by absorption in carbonates solutions supplemented with the enzyme carbonic anhydrase (CA) has been studied as a novel strategy alternative to the amine based absorption. The enzyme immobilized on fine dispersed solids promotes the heterogeneous biocatalysis of CO<sub>2</sub> hydration reaction and the enhancement of CO<sub>2</sub> absorption rate. In the present work, a theoretical model of a slurry absorption unit for CO<sub>2</sub> capture in CA supplemented K<sub>2</sub>CO<sub>3</sub> solutions was developed and solved using the commercial software package Comsol Multiphysics®. The model was developed through the 'tanks-in-series' approach applied to both the gas and liquid phases; the slurry biocatalyst was modelled as pseudo-homogeneous fine dispersed solids. The reversible Michaelis and Menten kinetics described CO<sub>2</sub> conversion by the slurry biocatalyst. The role of mass transfer rate and heterogeneous enzyme catalysis on the CO<sub>2</sub> capture rate and on the CO<sub>2</sub> concentration profiles in the liquid boundary layer was investigated. Simulation results showed that the CO<sub>2</sub> capture rate poorly increased when dissolved CA was used within the enzyme solubility limit, and that in the presence of CA immobilized on fine particles the CO<sub>2</sub> absorption rate was enhanced about three fold with respect to carbonate solvent.

\*Support from grant PON03PE\_00157\_1 'Smart generation' is gratefully acknowledged.

## BUBBLE GENERATED TURBULENCE IN TWO FLUID SIMULATION OF BUBBLE FLOW

Mark Philip Schwarz, CSIRO, Australia; Yuqing Feng, CSIRO, Australia

Bubbles rising in a liquid give rise to increased turbulence of the continuous phase, known as bubble-induced turbulence. Various models have been proposed in the literature to account for this mechanism in two-fluid Reynolds-averaged (RANS) simulations of bubbly flow. However in implementing these models in practical two-fluid simulations, researchers have employed a variety of different terms. Difficulties involved in deriving consistent expressions include definitional issues, the complexity of the flow physics, and the absence of a well-defined dataset for validation. The derivation of the models is analyzed to determine the most physically reasonable form of modification to the two-fluid equations, and these expressions have been shown to be consistent with turbulence values obtained from direct simulation of flow around a single bubble. The source term for turbulence kinetic energy is thus proportional to the product of drag force and bubble-liquid slip velocity, with a proportionality constant determined from the direct simulations.

## MODELLING OF MASS TRANSFER AND CHEMICAL REACTIONS IN BUBBLE COLUMN REACTORS USING OPENFOAM

Mark W. Hlawitschka, University of Kaiserslautern, Germany; Sebastian Drefenstedt, University of Kaiserslautern, Germany; Jan Schafer, University of Kaiserslautern, Germany; Hans-Jorg Bart, University of Kaiserslautern, Germany

Bubble column reactors enjoy wide spread use in process engineering. The phenomena encountered inside these reactors are complex and highly linked. Just in recent years progress has been made in calculating these systems in regard to hydrodynamics and bubble size. This study aims to simulate and predict mass transfer and reaction processes inside bubble column reactors with the help of the open source CFD tool OpenFOAM®.

OpenFOAM contains the multiphase solver multiphaseEulerFoam. This solver allows the calculation of hydrodynamics for any number of continuous or dispersed phases. The present study extends the solver by several models:

A model for mass transfer allows the modelling of absorption of a component from the bubbles into the liquid phase. Modified versions of the chemistry models provided by OpenFOAM enable the calculation of complex reaction systems. The dissolved chemical species inside the liquid phase are governed by transport equations and added source terms.

The developed solver has been tested for the absorption CO<sub>2</sub> in water and the reaction CO<sub>2</sub> in aqueous solution of NaOH. The performed simulations show good agreement to experimental data from literature. Further comparison to own experimental data will be given.

\*This work was supported by the German Research Foundation (DFG), reactive bubbly flows (SPP 1740, DFG HL-67/1-1).

## Influence of bubble bouncing on mass transfer and chemical reaction

Jens Timmermann, Hamburg University of Technology, Germany; Marko Hoffmann, Hamburg University of Technology, Germany; Michael Schlueter, Hamburg University of Technology, Germany

Gases have to be well mixed with a continuous liquid phase to perform a reaction with high yield and selectivity. The time scales of mixing are determined by the transport processes in the boundary layer close to the phase boundary and by bubble swarm turbulence. Tuning the time scales can lead to optimization of yield and selectivity. It has been found that mass transfer in bubble flows is influenced by the boundary layer deformation. Thus it could be expected that due to the change in mass transfer chemical reactions are also influenced. For adjustable interface deformation, an experimental setup was developed which makes it possible to induce bubble collisions and reproducible bubble sizes, shapes, trajectories and collision angles. Investigations for CO<sub>2</sub> in water without chemical reaction show an enhancement of the mass transfer rate in dependence on the collision frequency. In addition, LIF measurements at O<sub>2</sub> bubbles show qualitatively the detachment of O<sub>2</sub> eddies after a bubble collision as well as the influence on a chemical reaction. For quantification of the measured effects, a three-dimensional LIF technique is being developed. The experimental data will then be used for modeling local momentum and mass transfer as well as for validating numerical simulations.

\*The authors gratefully acknowledge the financial support provided by the German Research Foundation within the Priority Program "Reactive Bubbly Flows" (SCHL 617/12-1).

## THE WAKE EFFECT OF SOLID PARTICLES AND COMBUSTING DROPLETS ON TURBULENCE

Lixing Zhou, Tsinghua University, China; Ke Li, Inner Mongolia University of Science and Technology, China; Zhuoxiong Zeng, Shanghai University of Electricity, China

It is known that larger particles enhance gas turbulence due to their wake effect. However, there are less of accurate models, except some semi-empirical and hypothetical models. As for combustng droplets, it is not clear about their wake effect on turbulence. In this paper, numerical studies were made for turbulent gas flows passing a solid particle and a combustng droplet respectively. The continuity, momentum, energy and species equations with their boundary conditions were solved using fine grids, used in LES or DNS. The results indicate that the solid particle's wake effect is surely enhancing the gas turbulence, and the increase of turbulent kinetic energy is proportional to the particle size and the square of the gas relative velocity. A sub-model for turbulence enhancement was proposed and added to the source terms together with the ordinary point-particle dissipation term in two-phase flow equations. Conversely, it is found that the wake effect of a combustng droplet is reducing the gas turbulence, and the drag force of an evaporating/combustng droplet is much smaller than that of a solid particle. Future work remains to be done for developing the droplet-wake-effect term and a new drag model for simulating turbulent reacting gas-droplet flows.

#### DRAG REDUCTION IN DISPERSED MULTIPHASE TAYLOR- COUETTE TURBULENCE: A NUMERICAL APPROACH

Vamsi Arza Spandan, *Physics of Fluids, University of Twente, The Netherlands*;  
Roberto Verzicco, *Univ. Tor Vergata, Italy*; Detlef Lohse, *University of Twente, The Netherlands*

Frictional losses are a major drain of energy in turbulent flows. Several studies report that injection of a small concentration of dispersed phase into a turbulent carrier phase can result in significant drag reduction thus making it of interest for fundamental scientific research and engineering applications. In this work we use numerical simulations to study the effect of a dispersed phase on the flow dynamics of a turbulent carrier phase. Two-phase Taylor-Couette (flow between two co-axial independently rotating cylinders) is simulated using a two-way coupled Euler-Lagrange approach in which the dispersed phase (here spherical bubbles) are treated as point particles with effective forces such as drag, lift, added mass and buoyancy acting on them. For pure inner cylinder rotation, we find that the net drag reduction in the system (computed based on the torque required to drive the flow) depends strongly on the Froude number of the bubbles (ratio between centripetal and the gravitational acceleration) along with the operating Reynolds number. Additionally, we couple these simulations with a phenomenological deformation model to study the deformation and orientation statistics of tri-axial ellipsoidal droplets.

#### EULERIAN MULTI-FLUID SIMULATIONS OF LOCAL LIQUID SPREADING IN A TRICKLE BED

Vivek Buwa, *Indian Institute of Technology Delhi, India*; Arpit Jindal, *Indian Institute of Technology Delhi, India*

Gas-Liquid flows in packed/ porous media beds are important to several commercially important processes e.g. fuel hydrodesulphurization, oil-water-gas flow in oil reservoirs. Often, the local liquid distribution governs performance of aforementioned processes/reactors. While there exists several investigations that report development of Eulerian CFD models capable of predicting overall pressure drop and liquid hold-up, there are only a few papers that investigated the ability of these models to predict the local liquid distribution accurately. In the present work, experiments were performed on air-water flow in a flat 3D column (length: 120 cm, width: 40 cm, depth: 2 cm) using high-speed imaging to study effects of liquid velocity (0-0.023 m/sec), gas velocity (0-0.28 m/sec) and particle size (2, 4 and 8 mm) on liquid spreading. 3D transient Eulerian multi-fluid simulations were performed by implementing phase interaction and capillary pressure models proposed by Attou et al. (1999) and Attou and Ferschneider (1999) in a commercial solver. The comparison of high-speed visualizations and simulations showed that the capillary pressure model was essential to predict the liquid spreading correctly and that simulated effects of gas and liquid velocities and particle size on liquid spreading were in a good agreement with the experiments.

#### FREE SURFACE FLOW IMPROVEMENTS FOR POOR MESH

Vinay Kumar Gupta, *ANSYS Inc., India*; Hemant Punekar, *ANSYS Inc., India*; Srikanth Kvss, *ANSYS Inc., India*

While meshing complex geometries for CFD applications, resolving key areas with good quality mesh is a precondition to obtain accurate results. For free surface flows, handcrafted meshes are often needed. This can lead to increased meshing effort, mesh count, simulation time and hence overall turnaround time. To speed up this process meshing needs to be automated and simulation needs to be performed on as coarse mesh as possible. Employing automated meshing and controlling the cell count may result in poor quality mesh. Subsequent simulation can have issues in solution convergence and can have high interfacial numerical diffusion that may lead to inaccurate results.

Objective of present study is to improve free surface flow numerics that can be applied to poor quality meshes without losing accuracy. Numerical treatments called interfacial anti-diffusion and poor mesh numerics are developed and implemented in ANSYS Fluent R16. Interfacial anti-diffusion treatment helps reduce the numerical diffusion near the interface. Poor mesh numerics treatment is applied to bad quality cells to help stability and convergence. Results of industrial strength case studies are reported with and without these treatments. Simulation results with these treatments are also found to be in good agreement with the experimental data.

#### DRAG REDUCTION IN TURBULENT PIPE FLOW INDUCED BY SUPERHYDROPHOBIC SURFACE

Roberta Costantini, *La Sapienza, Italy*; Francesco Battista, *Sapienza Università di Roma, Italy*; Carlo Massimo Casciola, *Sapienza Università di Roma, Italy*

Superhydrophobic surfaces (SHSs), such as those inspired by the self-cleaning property of lotus leaf, are obtained by means of chemical coating and micro/nano surface texturing. At the macroscopic scales the micro-scale texturing generates a slip velocity at the wall due to vapor bubbles trapped inside the surface grooves (Cassie-Baxter State). Our aim is to address the effect of SHSs on the drag reduction in a turbulent pipe flow through the Direct Numerical Simulation (DNS) approach. In particular, the walls of the pipe are decorated with grooves aligned in the stream-wise direction where the Cassie-Baxter state is artificially enforced via a perfect slip boundary condition on the macroscopic fluid velocity. The work addresses the effects of the dimension of the grooves on the turbulent structures characteristic of wall-bounded flows. The main issue of this peculiar configuration consist in the fact that the turbulent pressure fluctuations are likely to trigger the transition from the Cassie-Baxter to the Wenzel state of the liquid/vapor interface. For this reason, a model mimicking the dynamics of the liquid/vapor interface is introduced in the DNS to account for the effects of the transition events on the turbulent flow.

\*The research leading to these results has received funding from the European Research Council under the European Union's Seventh Framework Programme (FP7/2007-2013)/ERC Grant agreement n. [339446].

#### A NOVEL MULTISCALE MODELLING APPROACH FOR FLOW THROUGH A CYLINDRICAL PACKED-BED REACTOR FILLED WITH POROUS NON-SPHERICAL PARTICLES

Saurish Das, *Eindhoven University of Technology, The Netherlands*; J. A. M. (hans) Kuipers, *Eindhoven University of Technology, The Netherlands*; Niels G. Deen, *Eindhoven University of Technology, The Netherlands*

The present study presents a numerical methodology to simulate flow through a cylindrical packed bed reactor filled with porous non-spherical particles. A multiscale modeling approach has been adopted to tackle the two different length scales, i.e. pores inside the porous particles and voidage between the closely packed particles. The flow inside the internal pores of the particles are not fully resolved, instead they are modeled by a momentum sink term in the Navier-Stokes equations. In this way, simulation results can be obtained in a feasible computational cost with reasonable accuracy. To generate random packings of non-spherical particles, a glued-sphere discrete element model (DEM) available with LIGGGHTS® has been used. An immersed boundary method (IBM) is incorporated to describe the cylindrical wall of the column in the Cartesian computational grid.

The porous particles are considered to be made of open-cell solid foams of different internal porosity and; non-spherical particles with different shapes and size have been considered. The effect of internal porosity, particle size and shape, and the particle to reactor diameter ratio on the reactor performance are demonstrated.

#### EULERIAN MODELLING OF LIQUID FLOW REDISTRIBUTION IN A RANDOM PACKED BED

Dariusz Asendrych, *Institute of Thermal Machinery, Poland*; Pawel Niegodajew, *Czestochowa University of Technology, Poland*

The paper present the numerical study of the liquid spreading taking place during the 2-phase gas-liquid countercurrent flow in an unstructured packed bed composed of Raschig rings. The Euler-Euler 2-fluid approach has been applied to resolve the flow. The impact of packed bed elements on the gas and liquid flows has been introduced into the model by the source terms in the momentum equation. The increased flow resistance has been modelled using an adequate correlations taking into account the packing element type and size, media flow rates and material properties. The liquid redistribution has been simulated by implementing the pdf distribution of the velocity vector orientation in a packed section, which has been adopted from the VOF simulation conducted in a corresponding packed bed with realistic geometry. The volume fraction profiles have been found to be fully consistent with the theoretical predictions provided by the advection-diffusion model. The spreading factor, determining the distributing action of the packing, turned out to be independent of liquid flux confirming the opinions available in the source literature.

\* Support from grant "2-phase flow modelling in complex geometrical structures" provided by the Polish National Science Centre under contract No UMO-2014/15/B/ST/04762 is gratefully acknowledged.

## **SIMULATION OF FIBER SUSPENSION IN A TURBULENT MIXING LAYER WITH AN EULER-LAGRANGIAN METHOD**

Kun Zhou, Wuhan University of Science and Technology, China; Wei Yang, Ningbo University of Technology, China

Fiber suspension flows are found in many industrial processes, such as the pulp and paper industry. Flow shear drives fibers to rotate and changes the fiber orientation, which has important implications on the quality of fiber-based products. In this work, a new Euler-Lagrangian method is developed to simulate the fiber orientation in a canonical mixing layer. Fibers are assumed to be inertialess hence to follow the fluid flow immediately, and spatial diffusion of fibers is neglected. During the simulation, a large number of Lagrangian particles are injected at the inlet, and each particle carries the concentration and orientation of fibers. The concentration and orientation of fibers evolve subject to the flow conditions (shear rate, etc) that the carrying Lagrangian particle undergoes. In the Eulerian framework, the concentration and orientation of fibers in a grid cell are obtained by averaging those carried by the Lagrangian particles inside the grid cell. A Monte Carlo method is used to deal with the orientation distribution, and the stochastic error and the computational efficiency is balanced by choosing appropriate number of samples for the orientation distribution. Simulation results show that fibers tend to align with the flow plane, and continuously rotate within the flow plane. Simulation results demonstrate the feasibility of the Lagrangian particle method.

## **MODELLING OF FIBRE SUSPENSIONS IN TURBULENT FLOW: A PSEUDO-HOMOGENEOUS APPROACH**

Maria Graca Rasteiro, Chemical Engineering and Forest Products Research Centre, Portugal; Carla Cotas, Chemical Engineering and Forest Products Research Centre, Portugal; Fernando Gracia, Chemical Engineering and Forest Products Research Centre, Portugal; Dariusz Asendrych, Institute of Thermal Machinery, Poland

The flow of non-Newtonian fluids is present in many industries such as the pulp and paper industry. In this industry, the piping systems are present in all production lines and their design represents an important step in minimizing the energy costs. The main objective of the present work is to use Computational Fluid Dynamics to model the fully developed turbulent flow of Eucalyptus and Pine pulp suspensions in a pipe. The commercial CFD software Ansys Fluent is used. A pseudo-homogeneous approach is considered in which the main characteristics of the multiphase pulp flow are represented by its rheology and turbulence. The non-Newtonian behaviour of pulp suspensions is introduced into the model by relating viscosity to the local shear rate and consistency. Additionally, a lubrication layer, surrounding the flow core, is taken into account where fibres can be present (consistency is considered dependent on the distance to the pipe wall). Low-Reynolds-number (LRN)  $k-\epsilon$  turbulence model, Chang-Hsieh-Chen, is applied to study the turbulent pulp flow. The standard damping function  $f_\mu$  of the LRN model is modified to include the characteristics of fibres represented by their aspect-ratio. Experimental information is used to tune the model parameters and validate the numerical results by comparison with experimentally-measured pressure drop. A good correspondence was obtained for different flow conditions and pulp consistencies for both Eucalyptus and Pine pulps.

## **FINITE-SIZE PARTICLE TRANSPORT BY TURBULENT CHANNEL FLOW**

Ivana Vinkovic, Laboratoire de Mécanique des Fluides et d'Acoustique (LMFA), France; Wenchao Yu, Laboratoire de Mécanique des Fluides et d'Acoustique (LMFA), France; Marc Buffat, Laboratoire de Mécanique des Fluides et d'Acoustique (LMFA), France

Particle transport by turbulent wall flow is a crucial issue in many environmental problems such as erosion, particle deposition and sand saltation. Despite great technological advances in measurement techniques as well as in numerical simulations, the understanding of the transport mechanism close to the wall is still incomplete. Many numerical studies have been devoted to this issue. By direct numerical simulations (DNS) of a turbulent channel flow coupled with Lagrangian tracking of pointwise particles, Zamansky et al. (2011) found that the rms of streamwise particle acceleration presents a peak close to the wall and that the value of the peak for particles with  $St=5$  is even higher than the rms of streamwise acceleration of the fluid. The authors explained this by considering the acceleration of the fluid at the location of particles. They found that contrary to what is observed in homogeneous isotropic turbulence, close to the wall pointwise particles cluster in regions of high streamwise rms acceleration. The aim of the present study is to explore if this is still the case when size effects are taken into account in the DNS. Therefore, a pseudo-spectral DNS code is coupled with an immersed boundary technique (Uhlmann 2005). 256 finite-size particles with different Stokes numbers and density ratios are tracked at each time step. The velocity and acceleration statistics are analyzed. Particular attention is given to acceleration PDF of the solid phase, the fluid phase and the fluid in the vicinity of particles. The results are compared to the pointwise model and to other experimental or numerical studies on this topic.

## **TURBULENT DISPERSION OF ELLIPSOIDAL PARTICLES IN A WALL-BOUNDED TURBULENT FLOW**

Rafik Ouchene, Université de Lorraine - LEMTA, France; Souria Hamidouche, Université de Lorraine - LEMTA, France; Boris Arcen, Université de Lorraine - LRGF, France; Mohammed Khaliq, Université de Lorraine - LEMTA, France; Anne Taniere, Université de Lorraine - LEMTA, France

To explore the motion of particles in turbulence, direct numerical simulation (DNS) of the Navier-Stokes equations can be coupled to a Lagrangian particle tracking (LPT), which solves the particle equation of motion under the point force approximation. Only quite recently, this numerical tool has been extended to predict the dynamics of nonspherical particles. Nevertheless, such DNS/LPT is generally limited since the models used to evaluate the hydrodynamic force and couple are only valid for low particle Reynolds numbers (creeping flow regime). In order to overcome this limit, we introduce in the present study models recently developed for the hydrodynamic force and couple which are valid outside Stokes regime. To evaluate the effects of these new models on the particle dynamics, we computed the particle-laden turbulent flow recently examined by Mortensen et al. (Phys. Fluids, 2008) with a DNS/LPT method based on theoretical models for the hydrodynamic force and couple valid under creeping flow conditions. We considered the same set of prolate ellipsoidal particles. Nevertheless, in opposite to them, we kept the ellipsoidal major axis length constant, and lower than the Kolmogorov spatial scale. From a comparison with the statistical results provided by Mortensen et al. (Phys. Fluids, 2008), it will be shown that we obtained similar tendencies for the translational motion while some differences are noted for rotational motion. These differences, which are induced by the model used to predict the hydrodynamic couple, will be examined in more details in the full length paper.

## **MRI OF CELLULOSE FIBRE SUSPENSIONS IN A STRAIGHT PIPE: A NEW METHODOLOGY FOR MEASURING THE REYNOLDS STRESS COMPONENTS**

Jordan Mackenzie, KTH Royal Institute of Technology, Sweden; Daniel Soderberg, KTH Royal Institute of Technology, Sweden; Fredrik Lundell, KTH, Sweden; Agne Swerin, SP Technical Research Institute of Sweden, Sweden

The focus of the present work is an experimental study of opaque fibre suspensions in pipe flow. We have extended a conventional 2D phase-contrast MRI (Magnetic Resonance Imaging) protocol to measure the mean streamwise velocity and the variance and co-variance components of the Reynolds stress tensor. We show the reliability of MRI for measuring time averaged turbulent fluctuations by comparing our results to diagnostic data from DNS (Direct Numerical Simulations). The diagnostic plot is a suggested method to judge the reliability of wall bounded turbulence data near the wall, around the maximum rms value, and the outer region. We show that MRI is a good method for measuring turbulent velocity data in straight pipes for water alone. In an effort to determine the cause for a net reduction in turbulent kinetic energy for fibre suspensions in the drag-reducing regime, we present the distribution of the viscous stress, and the variance and co-variance components of the Reynolds stress tensor, relative to the wall shear stress. We hypothesize that the net reduction in turbulent kinetic energy is likely a result of the velocity fluctuations de-coupling.

\* Support from the Bo Rydin Foundation and the RISE Research Institutes of Sweden AB are gratefully acknowledged.

## **EFFECT OF REYNOLDS NUMBER AND CONCENTRATION ON MODULATION OF TURBULENCE BY FINITE SIZE NEUTRALLY BUOYANT PARTICLES**

Guiquan Wang, INP Toulouse, France; Micheline Abbas, LGC-CNRS/University of Toulouse, France; Eric Climent, Université de Toulouse, IMFT, CNRS, France

Fluid flow agitation may be enhanced/damped by big/small particles depending on the ratio between the particle diameter and a typical length scale of turbulence. The driving mechanisms of turbulence modulation and their dependence on physical properties (particle size, concentration and inertia) are still widely open to investigation. The objective of our study is to evidence the interactions between particles and the coherent structures of plane Couette flow. Direct numerical simulations of particle laden flow are carried out with the Force-Coupling Method to model finite size neutrally buoyant particles. Two particle sizes and various concentrations (from 1% to 10%) are investigated for different Reynolds numbers above the transition to turbulence.

Our results show that particle dispersion is determined by a balance between hydrodynamic wall repulsion and turbulent mixing. Due to the presence of particles, large coherent structures are seeded with smaller eddies leading to more homogeneous two-phase turbulence. Streaks formation and the period of regeneration cycle of turbulence are both modified by the presence of particles. Wall friction and dissipation are increasing with solid concentration. We show that an estimate based on rheology of homogeneous particle suspension is flawed due to the finite size of particles and its non-uniform spatial distribution.

## PARTICLE RESOLVED SIMULATION OF A 3D PERIODIC COUETTE DENSE FLUID-PARTICLE FLOW

Oliver Scorsim, INP Toulouse, France; Pascal Fede, INP Toulouse, France; Olivier Simonin, INP Toulouse, France

Dense fluid-particle flow occurs in many industrial applications, such as fluidized bed technology. In order to compute these flows, statistical approaches are developed and, since quite recently, particle resolved simulations may be used as numerical experiments to support validation and development of models. 3D direct numerical simulations of a periodic Couette flow were performed for finite Stokes number and moderate Reynolds number values for dense flows (ranging from 5 to 30%). Bi-periodic conditions are applied in the stream-wise and span-wise direction and the shear is applied by fluid no slip condition on two moving walls. Particles are allowed to cross the periodic boundaries, and they collide without friction on the moving flat walls. The technique (viscous penalty method) allows to track moving solid particles coupled with the direct numerical simulation (DNS) of the interstitial fluid flow (Vincent et al., JCP 2014). Particle-particle hard-sphere collisions are taken into account by DEM as well as the lubrication forces. Such numerical simulations allow to measure fundamental parameters of the flow, such as: particle kinetic and collisional stress, particle fluctuating kinetic energy (granular temperature), particle number density profile, particle velocity correlation function, fluid-velocity fluctuations, fluid dissipation, etc. These results are compared with model predictions in the frame of kinetic theory of rapid granular flow with interstitial fluid effect (Parmentier and Simonin, JFM 2012; Abbas et al. AIChE 2010)

## DYNAMICS OF SHEAR-INDUCED MIGRATION OF SPHERICAL PARTICLES IN PIPE FLOW

Elisabeth Guazzelli, Aix-Marseille University, CNRS, France; Braden Snook, University of Florida, USA; Jason Butler, University of Florida, USA

Shear-induced migration is a conspicuous example of the effects of irreversible dynamics in shearing flows of Stokesian, non-colloidal suspensions. It can drive particles, irreversibly, from the high to the low shear rate regions of the flow. This phenomenon has been identified in Couette viscometers but also in pressure-driven flow through a pipe or channel. It has prompted a large number of experimental studies due to the impact that such migration has on the characterisation of suspension rheology. The objective of the present work is to examine the dynamics of shear-induced migration in a pipe flow, in addition to the steady and fully developed concentration and velocity profiles which has been the focus of most previous studies. We study the large-oscillation flow of a concentrated suspension in a pipe. Particle volume fraction and particle velocity are examined through refractive index matching techniques. The particles are seen to migrate toward the center of the pipe, i.e. from the region of high to low shear-rate. The dynamics of the shear-induced migration process is analyzed and in particular compared to the prediction of the suspension balance model using realistic rheological laws.

## EVIDENCE OF SHEAR-INDUCED MIGRATION IN A PIPE FLOW OF A DENSE EMULSION

Micheline Abbas, LGC-CNRS/University of Toulouse, France; Amelie Pouplin, LGC-CNRS/University of Toulouse, France; Olivier Masbernat, Tlemoine, France; Alain Line, University of Toulouse, France; Sandrine Decarre, IFPEN, France

The flow field of a 70% concentrated non-colloidal o/w emulsion in a pipe has been investigated by means of PIV in a matched refractive index medium. At steady state and in laminar regime, the shape of axial velocity profiles is not parabolic and exhibits a shear-thinning behavior of the dense emulsion, with a flow index of 0.5 and a negligible yield stress (less than 1 Pa). This behavior is consistent with previous rheological studies of dense emulsions in a wide-gap Couette geometry. However, the consistency factor appears to be a growing function of the mean flow velocity  $U_m$ , and instead of a square root law, the pressure drop increases linearly with  $U_m$ . These results are inconsistent with the assumption of homogeneity of the emulsion concentration over the pipe cross-section. Assuming a Newtonian behavior of the emulsion shear viscosity, a low gradient radial profile of drop concentration peaking at the pipe axis is derived from the experimental local flow field. Applying a suspension balance model on the emulsion flow nicely reproduces this profile and the best matching corresponds to a second normal stress difference slightly positive. The wall shear stress based on the calculated effective viscosity (at the wall) ensures a correct prediction of pressure drop.

## ANALYZING THE RELATION BETWEEN FINITE-SIZE RESOLVED PARTICLES AND COHERENT STRUCTURES IN FORCED HOMOGENEOUS-ISOTROPIC TURBULENCE

Markus Uhlmann, Karlsruhe Institute of Technology (IfH), Germany; Agathe Chouippe, Karlsruhe Institute of Technology, Germany

We have performed interface-resolved simulations of a dilute suspension of particles with diameters in the range of 5 to 12 times the Kolmogorov length in the absence of gravity, the Stokes number (based upon the Kolmogorov time scale) being in the range of 2.5-11. The direct numerical simulations feature Taylor micro-scale Reynolds numbers of 120 to 140 and large box sizes compared to the particle diameter. The spatial distribution of the disperse phase is found to be non-random, and the main objective of the present contribution is to analyze the instantaneous locations of the particles with respect to that of the coherent vortical structures of the flow. For this purpose we resort to coherent structure eduction criteria and quantify in a statistical sense the deviation with respect to randomly distributed particles. The deviation is quantified by computing the statistics of the distances between the educed coherent flow structures and these two sets of particles (resulting from DNS and randomly generated). By applying spatial filtering with different filter lengths to the flow fields we are able to determine for each simulation the characteristic scale of the flow structures with respect to which the particles behave most non-randomly. The results will shed further light upon the fundamental mechanisms of interaction between particles and turbulence.

## SHEAR FLOW-INDUCED ENHANCEMENT IN EFFECTIVE THERMAL CONDUCTIVITY OF NANOPARTICLE SUSPENSIONS

Chengzhen Sun, Xi'an Jiaotong University, China; Wenqiang Lu, University of Chinese Academy of Sciences, China; Bofeng Bai, Xi'an Jiaotong University, China

Nanoparticle suspensions have a broad application prospect in a wide range of fields, such as heat transfer enhancement, enhanced oil recovery. Due to the velocity difference between nanoparticles and base fluids, nanoparticle suspensions prefer to be considered as special liquid-solid two-phase fluids and therefore their thermal transport characteristics greatly depend on the corresponding flow process. We experimentally measured the effective thermal conductivity (ETC) of H<sub>2</sub>O+SiO<sub>2</sub> nanoparticle suspensions in a shear flow using an established coaxial-cylinder system. The results showed that the shear flow induced an ETC enhancement and ETC increased asymptotically with increasing the flow shear rate. At lower shear rates, ETC increased with the increase of shear rate; at higher shear rates, ETC reached a plateau value. We attributed this phenomenon to the dispersion of initially aggregated nanoparticles at high flow shear rates. We also found that the dependence of ETC with the flow shear rate was weakened at high temperatures, owing to the high mobility of nanoparticles and relatively uniform nanoparticle distributions. Finally, based on the measured data we proposed a correlation to predict ETC enhancement as a function of flow shear rate, nanoparticle concentration and diameter, and fluid temperature.

\* Support from NSFC for Distinguished Scientists (51425603) is gratefully acknowledged.

## DYNAMICS OF A SHEAR-INDUCED AGGREGATION PROCESS BY A COMBINED MONTE CARLO - STOKESIAN DYNAMICS APPROACH

Graziano Frunigieri, Politecnico di Torino, Italy; Marco Vanni, Politecnico di Torino, Italy

Several methods have been proposed to investigate the dynamics of aggregation in colloidal suspensions. Most resort to the Population Balance Equation, often solved in a stochastic way (Monte Carlo methods); though this method requires a relatively low computational cost, its application is hindered by the lack of suitable models for the description of aggregation rates and aggregates morphology. Conversely, highly accurate, but computationally demanding, description of aggregation events can be obtained by Discrete Element Methods (DEMs), where the motion of each primary particle is tracked by solving its equation of motion.

The present work aims to investigate the mechanism of flow-induced coagulation of a large population of aggregates suspended in a uniform shear flow. The developed method combines a Monte Carlo approach to determine, based on probabilistic considerations, the sequence of aggregation events, and a DEM model, built in the framework of Stokesian Dynamics, to accurately reproduce them; this approach reduces the dynamics of the system to a sequence of events, involving two aggregates at once, which can be simulated in reasonable time by the DEM (differently from a full DEM application to the whole population).

Simulations were performed to extrapolate reliable information about the aggregation rate and to predict the dynamic behavior of the suspension with particular regard to the determination of size distributions and morphologies.

## EVALUATION OF GENTOP CONCEPT FOR THE BUOYANCY-DRIVEN MOTION OF A SINGLE BUBBLE

Fabian Schlegel, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany;  
 Frederike Gauss, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany;  
 Eckhard Krepper, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany;  
 Dirk Lucas, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany

Different approaches for numerical simulations of multiphase flows have been developed, e.g., Volume-of-Fluid methods (VoF) for free surface flows and Euler-Euler (EE) methods for disperse flows. Nevertheless, most industrial applications involve interfacial structures with different morphologies. Therefore, hybrid approaches combining resolved interfaces between the gaseous and liquid phases with EE methods would be desirable. A recent development with promising results is the GENTOP concept [1], which is based on the multi-fluid concept and statistically resolved interfaces that are formed by a separate liquid and gas phase. The present contribution investigates the differences between VoF methods and the GENTOP framework using the example of a buoyancy-driven motion of a single bubble. Different CFD solvers are compared for different density ratios, Eötvös numbers, Morton numbers and grid resolutions. Results are checked against experiments and data obtained from more advanced methods, e.g., level-set methods. For instance, it is known from one-fluid approaches (e.g. VoF) that at least 20-30 cells per bubble diameter are required to reflect the bubble motion sufficiently. Despite a much coarser grid resolution, GENTOP shows a good agreement with experimental results.

[1] Hänsch, S., Lucas, D., Krepper, E. and Höhne, T. (2012) Int J Multiphase Flow, 47, 171-182.

## DETAILED NUMERICAL SIMULATION OF SWIRLING PRIMARY ATOMIZATION USING LEVEL SET METHOD

Changxiao Shao, Zhejiang University, China; Kun Luo, Zhejiang University, China; Jianren Fan, Zhejiang University, China

The detailed numerical simulation of swirling liquid atomization is investigated with level set method. A new mass conservative level set method is employed and the ghost fluid method is utilized to deal with the jump conditions. The inflow boundary condition is carefully conducted and the effect of inflow turbulence in the liquid phase is investigated. The numerical simulation shows that the Rayleigh-Taylor instability is the dominant primary atomization mechanism in the present simulation, qualitatively and quantitatively. Turbulence in the liquid phase enhances the flow instability resulting in a larger liquid radial spatial dispersion and the mixing with the ambient gas. The recirculation zone is smaller and farther from the nozzle exit for the turbulent inlet case than that for the laminar inlet case. The turbulence in the liquid phase significantly reduces the breakup length and the spray cone angle. The interaction between interfaces and vortices indicates that vortices tube directions determine the ligament formation direction.

\* Support from grant LR12E06001, 51222602, 51276163 is gratefully acknowledged.

## DIRECT NUMERICAL SIMULATION OF FLUIDIZATION OF SOLID PARTICLES DUE TO LIFT FORCES

Rohit Maitri, Eindhoven University of Technology, The Netherlands; Frank Peters, Eindhoven University of Technology, The Netherlands; Johan Padding, Eindhoven University of Technology, The Netherlands; J. A. M (hans) Kuipers, Eindhoven University of Technology, The Netherlands

Accurate numerical representation of particle-laden flows is important for fundamental understanding but also for design optimization of complex processes. Liquid-solid flows are fundamentally different from gas-solid flows because of lower density ratio and non-negligible lubrication and lift forces on solid particles. An accurate and efficient numerical method is presented here, which covers a wide range of parameter space of fluid-solid flows. In this interface resolved model, accurate fluid-solid coupling is achieved by incorporating the no-slip boundary condition at the particle surface using an efficient 2nd-order ghost-cell immersed boundary method implemented directly at the level of the discretized fluid equations. A fixed Eulerian grid is used for solving the Navier-Stokes equations and the particle-particle and particle-wall interactions are implemented using the soft sphere collision model. Moreover, the lubrication force is included as a sub-grid scale model due to its range of influence on a smaller scale than the grid size. The particles considered in this study are non-ideal, where the particle surface has roughness and the collisions are dissipative. Finally, the validated numerical model is used to simulate fluidization phenomena due to lift. The results are compared with findings from the literature.

\*Support from grant 13CSER014 of the Foundation for Fundamental Research on Matter (FOM), which is part of the Netherlands Organisation for Scientific Research (NWO) is gratefully acknowledged.

## AN IMPLICIT HIGH ORDER DISCONTINUOUS GALERKIN LEVEL SET METHOD FOR TWO-PHASE FLOW PROBLEMS

Thi Thanh Mai Ta, CNRS and Grenoble university, France; Franck Pigeonneau, UMR 125 CNRS/Saint-Gobain, France; Pierre Saramito, CNRS and Grenoble university, France

An implicit high order time (BDF) and polynomial degree discontinuous Galerkin (DG) level set method is presented in this talk. The major advantage of this new approach is an accurate mass conservation during the convection of the level set function, thanks to the implicit method. Numerical experiments are presented for the Zalesak and the Leveque test cases. The convergence rates versus time and space are investigated for both BDF and DG high orders. The capture of the zero level set interface is then improved by using an auto-adaptive mesh procedure. We finally present preliminary results for an application to the two-phase problem. The problem is approximated by using the discontinuous Galerkin method for both the level set function, the velocity and the pressure fields.

Keywords: Level set method, discontinuous Galerkin FEM, high order methods.

## A MULTI-SCALE EULERIAN-LAGRANGIAN METHOD FOR EFFICIENT DNS SIMULATION OF THE ASSISTED ATOMIZATION OF A PLANAR LIQUID SHEET

Jean-Luc Estivalezes, ONERA, France; Davide Zuzio, ONERA, France

The physics of assisted atomization process involves many spatial scales, where a wide variety of liquid inclusions of different sizes are produced. Large density and viscosity ratios associated with strong shearing flows make the process difficult to reproduce numerically. To correctly capture the dynamics of these spatial scales, each one should be resolved with an appropriate method, each one preserving the physical quantities and the correct interface topology. The simulation should reproduce enough physical time to allow reliable statistics.

To address these problems, an original multi-scale methodology has been developed. It consists of three main items. The first is a coupled Level Set/Volume of Fluid method (CLSVOF) for accurate capture of the primary atomization process. The second an adaptive mesh refinement technique (oct-tree AMR) to dynamically optimize the structured Cartesian mesh. The third consists of a particle tracking algorithm to capture droplet dynamics. An improved Eulerian-Lagrangian coupling has been developed to assure a smooth transition between the Eulerian and the Lagrangian modelling of the droplets, where both models approach their design limits. The overall procedure is tested on simplified numerical tests and validated on a planar liquid sheet assisted atomization case. Results show its ability to reproduce the whole atomization process, from large scale instabilities to small droplet dynamics. A preliminary statistical spray analysis is performed as well, showing a realistic distribution of droplet sizes and average droplet diameter.

## WALL BOUNDARY CONDITIONS FOR NUMERICAL SIMULATION OF BUBBLING FLUIDIZED BEDS

Mohammad Reza Haghgo, University of Saskatchewan, Canada; Donald J. Bergstrom, University of Saskatchewan, Canada; Raymond Spiteri, University of Saskatchewan, Canada

Different wall boundary conditions are proposed in the literature for the particulate phase in the context of a continuum description of gas-particle flows. It is not yet clear how these different wall boundary conditions affect the simulated flow behavior, and which are the most realistic. To investigate this issue, an Eulerian-Eulerian two-fluid model was used to investigate the effect of different particle-phase wall boundary conditions on the numerical prediction of bubbling/slugging gas-particle fluidized beds. Because the bed dynamics are strongly influenced by the motion of the gas bubbles, the impact of wall boundary conditions on the bubble statistics was specifically examined. In addition, the averaged field variables, such as the particle velocity, were compared to published experimental measurements. The comparison shows good agreement between the numerical results, generated by the MFIX code, and their experimental counterparts. It is found that the particle wall boundary conditions do have a significance effect on the predicted flow behavior. However, the effect is more significant for the instantaneous flow behavior and bubble statistics than for the averaged quantities.

## LAGRANGIAN MEASUREMENT OF FORCE ON COARSE OBJECTS IN GAS-SOLID FLUIDIZED BED

Wataru Yoshimori, Hokkaido University, Japan; Tomoki Ikegai, Hokkaido University, Japan; Kyohei Higashida, Osaka University, Japan; Kenta Rai, Osaka University, Japan; Takuya Tsuji, Osaka University, Japan; Jun Oshitani, Okayama University, Japan; Shusaku Harada, Hokkaido University, Japan

Density separation method utilizing a gas-solid fluidized bed has been developed as a new separation technique without usage of water. However, the detailed floating mechanism of coarse objects in gas-solid fluidized bed has not been fully understood. In this study, we measured the force acting on a free-moving coarse particle in gas-solid fluidized bed by Lagrangian sensor system, which has been established to measure the force acting on a particle non-invasively. The Lagrangian sensor, which is called "sensor particle", consists of a 3-axis acceleration sensor, a 3-axis magnetometer, a micro computer, a wireless module and cells. We put the sensor particle into a gas-solid fluidized bed and measure the force wirelessly under the condition of various gas velocity. We also recorded the behavior of the sensor particle by video camera and investigate the relationship between the instantaneous motion of the particle and the force acting on it. The measurement results indicate that the force varies periodically due to rising bubbles in the fluidized bed. Moreover, we compare the force obtained from the Lagrangian measurement with those from the numerical analysis which is performed under the same condition as the experiment. The measurement results are in agreement with the numerical results quantitatively. The both results suggest that the interaction with rising bubbles plays a key role on the floating and sinking of coarse objects in a bubbling fluidized bed.

## COMPARATIVE STUDIES OF SHOCK INTERACTION WITH STATIC AND MOVING CYLINDERS THROUGH IMMERSED BOUNDARY METHOD

Yujuan Luo, Zhejiang University, China; Kun Luo, Zhejiang University, China; Tai Jin, Zhejiang University, China; Jianren Fan, Zhejiang University, China

Shock interaction with rigid circular cylinder is simulated using immersed boundary method coupled with high-order weighted-essentially non-oscillatory (WENO) scheme based on an inviscid approach. Cases with static and moving cylinder which is driven by the surface stress exerted by the fluid are both discussed. In the latter case, the translation of the body and its rotation about the center of the mass are described by the Newton-Euler equations. Schlieren images for the two cases show very different features. However, we find that the approximate position of the primary reflected bow shock is hardly affected by the motion of the cylinder when taking the fixed origin of the coordinate as reference by comparing parameters like trajectories of triple points, stand-off distances and so on for the two cases. Besides, we also analyze influences of the shock-wave strength and cylinder diameter on the reflection patterns. Furthermore, variations of the dynamic drag coefficient are examined. The result indicates that the peak drag for the moving cylinder appears early than the static one in dimensionless time, with much lower value. A correlation to predict the occurrence of the peak drag and its value under different shock-wave strengths and cylinder diameters has been promoted.

## A FLUID-STRUCTURE INTERACTION SOLVER COUPLED TO A VOLUME OF FLUID METHOD

Daniele Cerroni, University of Bologna, Italy; Luca Fancelli, University of Bologna, Italy; Sandro Manservigi, University of Bologna, Italy; Filippo Menghini, University of Bologna, Italy; Ruben Scardovelli, University of Bologna, Italy

The study of Fluid-Structure Interaction (FSI) is becoming of great interest in many engineering applications. In this work we propose a new model to study the deformation of solid structures induced by a two-phase flow. We use a monolithic approach for the FSI problem while a Volume Of Fluid method (VOF) is considered for the reconstruction and advection of the interface. A PLIC method based on the ELVIRA algorithm for the reconstruction and a split algorithm for the interface advection is used. For an accurate reconstruction of the interface a huge number of computational elements are required and a multilevel algorithm coupled to an efficient compression-expansion technique is developed to reduce computational costs and memory requirements. An unstructured computational grid and a fine Cartesian mesh are used for the FSI and the VOF problem, respectively. The interaction between the two different grids is obtained by projecting the velocity field into the Cartesian grid and the Color function into the unstructured grid. This is performed with the MEDMEM libraries included in the Salome platform. The FSI problem is solved with a parallel multigrid C++ Finite Element code. Several test cases are presented.

## EVALUATION OF FLOWABILITY OF COHESIVE POWDER USING POWDER DISCHARGE METHOD BY AIR FLOW

Koichiro Ogata, National Institute of Technology, Oita College, Japan; Koki Ouchi, National Institute of Technology, Oita College, Japan; Seiji Fujita, National Institute of Technology, Oita College, Japan

Handling of the cohesive powder belong to Geldart C particle in the air is difficult extremely to the influence of the cohesive force. Therefore, the evaluation of flowability of powder in the air are important. This study experimentally examined the flowability of the cohesive powder using the operation of the powder discharge from an orifice at the bottom of vessel by the air flow. Two kinds of Calcium hydroxide, that is, Ca(OH)<sub>2</sub> A and B of the different particle diameter were used. As the experimental conditions, the initial void fraction and the interstitial air pressure were varied. We discussed the beginning of the powder discharge and the mass flow rate of the cohesive powder on the test device. As the result, the high pressure was needed to the beginning of the powder discharge of Ca(OH)<sub>2</sub> B of the small particle diameter and the high initial void fraction. We estimated the separation force from Rumpf equation, which substituted the initial interstitial air pressure, the initial void fraction and the particle diameter. In addition, we defined the ratio of separation force and gravity force as the influence of cohesion acting on a particle. From the data analysis, the ratio of separation force and gravity force of Ca(OH)<sub>2</sub> B was higher than that of Ca(OH)<sub>2</sub> A of large particle diameter. This result indicate that the influence of the cohesion force to Ca(OH)<sub>2</sub> B was strong. Furthermore, the mass flow rate of Ca(OH)<sub>2</sub> B through an orifice was decreased. We can infer from this result that flowability of Ca(OH)<sub>2</sub> B is low.

## INVESTIGATION OF GAS-LIQUID TWO-PHASE FLOW INDUCED VIBRATION CHARACTERISTICS

Shuichiro Miwa, Hokkaido University, Japan; Yang Liu, Virginia Tech, USA; Hibiki Takashi, Purdue University, USA; Michitsugu Mori, Hokkaido University, Japan

Two-phase flow induced vibration (FIV) experiments were performed with a 28 mm I.D., 90° elbow to study the internal two-phase flow induced force fluctuation characteristics on pipe bends. Dynamic force signals were obtained directly from the tri-axial force sensors, and experiments with various flow conditions were carried out to cover bubbly, slug, churn and annular flows. The excitation force dependencies on the momentum flux and pressure of two-phase flows were investigated. Based on the current and previously acquired database, improved two-phase flow induced excitation force model was developed based on the local instant formulation of two-fluid model. The impact force effect in slug flow regime, and oscillation due to dynamic wave in annular flow regime were found to be very important in determining the force fluctuations. Newly developed model is adopted with interfacial area concentration correlations and capable of predicting the force fluctuations in small diameter internal two-phase flow.

## MULTIPHASE FORCES ON BEND STRUCTURES

Erik Nennie, TNO, The Netherlands; Stefan Belfroid, TNO, The Netherlands

Piping structures are generally subjected to high dynamic loading due to multiphase forces. In particular subsea structures are very vulnerable as large flexibility is required to cope for instance with thermal stresses. The forces due to multiphase flow are characterized by a broadband spectrum with high amplitudes. These forces differ with flow regimes and are still difficult to predict. Therefore a joint industry project was setup to perform both large scale (6") flow tests and to qualify CFD procedures and settings to calculate these forces with sufficient accuracy. A 1.5D radius bend was fully equipped with force rings (2\*8), piezo strain gauges, classic strain gauges, dynamic pressure sensors (11) and both upstream and downstream a resistance tomography and transparent section and transparent. Both single phase (gas and liquid) as well as multiphase conditions were measured with maximum gas velocities up to 40 m/s and liquid velocities up to 4 m/s. In this paper, results will be presented of the link between the (upstream) hold-up variations and measured forces.

\*This work is done within the JIP Multiphase Flow Induced Vibration run by Xodus, sponsored by BP, Shell, Statoil, Total, Lundin, AkerSolutions, FMC Technologies.



## LAY DOWN SIMULATION OF VISCOELASTIC FLUIDS USING THE HYBRID IMMersed-BOUNDARY METHOD

Andreas Mark, *Fraunhofer-Chalmers Research Centre, Sweden*; Simon Ingelsten, *, Sweden*; Fredrik Edelvik, *Fraunhofer-Chalmers Research Centre, Sweden*

Laydown of viscoelastic fluids is common in several manufacturing processes. In the automotive industry, sealing material is sprayed onto the car body to prevent water leakage into cavities and to reduce noise. To predict the deposition and lower the environmental impact by reducing material consumption, a detailed physical understanding of the process is important. In this work the resulting surface multi-phase flow is modeled and simulated in IBOFlow, the in-house multi-phase flow solver at the Fraunhofer-Chalmers Centre. In the solver the two phase flow is modelled by the volume-of-fluid method and the viscoelastic fluid by a general Carreau rheology model. In the solver the scanned or CAD geometry is handled by the hybrid immersed boundary method and the material interface is resolved by the adaptive anisotropic octree grid. The resulting hanging octree and triangular nodes along the geometry are automatically handled by the immersed boundary method. To boost the computational performance the simulation domain is in a novel way dynamically divided into an active and an inactive part. The governing equations are only assembled and solved for in the active part, which is determined by the local position of the injection nozzle. The interface between the active and the inactive cells are handled by symmetry boundary conditions and the pressure is always set for a point inside the active domain. Finally, the simulation framework is successfully validated for a number of real sealing beads on a Volvo XC60.

## INVESTIGATION OF AIR INJECTION PARAMETERS TOWARD OPTIMUM FUEL SAVING EFFECT FOR SHIPS

Inwon Lee, *Pusan National University, Republic of Korea*; Seong Hyeon Park, *Pusan National University, Republic of Korea*; Ho Hwan Chun, *Pusan National University, Republic of Korea*

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 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).

## RECONNECTION OF CAVITATING VORTEX TUBES WITH FORMATION OF VORTEX RINGS IN A SWIRL FLOW

Sergey Skripkin, *Institute of Thermophysics SB RAS, Russian Federation*; Pavel Kuibin, *Institute of Thermophysics SB RAS, Russian Federation*; Sergey Shtork, *Institute of Thermophysics SB RAS, Russian Federation*; Sergey Alekseenko, *Institute of Thermophysics SB RAS, Russian Federation*

We present the experimental results of observation of viscous vortex reconnection in swirling flow. The studies were performed using a closed hydrodynamic loop with the vertical layout. The working section is a simplified model of hydraulic turbine draft tube. The flow was swirled above the diffuser inlet by a vane swirler, which ensured the flow swirl parameter  $s = 0.47$ . Formation of isolated and linked vortex rings by reconnection between different parts of a spiral vortex tube formed in the swirl flow in the expanding channel is observed in detail. The vortex structures were visualized by the vapor bubbles formed due to cavitation in the zones of low pressure. The formation of an isolated vortex ring has been studied in detail using high speed visualization technique. Getting into the area of high pressure cavitation vortex ring collapses generating pressure pulsations. We consider this mechanism as a possible source not periodic pressure pulsations in hydraulic turbines in non optimal conditions.

## EFFECTS OF AZIMUTHAL ANGLE OF AERATION HOLES ON FLOWS INSIDE AND OUTSIDE A BUBBLE DIFFUSER PIPE

Ryo Sato, *Kobe University, Japan*; Tatsuya Miyayoshi, *Kobe University, Japan*; Kosuke Hayashi, *Kobe University, Japan*; Akio Tomiyama, *Kobe University, Japan*

Experiments on aeration from a bubble diffuser pipe, which has evenly positioned five aeration holes, are carried out to investigate the effects of the azimuthal angle of the holes on the flows inside and outside the pipe. The azimuthal angle, which is defined as the angle of the hole center to the vertical axis, varies from 0 to 180 degrees. The hole diameters tested are 1, 3 and 5 mm. Air and water are used for the gas and liquid phases, respectively. The total gas flow rate ranges from  $2 \times 10^{-5}$  to  $7 \times 10^{-4}$  m<sup>3</sup>/s. The gas flow rate from each aeration hole is measured by capturing generated bubbles. The flows inside and outside the pipe are observed using a high-speed camera. Stratified or wavy two-phase flows are formed inside the pipe. At azimuthal angle  $\theta$  and low total gas flow rates, the aeration is likely to be non-uniform due to the transition of interfacial waves to liquid slugs. The increase in the azimuthal angle of the holes restricts the liquid height inside the pipe less than or equal to the hole, which results in the suppression of slugging. Hence uniform aeration can be realized by increasing the azimuthal angle even at low total gas flow rates. A graphical correlation of the aeration state is also presented.

## VALIDATION OF CFD PREDICTION OF SLOSHING EFFECTS FOR NUCLEAR REACTOR SEISMIC SAFETY

Konstantinos Myrillas, *von Karman Institute for Fluid Dynamics, Belgium*; Philippe Planquant, *von Karman Institute for Fluid Dynamics, Belgium*; Jean-Marie Buchlin, *von Karman Institute for Fluid Dynamics, Belgium*; Marc Schyns, *SCKCEN, Belgium*

Seismic safety of nuclear power plants is of primary importance, highlighted by the events of the Fukushima Daiichi nuclear disaster in March 2011. Seismic excitation can set in motion the liquid coolant inside the reactor vessel, a phenomenon known as sloshing. This can introduce extra loads on the nuclear reactor structure or internal components, especially when heavy liquid metal is used as coolant. Advanced tools are needed to evaluate the effects of sloshing on the nuclear reactor structures due to the non-linear nature of the phenomena. In the present study, sloshing in a Heavy Liquid Metal reactor is investigated for the safety analysis of the MYRRHA, a versatile experimental reactor using Lead-Bismuth-Eutectic alloy as coolant, currently under development at the SCK-CEN. Sloshing is simulated using the CFD code OpenFOAM, with the VOF method for interface tracking and a moving mesh approach to apply the seismic excitation. The results are compared to dedicated experiments with a force-moment balance and optical methods for validation. Good agreement is found for cases with linear sloshing, below and above the first sloshing mode, whereas discrepancies are observed for the resonance case. The scaling rules for extrapolation of the results from the reduced scale experimental model to the reactor prototype are verified, while the effect of the geometry on the natural frequency of sloshing is discussed. The CFD simulations are shown to be a valuable tool for sloshing prediction for the nuclear reactor safety assessment.

## LIGAMENTS FORMATION IN CO-FLOWING JETS WITH VISCOSITY STRATIFICATION

Jean-Christophe Loiseau, *Royal Institute of Technology (KTH), Sweden*; Outi Tammsisola, *University of Nottingham, UK*; Luca Brandt, *KTH Royal Institute of Technology, Sweden*

Two-phase flows are encountered in numerous industrial applications such as jet atomization in fuel injectors. When subject to shear, the interface between the different phases can undergo instabilities eventually yielding to wave overturning and the formation of streamwise-oriented ligaments. Such ligaments being precursors to droplets entrainment, a better understanding of their formation mechanisms is crucial for numerous engineering situations. Based on experimental observations on density-constrained co-axial jets, Marmottant & Villermaux (JFM 2004) proposed a scenario involving secondary instability of the interface of Rayleigh-Taylor type. However, as pointed by O'Naraigh et al. (JFM 2014), streamwise-oriented ligaments are also observed in density-matched but viscosity-constrained flows, where the Yih mechanism is the dominant one, thus ruling out the Rayleigh-Taylor instability as the only mechanism responsible for their formation. Based on direct numerical simulations of the non-linear flow, the present work aims at shedding some light on the role of surface tension and viscosity stratification on the formation of streamwise-oriented ligaments and on their dynamics. The solver used (TPLES) is based on a finite-volume discretization of the equations and uses a levelset approach to capture the interface. Analysis of the non-linear flow fields is based on the joint use of Principal Components Analysis and Dynamic Modes Decomposition.

## JUMPING HOOPS ON LIQUID

Eunjin Yang, Seoul National University, Republic of Korea; Ho-Young Kim, Seoul National University, Republic of Korea

Small aquatic living creatures, such as water striders, and fishing spiders, are able to jump off water to a height several times their body lengths without sinking. Inspired by their exceptional motility on water, we study a simple model using a flexible hoop to provide fundamental understanding and a mimicking principle of jumping on water. Behavior of a hoop, which is initially bent into elliptic shapes from their equilibrium circular shapes, is visualized with a high speed camera after the jump is triggered. We observe jumping of hoops with different materials, dimensions, and wettabilities on different liquids and calculate the forces on the hoop and resulting momentum transfer between the hoop and the liquid. We also compare the effect of the wettability of the hoop surface on the work required to lift off the liquid. Our analysis allows us to predict the jumping performance including the jump height and efficiency of the hoop on liquids in comparison to that on ground, and to discuss the evolutionary pressure rendering small aquatic creatures adopt such dynamic characteristics.

## INFLUENCE OF DC ELECTRIC FIELD UPON THE PRODUCTION OF OIL-IN-WATER-IN-OIL DOUBLE EMULSIONS IN MM-SCALE CHANNELS.

Federico Alberini, University of Birmingham, UK; Romain Enjalbert, University of Birmingham, UK; Yann Van Crombrugge, University of Birmingham, UK; Davide Dapelo, University of Birmingham, UK; Mark Simmons, University of Birmingham, UK

A high speed imaging technique has been applied to determine the formation of low viscosity oil and water drops in an oil-in-water-in-oil (O/W/O) double emulsion under the effect of a DC electric field. Using a test cell, the DC field is applied between either a 0.8 mm or 1.6 mm internal diameter (ID) needle (from which the O/W emulsion emits upwards into a continuous oil phase) and a grounded metal ring which was located at 130 or 170 mm from the needle top. Without the application of the electric field (EFS), several flow regimes were observed; stable formation of both the O/W/O emulsion and the O/W emulsion upstream of the cell was possible over a range of Reynolds numbers from 80-100. Application of the electric field was found to have little effect below 60 kV m<sup>-1</sup>, beyond this critical value there was a significant effect upon the flow regime and consequently upon the drop size and emulsion structure. An electrostatic model has been presented to identify the critical electric field strength for each presented condition. A qualitative and quantitative discussion of the impact of the electric field strength upon flow pattern and emulsion structure and a quantitative analysis of droplet size are presented. The work shows the potential of electric fields for the controlled creation of complex emulsion droplets.

## EFFECT OF SOLID FOAM ON SEGMENTATION FOR CO<sub>2</sub> DISSOLUTION FOR MILLI-CHANNEL TAYLOR FLOW

Serge Simoens, CNRS, France; Guanyu Zhou, CNRS, France; Regis Philippe, LGPC/CNRS-CPE Lyon-Université de Lyon, France; Marie-Line Zanotta, LGPC/CNRS-CPE Lyon-Université de Lyon, France; Pierre Valiorgue, University Claude Bernard Lyon I, France

We measure gas-liquid transfers for segmented flow (Taylor) and a porous solid part (foam) with microscopy Induced Fluorescence (IF). This allows to quantify concentration field of dissolved CO<sub>2</sub> in a pH-basis aqueous solution with or without foam. We used a channel of 2 mm x 2 mm squared section and 70mm long with perpendicular and cylindrical 1 mm diameter channels spaced by 10 mm, allowing injection of liquid and gas phases. The basic solution of pH 10 with fluorescein dye for « μ-pH-IF » allowed to obtain concentration fields of dissolved gas from recorded images of fluoresced intensity. The foam was constituted of block of 2mm x 2mm x 20 mm, of Nickel-Chroma porous medium, centered at 30 mm downstream the gas inlet. We use and inverted ZEISS epifluorescent microscope with an HBO 100 lamp, 2.5X lense and Dichroic filters. Recording was done with fast acquisition CMOS Camera Hamamatsu. Analysis of recorded images was conducted with a Matlab soft. It allowed also to obtain size of gas bubbles and liquid segments giving global phase velocity. This was done for various axial locations and various G/L flow rate ratio, without and with solid foam. This allowed to obtain statistical mean concentration fields of dissolved gas and G/L interface geometrical information and global K<sub>La</sub> mass transfer coefficient.

## SIMULATIONS OF A SHOCK WAVE - PARTICLE CLOUD INTERACTION IN A SHOCK TUBE

Bertrand Rollin, Embry-Riddle Aeronautical University, USA; Subramanian Annamalai, University of Florida, USA; Frederick Ouellet, University of Florida, USA; Rahul Koneru, University of Florida, USA

Shock-particle interaction is a complex multiphase phenomenon occurring in natural flows such as volcanic eruptions or in engineering applications such as multiphase explosives. In this work we conduct a numerical experiment studying the evolution of a cloud of particles in a shock tube following its interaction with a shock wave. Using a Eulerian-Lagrangian approach, we track the trajectory of computational particles initially at rest in a square-shaped domain laying on the bottom surface of a shock tube, but not extending all the way to the top. When the cloud of particle is swept by the shock wave, it is expected to experience interfacial instabilities similar in nature to Kelvin-Helmholtz and Richtmyer-Meshkov instabilities. Our study uses state-of-the-art particle force models including unsteady forces, and explores the effects of the relative strength of the shock wave, of the particles intrinsic characteristics (size, mass, etc.), and of the initial volume fraction of the particulate phase on the behavior of the cloud of particles. Complexities associated with compaction of the cloud of particles are avoided by limiting the simulations to modest initial volume fraction of particles.

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

## OIL-WATER PLUG FLOW FORMATION IN SMALL CHANNELS

Maxime Chinaud, University College London, UK; Evangelia Roumpea, University College London, UK; Lyes Kahouadji, Imperial College London, UK; Omar K. Matar, Imperial College London, UK; Panagiota Angeli, University College London, UK

In this study, the formation of two-phase plug flow in microchannels has been studied both numerically and experimentally. Experiments are performed on T-junction while velocity fields are obtained using two innovative micro-PIV techniques, bright field and two-colour. It was found that the velocity profiles within the forming plugs depend on the aqueous to organic phase flow rate ratio with a vortex forming at the tip of the plug under certain conditions. The interface curvature at the rear of the forming plug changes sign at the later stages of plug formation and accelerates the thinning of the meniscus leading to plug breakage. The spatially resolved velocity fields show that the continuous phase resists the flow of the dispersed phase into the main channel at the rear of the plug meniscus and causes the change in the interface curvature. The experimental results are compared against direct numerical simulations. A new solver is used capable of massively parallel simulations of three-dimensional multiphase flows which is based on high fidelity hybrid Front-Tracking/Level Set algorithm for Lagrangian tracking of arbitrarily deformable interfaces and a precise treatment of surface tension forces, interface advection and mass conservation.

## EXPERIMENTAL INVESTIGATION OF AIR WATER HYDRODYNAMICS IN PEMFC MICRO GAS CHANNELS

Sneha Malhotra, IIT roorkee, India; Sumana Ghosh, Indian Institute of Technology Roorkee, India

Water management plays an important role in commercialization of proton exchange membrane (PEM) fuel cell. A well hydrated membrane operated with a saturated gas may lead to the condensation of water drop in gas flow channels of these fuel cells. As a result, the performance of cell decreases due to blockage of reactant site by water. Present work investigates the behavior of two-phase flow in a series of 6 serpentine mini-channels in a acrylic resin sheet. Each channel has a cross sectional area of 1mm<sup>2</sup> and a length of 50mm. Flow distributions are observed for a wide range of air and water using a high speed camera. Later the channel surfaces are treated application of PTFE coating such that there is a variation of contact angle at the bottom surfaces of the channels. Flow distributions for the treated and untreated channels are compared. It has been noted that flow patterns are influenced by the surface contact angle. The exit time for different volume of water on both the cases are also investigated. It has been observed that the treated surfaces facilitates breakage of films due to reduction in contact area and usually has lower exit time than the treated surface.

\* Support from Department of Science and technology is gratefully acknowledged.

## DIRECT NUMERICAL SIMULATION OF A SINGLE RISING BUBBLE WITH INTERFACIAL SPECIES TRANSFER AND CHEMISORPTION IN THE LIQUID PHASE

Daniel Bezecny, Hamburg University of Technology, Germany; Daniel Deising, Technische Universität Darmstadt, Germany; Sebastian Eller, Hamburg University of Technology (TUHH), Germany; Holger Marschall, Technische Universität Darmstadt, Germany; Michael Schlueter, Hamburg University of Technology, Germany; Dieter Bothe, Technische Universität Darmstadt, Germany

The transfer of species from rising bubbles into surrounding liquids is a major issue in process engineering since it can be the limiting factor in chemical reactors at industrial scale. Still, the species transfer for industrial relevant substances and process parameters is poorly understood due to the complexity of the physics at small scales. Many methods for the Direct Numerical Simulation of interfacial species transfer are affected by numerical instability, low accuracy or the loss of conservativeness or boundedness. To overcome this, a single-field model for species transfer termed Continuous Species Transfer model (CST) has been developed at the Center of Smart Interfaces. The CST is combined with an algebraic Volume-of-Fluid interface capturing method. The implementation is based on the interFoam solver family from the open source CFD platform OpenFOAM. This presentation comprises a study of single rising bubbles in a quiescent liquid at a realistic range of Henry coefficients and diffusivity ratios. The results are compared to 2D and 3D data from the literature and experiments. The main focus lies on the dependency of Sherwood Numbers on Schmidt and Reynolds Numbers in different hydrodynamic regimes, while also considering the chemisorption in the liquid phase.

## LARGE-EDDY SIMULATION OF SODIUM RELEASE FROM PULVERIZED-COAL JET FLAMES

Kaidi Wan, Brunel University London, UK; Jun Xia, Brunel University London, UK; Zhihua Wang, Zhejiang University, China; Luiz Wrobel, Brunel University London, UK; Kefa Cen, Zhejiang University, China

The release of sodium during turbulent pulverized-coal combustion can cause fouling and corrosion of heat transfer surfaces in industrial coal-fired boilers. In the present study, large-eddy simulation (LES) is used to investigate the sodium release characteristics in pulverized-coal jet flames. The chemical percolation devolatilization model is employed to describe the pyrolysis of each coal particle, and the partially stirred reactor (PaSR) model and the kinetic/diffusion model are used for modeling gas-phase volatile and solid-phase char combustion, respectively. The kinetics of sodium release determined from burning a single coal particle is incorporated. To validate the developed method, LES is applied to a pulverized-coal jet flame ignited by a preheated gas flow for three coal-feeding rates reported in Proc. Combust. Inst. 33 (2011) 1771-1778. The simulation results of gas temperature, coal burnout and lift-off height agree well with the experimental data. The sodium release characteristics during the pyrolysis and char combustion stages are then analyzed. The results demonstrate the potentials of using LES for modeling the release of impurities within coal during pulverized-coal combustion.

\* Support from the China Scholarship Council, the Engineering and Physical Sciences Research Council (EPSRC) of the UK, the National Basic Research Program of China (2012CB214906), National Natural Science Foundation of China (51390491, 51422605), and the ARCHER UK National Supercomputing Service are gratefully acknowledged.

## EULERIAN MULTIPHASE PREDICTIONS OF TURBULENT BUBBLY FLOW IN A STEP-EXPANSION CHANNEL

Dimitrios Papoulias, CD-ADAPCO, UK; Andrew Splawski, CD-ADAPCO, UK; Alexander Vichansky, CD-adapco, UK; Simon Lo, CD-ADAPCO, UK

In multiphase flow regimes the exchange of characteristic physical properties between bubbles and their carrier liquid-phase, such as momentum, heat and turbulence, correlates with the dynamic behavior of the dispersed phase as well as the activity of eddies. In this paper we examine the reciprocal interactions between bubbles and turbulent structures, encountered downstream of a step expansion channel. For the purposes of this analysis computational fluid dynamic (CFD) simulations are performed, using the Eulerian-Eulerian two-fluid model available in STAR-CCM+. The liquid phase and the homogeneous bubbles are coupled by integrating momentum source-terms due to drag, lift, virtual mass and turbophoresis forces as well as by taking into account turbulence dispersion effects. Moment closure techniques included two-equation kappa-epsilon formulations (k- $\epsilon$ ) and higher-order anisotropic Reynolds-stress models (RSM). The realizability of empirical turbulence models derived by Issa (2004) and Troshko (2001) is also evaluated in this study. In addition to the Reynolds-averaged Navier-Stokes approach (RANS) also large-eddy filtering techniques (LES) were used, based on the Smagorinsky sub-grid scale model (SGS). The calculated flow predictions are validated against experimental measurements reported in the work of Bel F'Dhila (1990), which characterized the spatial profiles of the mean and fluctuating velocity components of the fluid-phase as well as the bubble's volume-fraction distribution at different sections of the step-channel.

## WHAT IS THE EFFECT OF TURBULENCE ON THE HETEROGENEOUS REACTION RATES IN THE HIGH AND LOW DAMKÖHLER NUMBER REGIMES?

Jonas Krueger, Norwegian University of Science and Technology, Norway; Nils Erland L. Haugen, Norwegian University of Science and Technology, Norway; Terese Lovas, Norwegian University of Science and Technology, Norway

Heterogeneous reaction rates depend, amongst other factors, on the particle number density through the Damköhler number (Da). For diffusion controlled heterogeneous reactions, two distinct regimes exist: 1) at low Damköhler numbers, where diffusion of reactants to the single particle is the limiting factor and 2) at high Damköhler numbers, where diffusion to the local particle cluster is the limiting factor. This work looks at the influence of the flow Reynolds number in these two regimes, using Direct Numerical Simulation with isotropic turbulence in an isothermal flow. The reaction that is used is a simple one-step heterogeneous consumption of a reactant at the particles surface. In the case of low Da, the reaction rate scales with the Da, and compared to the laminar case the consumption rate is increased by the turbulence due to the enhanced mass transfer to the individual particles. For large Da, however, the reaction rate limit is independent of Da and the turbulence is found to slow down the overall consumption rate due to the clustering of particles.

## FLOW PATTERN VISUALIZATION OF HEATED KEROSENE AT SUB- AND SUPER-CRITICAL PRESSURES

Zhaohui Liu, Xi'an Jiaotong University, China; Hui Pan, Xi'an Jiaotong University, China; Song Feng, Xi'an Jiaotong University, China; Zhuqiang Yang, Xi'an Jiaotong University, China; Fei Wang, Xi'an Jiaotong University, China; Qincheng Bi, Xi'an Jiaotong University, China

Flow patterns of boiling of a kerosene-kind hydrocarbon fuel were visualized and studied in a 2 mm circular quartz channel, before which the kerosene fuel were electrically heated in a 2 mm nickel alloyed channel. Effects of pressure, heat flux, and mass flux on flow pattern were studied. As fuel temperature increased, different flow patterns were observed at subcritical pressures, such as bubbly flow, slug flow, stratified flow and annular flow. An interesting phenomenon we found is that while the flow pattern shifted the fuel color in the quartz channel became darker gradually, from transparent to slight yellow, slight black, and then dark black. At supercritical pressures, black boundaries were observed at both the top and bottom side of the quartz channel. Chemical reaction should be a must considered influence on the color change for different flow patterns. However, while fuel temperature was approaching the critical temperature, it is still a puzzle that the black color of the fuel, which is obviously different with the critical opalescence phenomenon, appeared at a wide range of fuel temperature at both sub- and super-critical pressures.

\* Support from National Natural Science Foundation of China (grant No. 21306147) is gratefully acknowledged.

## DISPERSION OF FINITE SIZE DROPLETS AND SOLID PARTICLES IN ISOTROPIC TURBULENCE

Michele Rosso, University of California, USA; Hengjie Wang, University of California, USA; Said Elghobashi, University of California, USA

The paper presents a comparison between the dispersion characteristics of finite size liquid droplets and finite size solid particles in isotropic turbulence at moderate values of  $Re_\lambda$ . The droplets and particles have equal diameters (larger than the Kolmogorov length scale) and equal densities. The immersed boundary method is used for direct numerical simulations (DNS) of the solid particles. The level set method is used for DNS of the droplets 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

## LARGE-EDDY SIMULATION OF PARTICLE TRANSPORT IN A VOLCANIC PLUME

Hitoshi Suto, Central Research Institute of Electric Power Industry, Japan; Yasuo Hattori, Central Research Institute of Electric Power Industry, Japan; Kiyoshi Toshida, Central Research Institute of Electric Power Industry, Japan

The large-eddy simulation of a volcanic plume based on a multi-fluid approximation was performed to investigate the transport process of pyroclastic material in a plume. As the subgrid-scale (SGS) turbulence models, the Smagorinsky model and the Yuu model (Yuu et al. 2001) for gas phases, and the equation proposed by Hinze (1975) for particle phases were adapted. The validity of this code was ascertained by comparison with existing experimental data of basic flows. The numerical results for a volcanic plume showed that (1) the spatial distributions of particles with different sizes are not identical, (2) these nonidentical spatial distributions are mainly caused by the difference in the mean vertical velocity of particles with different sizes especially in the umbrella region in a volcanic plume and are seldom affected by the turbulence properties. This supports the validity that the sedimentation rate is used as a parameter in existing source term models for long-range transport simulation of volcanic ash. However, this also suggests that existing source term models based on the one-fluid approximation can cause considerable approximation error for large particles.

## CFD MODELLING OF LIQUID-LIQUID SLUG FLOW IN CAPILLARIES

Julieth Figueroa Rosette, University of Leeds, UK; Alan Burns, University of Leeds, UK

Computational fluid dynamics is used to study the flow behaviour of the liquid-liquid slug flow in a circular capillary. The main flow parameters such as, film thickness, droplet and slug lengths, droplet velocity and pressure drops, are considered during this study. The role of the interfacial forces in the liquid-liquid slug flow is investigated under the limits of high and low viscosity ratios and high surface tension between the two fluids in low Reynolds number pressure driven flow. A train of droplets freely rising or falling under gravity through the capillary is also investigated. The velocity profiles and shear stress profiles across the fluid-fluid interface are calculated for the case of elongated droplets dispersed in a continuous phase in a capillary. The results are compared with experimental data from the literature and are also compared with similar theoretical approaches.

\* Support from grant National Council of Science and Technology (CONACYT) is gratefully acknowledged.

## CFD MODELLING OF DOWNWARD TWO PHASE PIPE FLOW

Eckhard Krepper, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Dirk Lucas, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany; Roland Rzehak, Helmholtz-Zentrum Dresden-Rossendorf (HZDR), Germany

A widely used approach to model two-phase bubbly flows for industrial applications is the Eulerian two-fluid framework of interpenetrating continua. The loss of details caused by the averaging procedure has to be compensated by consideration of additional closure relations. These concern the momentum exchange between the phases, the effect of the bubbles on liquid turbulence and bubble breakup and coalescence. The quest for models with a broad range of applicability allowing predictive simulations is an ongoing venture. A set of best available sub models was assembled and validated against different bubbly flow situations (Rzehak and Krepper 2015).

The present contribution deals with two phase downward pipe flow. Experiments were performed at HZDR and used a fast X-ray tomographic measurement technique. Gas fraction distribution, gas velocities and bubble size distributions were measured at different distances from the fluid injection. Deduced from the experimental data, in some tests the complexity of the closure problem could be reduced imposing a fixed bubble size distribution. Considering the effect of bubble sizes on the closure relations the agreement of the simulations with the measurements could be improved remarkably.

Rzehak, R. and Krepper, E. Bubbly flows with fixed polydispersity: validation of a baseline closure model, Nuclear Engineering and Design, 2015, 287, 108–118

## DNS OF A TURBULENT STEAM/WATER BUBBLY FLOW IN A VERTICAL CHANNEL

Guillaume Bois, CEA Saclay, France; Benoit Mathieu, CEA Saclay, France; Gauthier Fauchet, CEA Saclay, France; Adrien Toutant, Université de Perpignan Via Domitia (UPVD), France

Two-phase turbulence has been studied using a DNS of an upward turbulent bubbly flow in a so-called plane channel. Fully deformable monodispersed bubbles are tracked by the Front-Tracking algorithm implemented in TrioCFD code on the TRUST platform. Realistic fluid properties are used to represent saturated steam and water in pressurized water reactor (PWR) conditions. The large number of bubbles creates a void fraction of 10%. The Reynolds friction number is 180.

After the transitional regime, the flow is simulated over 4 crossing of the channel (of length  $2\pi h$ ). Time- and space-averaging is used to compute the main variables of the average scale description (e.g. void fraction, liquid and vapour velocities...). Budget of forces and Reynolds stresses are also computed from the local fields. They provide reference profiles for future analysis and to improve turbulence modelling and momentum transfer closures.

The velocity profile and the flow-rate are compared to a similar single-phase flow simulation. Strong buoyancy forces create a large relative velocity. Averaged surface tension forces also play a significant role in the flow equilibrium.

\* Support from the NURESAFE project is gratefully acknowledged.

This work was granted access to the HPC resources of TGCC under the allocation 20XX-t20142b7239 made by GENCI

## A MULTI-FLUID MODELLING APPROACH FOR THE AIR ENTRAINMENT OF VENTILATED PARTIAL CAVITY

Min Xiang, National University of Defense Technology, China; Jiyuan Tu, RMIT University, Australia; Longbin Liu, National University of Defense Technology, China

Ventilated cavitation is commonly acknowledged as a prospective drag reduction technology. This paper proposed a numerical modelling framework aiming to couple both Volume-Of-Fluid (VOF) model and Multiple-Size-Group (MUSIG) model to resolve the multiscale flow field generated by ventilated partial cavity. A sub-grid scale air entrainment model was implemented within the Eulerian-Eulerian multi-fluid framework to model the air entrainment at the cavity tail. To evaluate the capability of the proposed model, model predictions were validated against experimental data of Simo et al. at Michigan University. The large-scale cavity profile and the entrainment rate was correctly predicted. The bubbly parameters including the void fraction and the bubble size distributions in the recirculation region and the wake region are compared with the experimental data, demonstrating the potential of the proposed methodology. It is concluded that due to the strong interaction between entrained bubbles and wall surface, great difference was observed for the flow parameters between horizontal cavity and vertical cavity. Discussions on the drawbacks and deficiencies of the current model are also included.

\* Support from National Natural Science Grant (51406230) is gratefully acknowledged.

## SIMULATION OF TWO PHASE FLOW INSIDE AIR CONDITIONING EVAPORATORS

Jan Schlottke, MAHLE Behr GmbH & Co. KG, Germany; Wolfram Kuhnel, MAHLE Behr GmbH & Co. KG, Germany

Inside evaporators of air-conditioning systems, uneven mass flow distribution of refrigerant leads to a loss of efficiency and a reduction of comfort in passenger compartments. The distribution is influenced by the flow field and two phase distribution (flow pattern) in various elements, comprising developing and developed two phase flow in straight and angled ducts, headers, flat tubes and the connection between these.

We present a three-dimensional, transient Euler-Euler approach with modelled size distribution of the disperse phase and special treatment of the free surface. Investigations have been performed on the influence of turbulence modelling, the choice of grid, and the prescription of boundary conditions.

In comparing simulation results of generic headers and real evaporators to experimental data, we find both good agreement as well as discrepancies.

## RHEOLOGICAL CHARACTERIZATION OF SUSPENSIONS OF RIGID FIBERS: THE STRONG EFFECT OF CONFINEMENT

Jason Butler, University of Florida, USA; Braden Snook, University of Florida, USA; Elisabeth Guazzelli, Aix-Marseille University, CNRS, France; Olivier Pouliquen, Aix-Marseille University, CNRS, France

Rheological measurements of concentrated suspensions of rigid, non-Brownian fibers are shown to depend on confinement. To minimize any influence of boundaries, the first and second normal stress differences of the suspensions were calculated from measurements of the deformation of the interface in large-scale free surface flows. The free surface deformation in a tilted trough gives the second normal stress difference and the surface deformation in the rotating rod, or Weissenberg geometry, gives a linear combination of the first and second normal stress differences. Comparisons with results from standard rheological equipment, where the fibers are inevitably confined, demonstrate a clear discrepancy. Stresses calculated from particle-level simulations likewise indicate a dependence on confinement and reveal that bounding walls alter the microstructure more strongly than typically assumed.

\* This work was supported by the National Science Foundation under grants no. 0968313 and 1511787.

## DYNAMICS AND RHEOLOGY OF A DILUTE VISCOELASTIC SUSPENSION OF SPHEROIDS IN AN UNBOUNDED SHEAR FLOW

Gaetano D Avino, Università di Napoli Federico II, Italy; Francesco Greco, Istituto di Ricerche sulla Combustione, IRC-CNR, Italy; Pier Luca Maffettone, Università di Napoli Federico II, Italy

Suspensions of non-spherical particles are encountered in a variety of systems, such as fiber-reinforced polymers, biological fluids, etc.

Shape anisotropy is responsible for peculiar dynamics (as compared to spheres) affecting, in turn, the suspension bulk rheology.

A spheroidal particle in a sheared Newtonian fluid follows the well-known periodic Jeffery orbits. In contrast, a spheroid in a viscoelastic suspending liquid orients towards specific directions, depending on the Deborah number.

In this work, we study through 3D finite element simulations the dynamics and the resulting rheology of a dilute viscoelastic suspension of spheroids in an unbounded shear flow.

For relatively low and high Deborah numbers, alignment along the vorticity and flow direction, respectively, is found. The transition between vorticity and flow orientation occurs through two intermediate regimes. The first one is characterized by small amplitude oscillations around an average orientation between vorticity and flow direction. In the second regime, such an orientation coexists with flow alignment.

The particle contribution to the stress tensor depends on the specific regime of single particle dynamics. Numerical predictions for the viscosity, the first and the second normal stress differences will be presented for different Deborah numbers and spheroid aspect ratios.

## INERTIA-DRIVEN MIGRATION AND MIXING OF PARTICLES IN A SUSPENSION CHANNEL FLOW

Vincent Loisel, LGC-CNRS/University of Toulouse, France; Micheline Abbas, LGC-CNRS/University of Toulouse, France; Olivier Masbernat, Tlamoine, France; Eric Climent, Université de Toulouse, IMFT, CNRS, France

Laminar pressure-driven suspension flows are studied by Force-Coupling Method based on numerical simulations in the case of neutrally buoyant particles at finite Reynolds number. Migration of freely moving particles towards the channel walls due to the Segre-Silberberg effect is observed, leading to the development of a concentration profile, which peaks in the near-wall region and tends towards zero at the channel axis. Particle accumulation in the region of highest shear favours the shear-induced particle interactions and agitation, the profile of which appears to be correlated to the concentration profile. A 1-D model predicting particle agitation, based on the kinetic theory of granular flows in the quenched state regime for Stokes number  $St=O(1)$  and from numerical simulations for  $St<1$ , fails to reproduce the agitation profile in the wall normal direction. Instead, the existence of secondary flows is clearly evidenced by long time simulations. These are composed of a succession of contra-rotating structures, correlated with the development of concentration waves in the transverse direction. The mechanism proposed to explain the onset of this instability is based on the development of a lift force induced by a spanwise gradient of the axial velocity fluctuations.

## FLOW RHEOLOGY OF FIBER-LADEN AQUEOUS FOAMS

Antti Koponen, VTT Technical Research Centre of Finland Ltd, Finland; Pasi Selenius, VTT Technical Research Centre of Finland Ltd, Finland; Ari Jasberg, VTT Technical Research Centre of Finland Ltd, Finland

We have studied fully developed pipe flow of fiber-laden aqueous foams, and decoupled their bulk rheological properties from boundary effects like slippage at pipe wall. The air volume fraction of the foams varied between 60% and 75%. Addition of hardwood fibers at consistency 20 g/kg to plain aqueous foam increased viscosity by ca. 100 %, while with microfibrillated cellulose at consistency 25 g/kg the increase is less than 25%. The effect of synthetic rayon fibers is at consistency 20 g/kg is negligible. All the studied foams can be described as shear-thinning power-law fluid with significant slippage (up to 40% of the mean flow velocity) at the pipe wall. The observed effect of hardwood fibers on the viscosity of foam is much less than in the case of plain aqueous fiber suspension, where the viscosity increases by a factor five or more due to fibers being in continuous contact in shearing. Thus the current results imply that fibers do not interact or flocculate in foams to the same extent as in plain aqueous suspensions. This finding correlates with the high formation of the paper samples produced by using foam forming.

## BROWNIAN DIFFUSION OF NANO-FIBERS

Lin Tian, RMIT University, Australia; Goodarz Ahmadi, Clarkson University, USA; Jiyuan Tu, RMIT University, Australia

Motions of nano-particles suspended in a fluid are dominated by Brownian diffusion. The process of Brownian motion has long been well understood following Einstein (1905), and the theoretical derivation of the diffusion coefficients for spherical particles has been well developed. For non-spherical particles where there are coupling between translational and rotation motions, a clear description of the overall Brownian diffusion process is not available. In this study, the Brownian diffusion of elongated nano-fibers is investigated numerically. Motion of the nano-fiber is resolved by solving the system of equations governing the fiber coupled translational and rotational motions. Test fibers are immersed in an unbounded quiescent fluid, where only the hydrodynamic and Brownian forces are present. The study allows examination of the Brownian diffusion of non-spherical particles. Empirical equations are developed to quantify the diffusion coefficients of nano-fibers. The results are compared with the theoretical results and experimental data for the limiting case of spheres, where excellent agreement is found. The study opens up a new approach for studying the diffusive processes of nano-particles of complex shapes which are abundant in natural environment.

## SETTLING VELOCITY AND MICROSTRUCTURE OF SOLID PARTICLES IN SUSPENSIONS SIMULATIONS WITH LATTICE-BOLTZMANN METHOD

Yi Huang, Huazhong University of Science and Technology, China; Jing Li, Huazhong University of Science and Technology, China; Lu Zha, Huazhong University of Science and Technology, China; Zhaohui Liu, Xi'an Jiaotong University, China; Chuang Zheng, Huazhong University of Science and Technology, China

Lattice-Boltzmann method with discrete external boundary force is employed to simulate motions of solid spheres settling in suspensions, which determines the mean settling velocity and microstructure in a sedimentation system. In this paper, we investigate different spherical particles settlement systems with solid volume fractions varied from 0.92% to 8.59% at Reynolds number (based on the terminal velocity) about 100. During the sedimentation process, three different regimes are identified: acceleration, deceleration and stability, characterized by the ensemble average vertical velocity. In the stability regime, the velocity distribution of particles is similar to the Gaussian distribution. The hindered settling velocity is relevant to solid volume fraction, and the ratio of the mean settling velocity to the terminal velocity can be fit by a power-law expression, which is compared to equations in literatures. We discover the velocity fluctuation which is not apparent in horizontal direction but strong and anisotropic in vertical direction increases with increasing solid volume fraction, and is affected by the size of simulation domain. As for the microstructure of particles in a suspension, it presents intensely anisotropic and causes particles to cluster easily, which results a non-uniform flow.

## THE MOTION OF A CLOUD OF SOLID SPHERICAL PARTICLES FALLING IN A CELLULAR FLOW FIELD AT LOW STOKES NUMBER

Benjamin Marchetti, Aix-Marseille University, CNRS, France; Laurence Bergougnoux, Aix-Marseille University, CNRS, France; Gilles Bouchet, Aix-Marseille University, CNRS, France; Elisabeth Guazzelli, Aix-Marseille University, CNRS, France

We present a joint experimental and numerical study examining the influence of vortical structures on the settling of a cloud of solid spherical particles under the action of gravity at low Stokes numbers. The two-dimensional model experiment uses electro-convection to generate a two-dimensional array of controlled vortices which mimics a simplified vortical flow. Particle image-velocimetry and tracking are used to examine the motion of the cloud within this vortical flow. The cloud motion is compared to the predictions of a two-way-coupling numerical simulation.

\* This work was undertaken under the auspices of ANR project "Collective Dynamics of Settling Particles In Turbulence" (ANR-12-BS09-0017-01), "Laboratoire d'Excellence Mécanique et Complexité" (ANR-11-LABX-0092), and "Initiative d'Excellence" A\*MIDEX (ANR-11-IDEX-0001-02).

## FIDELITY REQUIREMENTS FOR THE LARGE EDDY SIMULATION OF DIESEL SPRAY

Francois Doisneau, Sandia National Laboratories, USA; Marco Arienti, Sandia National Laboratories, USA; Joseph Oefelein, Sandia National Laboratories, USA

In Diesel engines, ignition takes place after a complex process involving atomization, droplet transport and evaporation, mixing, and low temperature chemistry. We investigate the fidelity requirements on describing these processes by using state-of-the-art methods for the Large Eddy Simulation of primary atomization, dense spray, and chemistry. The simulation results are discussed for their ability to reproduce experimental data with minimum tuning as well as to account for some level of variability of the configuration. The objective is to assess the relevance of an approach that is affordable compared to the current DNS with sharp-interface capturing. The qualification of a massively parallel LES solver for performing predictive industrial simulations of sub-critical injection and combustion is a long term goal and would provide an important tool for the design of advanced engines.

\*Support by Sandia National Laboratories' Laboratory Directed Research and Development (LDRD) program is gratefully acknowledged.

## THERMOCAPILLARY DROPLET ACTUATION ON A WALL

Anja Fath, TU Darmstadt, Germany; Dieter Bothe, Technische Universität Darmstadt, Germany

Droplet actuation is attracting widespread attention because of its promising potential in droplet-based devices developed for various applications in industry. One way to actuate a droplet is to apply a thermal gradient along the solid wall. This phenomenon is to be analyzed numerically in the Volume of Fluid (VOF) framework. The two main physical effects to be captured are the thermal Marangoni effect and the contact line dynamics. The so-called apparent contact angle in the present study is given by the empirical model by Kistler. This is the angle between the fluid/fluid-interface at a certain distance and the solid wall and can be measured in experiments or given by asymptotic theory for small capillary numbers. The apparent contact angle, which is calculated from the velocity in the vicinity of the contact line, is then transformed into a numerical contact angle, building on the method developed by Afkhami, Zaleski and Bussman (2009). This numerical contact angle accounts for the resolution effects and non-physical large slip length. First simulations of thermal droplet actuation with a squalane droplet in a fully three-dimensional setting show the potential of the developed algorithm. Our first aim is to determine the relation between droplet deformation, migration velocity and acting forces depending on the temperature gradient and fluid parameters. The study focuses on larger angles, where lubrication theory is losing its validity.

## TUMBLING DYNAMICS OF NON-SPHERICAL PARTICLES FALLING IN A TURBULENT FLOW

Kristian Gustavsson, University of Gothenburg, Sweden; Bernhard Mehlig, University of Gothenburg, Sweden

To understand the dynamics of small particles suspended in turbulent flows is important to shed light on many problems in Natural Sciences and in technology. We consider the dynamics of small, heavy, spheroidal particles falling in a turbulent aerosol. The flow is modelled using a statistical velocity field which is smooth in both space and time. In this model we can solve the dynamics analytically, allowing us to find the functional dependence of statistical quantities on the relevant particle parameters. Rotational symmetry in the problem is broken due to gravity and due to the non-spherical shape of the particles. We study how this affects the settling velocity and tumbling of particles. We also study how the particle shape affects small-scale fractal clustering and large-scale preferential sampling of inertial particles.

## SIMULATION OF EVAPORATING DROPLETS WITHIN A DISCONTINUOUS GALERKIN MULTI-SCALE FRAMEWORK

Timon Hitz, Stuttgart University, Germany; Stefan Fechter, Stuttgart University, Germany; Claus-Dieter Munz, University of Stuttgart, Germany

In the present study, a compressible multi-scale flow solver is described, based on the discontinuous Galerkin (DG) spectral element method. The numerical method consists of the following building blocks. The phase interface is tracked via a level-set method embedded in the DG method which allows an accurate curvature calculation. The interface is resolved sharply using a ghost fluid approach. The thermodynamic consistent interface jump conditions for the state variables are provided by a newly developed approximative interface Riemann solver. An additional estimate for the mass flux is required such that the thermodynamic laws are fulfilled and a unique solution is obtained. Here the evaporation model by Schrage is used. We apply the DG method for the resolution of the bulk phases. To increase the interface resolution and robustness we switch locally at the interface to a sub-cell finite volume method inside the high order DG cell. We show results for the extension towards multicomponent problems using the Peng-Robinson EOS with Van-der-Waals mixing rules. The solver is validated for the case of a single component droplet evaporating in an inert carrier gas.

\* Support from the German Research Foundation (DFG) through SFB TRR 75 and SFB TR 40 is gratefully acknowledged.

## IMPLEMENTATION OF THE IMPLICIT-CONTINUOUS-EULERIAN SCHEME FOR MULTIPHASE FLOWS IN THE PARALLEL DUNE FRAMEWORK

Christophe Fochesato, Commissariat à l'Energie Atomique, France; Magali Zabiego, CEA, France

In the frame of the safety analysis of the future ASTRID Fast Neutron Reactor, CEA has started the development of the SCONE software for the corium-sodium interaction assessment. The Implicit-Continuous-Eulerian method has been extensively used in the nuclear safety community as it turned out to be a robust technique applicable for multiphase flows at all flow-speed regimes. It relies on two main ingredients that are the time solving procedure and the specific degrees of freedom. The time discretization is a semi-implicit method in which pressure terms are implicit, as well as all exchange terms depending on velocities whereas advection terms are explicit. The space discretization is based on different locations for the degrees of freedom since the pressure and all other thermodynamical quantities are cell-centered while normal velocities are known at faces. It typically combines an upwind finite volume method for mass conservation and for evolution of the internal energy whereas the momentum equation is discretized by a finite difference scheme for each velocity components on a rectangular mesh. In this study, the implementation of the ICE scheme in the DUNE framework, an open source software for solving PDEs with grid-based methods on parallel architectures, is presented and some comparison are performed between ICE and other schemes yet existing in the platform. In particular, intending to be used in the context of steam explosion, lack of conservation leading to wrong shock propagation is investigated. Moreover the performance of the parallel software is tested.

## EFFECT OF FRICTIONAL PACKING LIMIT ON NUMERICAL SIMULATION OF HYDRODYNAMICS OF DENSE GAS-SOLID FLOWS

Akhilesh Kumar Sahu, Indian Institute of Technology Madras, India; Raghavan Vasudevan, Indian Institute of Technology Madras, India; Prasad Bhamidi, Indian Institute of Technology Madras, India

Numerical simulations to determine the hydrodynamics of dense gas-solid flows using Eulerian-Eulerian methodology are reported. In dense gas-solid flows, particle-particle friction becomes appreciable and should be accounted in the numerical model. Frictional packing limit (FPL) is the minimum value of solids volume fraction at which inter-particle friction forces becomes significant. A proper value should be chosen for this. To establish the range of FPL values, simulations are carried out within the bubbling fluidized bed regime. Bed pressure drop values predicted by the numerical model with different FPL values, for various superficial velocities, are validated against the experimental data from literature. It is observed that the predictions become better when FPL lies in the range of 0.61-0.63 and the bed pressure drop is under predicted for FPL values between 0.50 and 0.55. At higher superficial gas velocities, when the bed tends to become dilute, predicted pressure drop is unaffected by FPL. Further, gas-solid flow characteristics such as solids velocity, voidage profile and bubble patterns are observed to be affected by the choice of FPL value.

\*Authors acknowledge the financial support received from European Union project OPTIMASH FP7-ENERGY, 2011-15.

## FLUID PERMEABILITY THROUGH FIBROUS BED WITH TORTUOUS STRUCTURE

Ryoko Otomo, Kansai University, Japan; Mori Kazuki, Kansai University, Japan; Sakai Sho, Kansai University, Japan

Fluid permeation through fibrous materials has been investigated by theoretical calculations. In a number of previous researches, the fibrous bed was modeled by the assembly of circular cylinders. The fluid permeability was evaluated in terms of the direction of the straight fibers to the flow while the effect of the fiber tortuosity on the permeability received less attention. We employ a particle model in which a fiber consists of spherical particles arranged in line instead of a cylinder. A certain angle (tortuous angle) is formed at equal spaces in a particle line corresponding to a fiber. The tortuous fibrous bed is represented by arranging randomly such bended particle lines. On the assumption of low Reynolds number, the permeability has been calculated based on the Stokesian dynamics method. The results show that the increase of the tortuous angle from 0 deg. (straight fiber condition) to 45 deg. causes the decrease of the permeability. The difference in the permeability between the cases of 0 deg. and 45 deg. reaches more than 10% under the same particle volume fraction. In order to examine intensively the reason for the dependence of the permeability on the tortuous angle, we have defined pore radius and evaluated quantitatively the pore distribution in the particulate bed with different tortuosities. It is found that the pore connectedness in the flow direction becomes higher with increase in the tortuous angle and it would affect the permeability.

\* Support from JSPS KAKENHI Grant Number 26820050 is gratefully acknowledged.

## MEASUREMENT OF LIQUID FILM THICKNESS IN TWO PHASE FLOW OF HFC134A GAS-ETHANOL LIQUID IN A VERTICAL TUBE

Shoji Mori, Yokohama National University, Japan; Yuta Tanaka, Yokohama National University, Japan; Kunito Okuyama, Yokohama National University, Japan

Gas-liquid two-phase flow is encountered in many industries, including flow near nuclear fuel rods in boiling water reactor (BWR). Especially dryout of water film occurs at the thin base film between two successive disturbance waves. Therefore, it is important to clarify the detailed behavior of the disturbance wave. In previous studies, many experiments were performed under atmospheric condition. However the properties of liquid and gas under atmospheric pressure condition are quite different from those of a BWR operating condition (7MPa, 285C). Therefore, in the present study, HFC134a gas and ethanol liquid, whose properties are similar to those of the BWR operating conditions, are used as the working fluids at comparatively low pressure and low temperature (0.7MPa, 40C). In this paper, the liquid film thickness characteristic in HFC134a gas/ethanol liquid two phase flow is reported.

## THE DENSE DISCRETE PHASE MODEL FOR SIMULATION OF BUBBLING FLUIDIZED BEDS: VALIDATION AND VERIFICATION

Schalk Cloete, SINTEF Materials and Chemistry, Norway; Shahriar Amini, SINTEF Materials and Chemistry, Norway

The dense discrete phase model (DDPM) is a Lagrangian parcel-based approach which models particle collisions and uncorrelated translations using the kinetic theory of granular flows (KTGF). This approach has numerous advantages over the established Eulerian two fluid model (TFM) such as better resolution of particle clusters and bubbles, more natural incorporation of particle size distributions and better handling of crossing particle jets/clusters. In this study, results from the DDPM are compared to dedicated experiments and TFM simulations over a wide range of fluidization velocities, particle sizes and bed loadings. Special attention was given to ensure that the frictional effects typical of such dense particle flows are accounted for. The DDPM proved capable of reproducing the good fit to experiments achieved by the TFM and achieve grid independent results on significantly coarser meshes. The overall speedup of the DDPM relative to the TFM is also quantified.

\* The authors would like to acknowledge the financial support of the Research Council of Norway under the FRINATEK grant (Acronym: CSR, Project number: 221902) which made this work possible.

## MACROSCOPIC MODELLING OF PARTICLE FLUCTUANT KINETIC ENERGY IN A LIQUID FLUIDIZED BED

Ali Ozel, Princeton University, USA; Jorge Cesar Brandt De Motta, CORIA, France; Olivier Masbernat, Tiemoine, France; Olivier Simonin, INP Toulouse, France

Particle and fluid agitation in a liquid fluidized bed has been investigated both experimentally and numerically. Analysis of the particle fluctuant kinetic energy is performed with a macroscopic model based on a kinetic description of the solid phase with simplified modelling approaches for the fluid-particle momentum transfer and for inter-particle short-range interactions. The kinetic description of the solid phase results in macroscopic balance equations for the mean particle vertical velocity, the particle velocity variance and the fluid-particle velocity covariance. For collision modelling, the particle velocity variance is split in two contributions: a correlated part, due to the interaction with the larger scale fluid motion and a random uncorrelated part (the granular temperature) which results from the local relative particle-to-particle motion. Predictions of the fluid-particle velocity covariance and particle velocity variance have been carried out with this model and compared with the particle resolved simulation results and available experimental data. It is shown that the proposed model is able to predict the balance of the fluid kinetic energy in the fluidized bed, showing the effect of the production by the mean drag, viscous dissipation and transfer terms.

## DIRECT NUMERICAL SIMULATIONS OF TURBULENT BOUNDARY LAYER OVER HEMISPHERICAL ROUGH WALL

Xiaofei Liu, Zhejiang University, China; Hui Zhao, Zhejiang University, China; Anyang Wei, Zhejiang University, China; Kun Luo, Zhejiang University, China; Jianren Fan, Zhejiang University, China

Direct numerical simulations (DNSs) were performed to investigate the effects of hemispherical roughness elements on the properties of the spatially developing turbulent boundary layer. The hemispheres were staggered in the downstream direction and periodically arranged in the streamwise and spanwise directions with pitches of  $p_x/d=2, 4, 8$  and  $p_z/d=2$ , where  $p_x$  and  $p_z$  are the streamwise and spanwise spacings of the hemispheres, and  $d$  is the diameter of the hemisphere. The inlet Reynolds number based on the momentum thickness was set to be  $Re_{\Theta in}=800$  and the height of the hemisphere was  $k=1.5d=1.5\Theta in$ , where  $\Theta in$  is the momentum thickness at the inlet. The effect of different spacings on the turbulent statistics and coherent structure were examined. With the reduction of spacing, the vortex structures originating from head and wake of hemispheres gradually become the dominant source in the boundary layer. Their effects extend from the near wall region into the entire width of the boundary layer indicating that the outer-layer similarity was not established for current conditions. In addition, we also found that the ejected- and swept- events do not relate to the spacing, but rely on the different generated forms of vortex structures behind roughness elements.

\* Support from the National Natural Science Foundation of China (No. 51136006) is gratefully acknowledged.

## LIQUID-GAS DYNAMIC SYSTEM UNDER VIBRATION

Timothy O Hern, Sandia National Laboratories, USA; John Torczynski, Sandia National Laboratories, USA; Jonathan Clausen, Sandia National Laboratories, USA

Adding a little gas can completely change the dynamics of a spring-mass-damper system subject to vibration. Experiments and modeling have been performed on a spring-supported piston in a liquid-filled cylinder, where the gaps between piston, cylinder, and shaft through the cylinder are narrow and depend on the piston position. When gas is absent, the piston's vibrational response is highly overdamped due to viscous liquid being forced through narrow gaps. When a small amount of gas is added, Bjerknes forces cause some of the gas to migrate below the piston. The resulting gas regions above and below the piston form a pneumatic spring that enables the liquid to move with the piston, with the result that very little liquid is forced through the narrow gaps. This "Couette mode" has low damping and thus has a strong resonance near the frequency given by the pneumatic spring constant and the piston mass. At this frequency, the piston response is large, and the nonlinearity from the gap geometry produces a net force on the piston. This "rectified" force can be many times the piston's weight and can cause the piston to compress its supporting spring. Experiments have been performed with liquid-only systems, with bubbles present, and with flexible bellows that act as non-mobile bubbles with bubble-like compressibility. \*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.

## STEADY FILM FLOW OVER 2D TOPOGRAPHY WITH AIR INCLUSION

Styllanos Varchanis, University of Patras, Greece; Dionisios Pettas, University of Patras, Greece; Yiannis Dimakopoulos, University Of Patras, Greece; John Tsamopoulos, University Of Patras, Greece

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, are revealed by pseudo arc-length continuation. All possible cases of a single air inclusion inside the trench along with special cases are examined. The penetration of the liquid inside the trench is enhanced primarily by increasing the wettability of the substrate, or by decreasing either the liquid viscosity or the flow rate. Flow hysteresis may occur when the liquid does not penetrate deep enough inside the trench allowing the two liquid-air interfaces to interact. 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.

## THE INFLUENCE OF FLUID VISCOSITY ON SIZE-BASED PARTICLE SEPARATION IN THE MICROCHANNEL WITH SYMMETRIC SHARP CORNER STRUCTURES

Liang-Liang Fan, Xi'an Jiaotong University, China; Xukun He, Xi'an Jiaotong University, China; Xiao-Juan Niu, Xi'an Jiaotong University, China; Jiang Zhe, University of Akron, USA; Liang Zhao, Xi'an Jiaotong University, China; Qing Yan, Xi'an Jiaotong University, China

Size-based particle separation has great potential in biomedical applications. We propose a new microchannel with symmetric sharp corners for size-based particle separation. In the present microchannel, particles suspended in DI water are mainly subjected to the momentum-change-induced inertial force ( $F_c$ ) and the inertial lift force ( $F_i$ ).  $F_c$  is dominant for big particles while  $F_i$  is dominant for small ones. Big particles driven by  $F_c$  migrate towards the center of the channel while small ones driven by  $F_i$  migrate towards the sides, achieving complete particle separation. In biomedical applications, the fluid, such as the blood, has higher viscosity than DI water. To further investigate the influence of fluid viscosity on particle separation, experiment with high viscosity fluid is conducted in the microchannel. The results show that particle separation efficiency decreases from 100% to 71.5% due to the enhanced viscous drag effect in the high-viscosity fluid. Hence, the fluid viscosity has great influence on the passive particle separation. Viscous drag effect should be considered in the design of the particle separation microchannel. The study is helpful for the efficient particle separation in the microchannel with high viscosity fluid.

\* Support from the National Natural Science Foundation of China (Nos. 51476127 and 51128601) is gratefully acknowledged.

## THE SWIRLING ANNULAR TWO-PHASE FLOW INSIDE A CIRCULAR PIPE - INTERFACIAL PHENOMENA, FLOW MECHANISM AND MODELLING

Bofeng Bai, Xi'an Jiaotong University, China; Wen Liu, State Key Laboratory of Multiphase Flow in Power Engineering, China; Li Liu, State Key Laboratory of Multiphase Flow in Power Engineering, China

Flow pattern modulation has been successfully applied in multiple industries. The formation of swirling annular flow is a successful modulation. The interfacial phenomena and flow mechanism of swirling annular flow in a vertical pipe are studied, employing both visualization experiments and theoretical models. We observe that a swirling annular flow both occurs at low and high void fraction. In the case of bubble flow with low void fraction, as the swirl decays along the streamwise direction, the liquid surrounding the gas column bridges, leading to the swirling annular flow transformed to slug flow. The formation of the swirling annular flow is studied by a single bubble trajectory model. To quantitatively describe the decay in the swirling annular flow, a simplified model based on the momentum theorem is established. In the case of annular flow with high void fraction, the linear temporal instability analysis is performed to investigate the stability of swirling annular flow, taking exactly the centrifugal force and curvature of the cylinder into account. Swirl has been considered by superposing a potential vortex to the traditional annular flow. Based on the two-fluid model, the dispersion equation is derived and the necessary condition for interfacial stability to three-dimensional disturbances is obtained. The gas-liquid swirl motion has complicated dual effects on the interfacial stability based on the relative magnitude of dynamic pressure of gas and liquid phases.

\* Support from National Natural Science Foundation of China under the contract No. 51276140 is appreciated.

## ON THE EFFECTS OF INERTIA IN TAYLOR FLOW IN RECTANGULAR MILLI-CHANNELS OF MODERATE ASPECT RATIO

Dominique Legendre, Institut de Mécanique des Fluides de Toulouse, France; Thomas Abadie, Institut de Mécanique des Fluides de Toulouse (IMFT), France; Joelle Aubin, Laboratoire de Génie Chimique (LGC), France

This study focuses on the hydrodynamics of gas-liquid Taylor flow (or slug flow) in milli-channels with square and rectangular cross-sections. Indeed, although Taylor flows in visco-capillary regimes in circular tubes have been well characterized since the early studies of Taylor (1961) and Bretherton (1961) or Aussilous and Quéré (2000) more recently, less attention has been paid to square and rectangular cross-sections even though these geometries appear interesting when dealing with transport phenomena in multiphase reactors for instance. In addition, the effects of inertia in micro or milli-channels are often neglected whereas the Laplace number can be of the order of several thousands in millimetric channels so that for a typical capillary number of  $Ca \sim 0.01$ , the Reynolds number can reach several tens to several hundreds. The flow and the bubbles are therefore subject to inertial effects that cannot be neglected. 3D numerical simulations of bubbles trains have been performed in rectangular geometries of moderate aspect ratios (1, 2.5 and 4) and wide ranges of capillary and Reynolds numbers have been covered ( $0.002 < Ca < 0.4$  and  $0.5 < Re < 1000$ ). While the bubble velocity is not strongly affected by inertial effects, the shape of the bubble changes and therefore the pressure drop increases significantly. In addition, at high Laplace numbers, different regimes where the bubbles are oscillating or the bubbles break are observed.

\* Support from the French "Agence Nationale de la Recherche", grant ANR-09-BLAN-0381-01 (MIGALI) is gratefully acknowledged.

## DYNAMICS OF NEUTRALLY BUOYANT SPHERES IN OBLIQUE MICRO-BIFURCATION CHANNELS

Bhaskar Medhi, Indian Institute of Technology, India; Vipin Agrawal, Indian Institute of Technology, India; Anugrah Singh, Indian Institute of Technology, India

Knowledge of flow as well as particle distribution in bifurcating channels is very important either for the separation or concentration of various types of particles. Flow of neutrally buoyant suspension in oblique bifurcating channels corresponding to various bifurcation angles were investigated experimentally. A combination of  $\mu$ -PIV and  $\mu$ -PTV studies was conducted to measure the fluid and particle velocities during transport of suspension in bifurcation channels. A MATLAB program was used to count the dispersed particles and compute the concentration profile in the micro channels. The experimental studies revealed that suspended solid particles undergo particle migration which plays an important role in deciding the actual position of flow bifurcation near the junction along with the velocity and concentration profile in the daughter branches. From the quantitative study of velocity and concentration profile we observed that after the bifurcation the peak of velocity profile shifts toward the outer wall (higher shear rate region) whereas concentration profile shifts towards the inner wall (lower shear rate region) for all the micro channels. As we move downstream locations in the daughter branches the velocity and concentration profile again shifts toward the center of the channel. We have observed from the position of dividing stream line that disperse solid particles do not follow the fluid path near the bifurcation which leads to unequal flow and particle partitioning in daughter branches.



# YIELDING THE YIELD-STRESS ANALYSIS: A STUDY FOCUSED ON THE EFFECTS OF ELASTICITY ON THE SETTLING OF A SINGLE PARTICLE

Dimitrios Fraggedakis, University Of Patras, Greece; Yiannis Dimakopoulos, University Of Patras, Greece; John Tsamopoulos, University Of Patras, Greece

For several decades, Carbopol has been assumed to be the ideal viscoplastic material, exhibiting only yield phenomena without viscoelastic effects in yielded regions. Recently, experiments (Putz et al. (2006)) reveal phenomena, which can be attributed only to elastic properties of the fluidized region, such as the appearance of the so-called "negative wake" downstream a falling sphere and the loss of fore-aft symmetry of the yield surface around a sedimenting particle. To this end, we use the constitutive model for elastoviscoplastic fluids proposed by Saramito (2007) and extract the property values from the rheological characterization of Carbopol by Holenberget al. (2012) via the LAOS method. Our study is based on the axisymmetric sedimentation of a particle in materials which exhibit elastoviscoplastic behavior and indicates that Carbopol cannot be considered as the ideal plastic material any more. We examine the settling in a cylinder, and how the wall position in relation with the sphere affects the yield surface position. Moreover, when elasticity comes into play, the derived stoppage criterion for a sedimenting sphere by Beris et al. (1985) is not satisfied, as a complex stress field is developed around the particle and fluidization near the rigid surface is favored.

# REMOVING DRILLING MUD IN CEMENTING OPERATIONS: A FAMILY OF CHALLENGING DISPLACEMENT FLOWS

Ian Frigaard, University of British Columbia, Canada; Ali Roustaei, UBC, Canada; Jaewoo Jeon, UBC, Canada; Marjan Zare, University of British Columbia, Canada; Mohammad Taghavi, University of Laval, Canada; Alba Kamran, University of British Columbia, Canada

Oil and gas wells undergo cementing operations during construction, remediation and eventually abandonment. From a fluid-dynamical perspective the in situ fluid must be replaced by another fluid. In general both fluids are fine colloidal suspensions, having the dominant rheological characteristics of yield stress, shear thinning fluids, e.g. drilling muds, spacer fluids and cement slurries. In order to achieve a good hydraulic seal of the well, drilling fluid must be removed from the walls and cement contamination be minimized. We present flow results related to these processes, including visco-plastic displacement flows in pipes [1], eccentric annuli [2], channels [3] and washouts [4]. A key question concerns the identification of stationary regions of the displaced fluid. This is done for single and multi-fluid flows. Analytical and computational results are supplemented by experiment results conducted in a pipe loop that show surprising complexity.

- [1] K. Alba, S.M. Taghavi, J. de Bruyn, I.A. Frigaard, 201, pp. 80-93, (2013).
- [2] S. Pelipenko, I.A.Frigaard, J. Fluid Mech. 520, pp. 343-377, (2004). M. Carrasco-Teja, I. Frigaard\*, B. Seymour, S. Storey, J. Fluid Mech. 605, pp. 293-327 (2008).
- [3] M. Allouche, I.A. Frigaard, G. Sona, J. Fluid Mech. 424, pp. 243-277, (2000).
- [4] A. Roustaei, A. Gosselin, I.A. Frigaard, J. non-Newt. Fluid Mech., doi:10.1016/j.jnnfm.2014.09.019. (2014).

# DIRECT NUMERICAL SIMULATION OF SUPERSONIC TURBULENT SPRAY JET FLAME IN HEATED COFLOW

Tai Jin, Zhejiang University, China; Kun Luo, Zhejiang University, China; Jianren Fan, Zhejiang University, China

Direct numerical simulations of three-dimensional spatially-developing supersonic turbulent spray reacting flows in subsonic heated coflow have been conducted. The gas-droplet flow system has been captured with the hybrid Eulerian-Lagrangian approach. High-resolution bandwidth-optimized weighted essentially non-oscillatory (WENO) scheme of spatial discretization and total variation diminishing Runge-Kutta temporal integration are used to capture the compressible turbulent flow. Arrhenius-type finite-rate chemistry is employed for the chemical reaction. Compared with the non-reacting case, the decay of the mean axial velocity at the centerline has been found much slower for the reacting case. The half-width of the spreading reacting jet is larger than the non-reacting case. The velocity of droplet has been found to be larger compared with the droplet at the same position in non-reacting flow. Reynolds stress has been found to be slightly larger in the mixing layer with intense heat release. The fluctuation of temperature, mixture fraction has been found to be amplified significantly in the reacting flow. The spray jet flame has been found to be lifted. Premixed and diffusion modes are found to coexist. Heat release contributed by premixed flame to the total heat release occupies more than 50% and peaks at 71.4% downstream of the flame.

\* Support from grant NSFC(No.51222602) is gratefully acknowledged.

# SIMULATION OF DROPLET JETTING OF A NON-NEWTONIAN MIXED SUSPENSION

Martin Svensson, Fraunhofer-Chalmers Research Centre, Sweden; Andreas Mark, Fraunhofer-Chalmers Research Centre, Sweden; Gustaf Martensson, Chalmers University of Technology, Sweden; Andrzej Karawajczyk, Mycronic AB, Sweden; Fredrik Edelvik, Fraunhofer-Chalmers Research Centre, Sweden

The jet printing of a dense mixed non-Newtonian suspension is based on the rapid displacement of fluid through a nozzle, the forming of a droplet and eventually the break-off of the filament. The ability to model this process would facilitate the development of future jetting devices. The purpose of this study is to propose a novel simulation framework and to show that it captures the main effects such as droplet shape, volume and speed. In the framework, the time dependent flow and the fluid-structure interaction between the suspension, the moving piston and the deflection of the jetting head is simulated. The system is modelled as a two phase system with the surrounding air being one phase and the dense suspension the other. Hence, the non-Newtonian suspension is modelled as a mixed single phase with properties determined from material testing. The simulations were performed with two coupled in-house solvers developed at Fraunhofer-Chalmers Centre: IBOFlow, a multiphase flow solver and LaStFEM, a large strain FEM solver. Jetting behaviour was shown to be affected not only by piston motion and fluid rheology, but also by the energy loss in the jetting head. The simulation results were compared to experimental data obtained from an industrial jetting head.

\* Support from the Swedish Research Council (contract 2010-4334) is gratefully acknowledged.

# DEVELOPMENT OF A NEW OPENFOAM VOF-BASED SOLVER COUPLED WITH POPULATION BALANCES FOR THE SIMULATION OF EXPANDING POLYURETHANE FOAMS

Mohsen Karimi, Chemical Engineering, Politecnico di Torino, Italy; Daniele Marchisio, Chemical Engineering, Politecnico di Torino, Italy

This study reports on the development of a multiphase solver for modelling and simulation of expanding polyurethane foams (PU) for mold filling applications. The three-phase system, including the liquid polymer, the gas bubbles forming during the expansion, and the surrounding air is modelled as a pseudo-fluid having a common interface with the air. The interface is captured using a Volume-of-Fluid (VOF) based compressible solver available in open source CFD code OpenFOAM. The solver is, however, supplemented with a PBE to identify the evolution of gas bubbles (or cells) size distribution (BSD) throughout the polymerization process owing to the growth and coalescence of bubbles. The Quadrature Method of Moments (QMOM) is applied to solve the PBE, whereas optional choices for determining the foam rheology (i.e., Newtonian or non-Newtonian) are embedded in the solver. Typical cup-experiment test cases for the PU foam shows that the developed solver, "QmomKinetics", is able to describe the fundamental physical phenomena of the foaming process including the density and viscosity variations, kinetics of the reactions and the evolution of BSD.

\* Support from European Commission under the grant agreement 604271 (Project acronym: MoDeNa; call identifier: FP7-NMP-2013-SMALL-7) is acknowledged.

# DIRECT NUMERICAL SIMULATION OF N-HEPTANE DROPLETS IGNITING IN A TURBULENT FLOW

Abouelmagd Abdelsami, University of Magdeburg, Germany; Dominique Thevenin, University of Magdeburg "Otto von Guericke", Germany

The main objective of this work is to investigate the impact of turbulence statistics and shear flow on droplets ignition and evaporation in a temporally-evolving turbulent jet. This investigation has been performed by means of 3D Direct Numerical Simulation (DNS) using the high order in-house code DINOSOARS. In this work, the flow governing equations with a low-Mach number formulation have been discretized on a Euler frame while the droplets are tracked using a Lagrangian point-force approach. The momentum, mass and heat transfer are considered by adopting an infinite-liquid-conductivity model (Abramzon & Sirignano, 1988). In the current simulation, droplets with various initial specific diameters and temperatures are distributed randomly in a central slab of width H. The whole domain contains a hot oxidizer (air) with initial uniform gas temperature. The gas surrounding the droplets inside the central slab is moving with jet velocity  $U_j$ , whereas the surrounding co-flow moves with velocity  $U_{co}$ . The turbulence signals are filtered to be imposed only in the middle of the domain. First results show that the selected configuration is very promising to investigate shear flow/droplet interaction, revealing new features considering mixing, evaporation and ignition. The complete results will be discussed at the conference.

#### A CFD-VOF BASED MODEL TO ADDRESS INTENSIVE PHOTOBIOREACTOR DESIGN

Giuseppe Olivieri, Università degli Studi di Napoli Federico II, Italy; Piero Salatino, Università degli Studi di Napoli Federico II, Italy; Antonio Marzocchella, Università degli Studi di Napoli Federico II, Italy; Luca Mazzei, University College London, UK; Paola Lettieri, University College of Londo, UK

The design and optimization of photobioreactors for intensive microalgal cultures are key issues to increase process performance. A model to assess the photosynthetic performance of tubular, bubble column and flat photobioreactors is presented. The model has coupled microalgal light distribution, photosynthesis kinetics and gas-liquid hydrodynamics. A lumped kinetic parameter model of photosynthetic unit (PSU) has been adopted for photosynthetic reactions. The dynamics of a microalgal cell has been described according to the gas-liquid flow of a bubble column. The flow field induced by liquid turbulence and bubbles uprising throughout the photobioreactor have been simulated with ANSYS-FLUENT. A representative domain of the flat photobioreactor has been selected by adopting proper periodic boundary conditions. Turbulence dispersion fields have been assessed by numerical simulations for several bubble size. A random-walk model has been adopted to microalgal cells to assess the irradiance experienced by the PSU-cell in the photobioreactors. The photobioreactor performances - expressed in terms of global photosynthesis rate - have been assessed. Irradiance level and biomass concentration have been changed in the range of operating conditions typically adopted for known processes.

#### A COMBINED LES-PBE-PDF METHOD FOR MODELLING POLYDISPERSED PARTICLE FORMATION IN TURBULENT REACTING FLOWS

Fabian Sewerin, Imperial College London, UK; Stelios Rigopoulos, Imperial College London, UK

We present a comprehensive model and solution technique for predicting the evolution of a polydispersed particulate phase that is immersed in a turbulent reacting flow. Here, the particulate phase is described in terms of its number density distribution which evolves both in physical and particle property space according to the population balance equation (PBE). For the discretization of the PBE in particle property space, we develop a novel technique based upon a solution-adaptive and time-dependent coordinate transformation. This scheme dynamically redistributes resolution in particle property space and is able to capture peaks and near-discontinuities.

Upon discretization the PBE reduces to a collection of reactive scalar transport equations. Likewise, the fluid phase is described by distinct reactive scalars. For the turbulent carrier flow, we adopt a standard LES approach. The turbulence-chemistry interaction, moreover, is closed by considering a modelled transport equation for the joint scalar probability density function (pdf) of a single realization of the flow field. This equation is solved by the method of Eulerian stochastic fields.

As an application, we consider the precipitation of BaSO<sub>4</sub> particles from an aqueous solution in a coaxial pipe mixer. In this context, the main advantages of our approach are highlighted: On the one hand, both fluid and particle phase kinetics are accommodated without approximation. On the other hand, the entire particle property distribution is predicted at each location in the flow domain.

#### DNS OF CO-CURRENT TURBULENT STRATIFIED TWO-PHASE PIPE FLOW

Wim-Paul Breugem, Delft University of Technology, The Netherlands; Mehdi Niazi Ardekani, KTH Royal Institute of Technology, Sweden; Milos Birvalski, MARIN Academy, The Netherlands; Mark Tummers, Delft University of Technology, The Netherlands; Ruud Henkes, Delft University of Technology, The Netherlands

Direct Numerical Simulations (DNS) have been performed of co-current turbulent two-phase (gas-liquid) pipe flow in the stratified flow regime with a smooth (flat) interface. A rectangular computational domain of  $3.4R \times 0.72R \times 2R$ , where  $R$  is the pipe radius, was used in which only the liquid phase was simulated. The effect of the overlying air flow was modeled through imposing a mean interfacial shear stress acting in the streamwise direction on the liquid layer below. A volume-penalization Immersed Boundary Method (IBM) was used to mimic the curved pipe geometry on a  $384 \times 200 \times 240$  Cartesian grid. The DNS results were compared with Particle Image Velocimetry (PIV) data, obtained in our lab, for the vertical midplane at a friction Reynolds number of 54.2 (based on the vertical midplane height  $h$  and the interfacial friction velocity  $u^*$ ),  $h = 0.69R$  and a bulk velocity of  $42.2 u^*$ . The results for the mean and rms velocities and the Reynolds shear stress agree very well. The DNS results reveal the presence of a mean secondary flow pattern consisting of two counter-rotating vortex pairs oriented in the streamwise direction. Detailed statistics of the turbulent flow were obtained at various Reynolds numbers.

#### THE EFFECT OF TURBULENCE ON PARTICLE IMPACTION ON A CYLINDER IN A CROSS FLOW

Jorgen Roysland Aarnes, Norwegian University of Science and Technology, Norway; Nils Erland L. Haugen, Norwegian University of Science and Technology, Norway; Helge Andersson, Norwegian University of Science and Technology, Norway

Particle-laden fluid flows are common both in nature and in a large number of industrial applications. In certain industrial applications, like filters, designers are interested in maximizing the total particle deposition, while in other applications, like industrial boilers, the aim is to minimize the deposition. Direct Numerical Simulations (DNS) in 3D are used to investigate inertial particle impaction on a stationary cylinder, with a focus on the effect of turbulence on the impaction efficiency. Different Reynolds and Stokes numbers are considered. Isotropic turbulence is introduced on the inlet of the flow domain.

It is found that for particles with Stokes numbers in the boundary stopping mode the front side impaction is significantly increased for turbulence with a large integral scale compared to for a corresponding laminar flow. Here, the boundary stopping mode corresponds to the range of Stokes numbers that lack sufficient inertia to penetrate the boundary layer but still do not follow the flow perfectly. The back side impaction efficiency is also found to be influenced by the turbulence, with the highest back side impaction efficiency occurring for turbulence with small integral scales.

\* Support from Research Council of Norway under the FRINATEK Grant 231444 is gratefully acknowledged.

#### TURBULENT THERMAL CONVECTION INDUCED BY THE HEATING OF A PARTICLE SUSPENSION

Remi Zamansky, INP Toulouse, France; Filippo Coletti, University of Minnesota, USA; Marc Massot, CentraleSupélec, EM2C, France; Ali Mani, Stanford University, USA

When a fluid is heated in a non-homogeneous way, the buoyancy forces could result in convective motions. What happens when the heat sources are a suspension of particles convected by the flow? The vortical structures generated by the thermal convection can alter the spatial distribution of the particles, which will in turn affect the flow. We recently studied this feedback loop mechanism considering a dilute suspension of heated particles (Zamansky, Coletti, Massot, and Mani, Phys. Fluid, 26, 071701, 2014). Local temperature fluctuations in the fluid, which are due to the non-uniformity of the dispersed phase, are observed to be large enough to trigger turbulent convective motions, in turn advecting the particles. Depending on the particle inertia, the fluid motions alter the spatial distribution of the particles and possibly lead to a strengthening of its non-uniformities. In this talk we explore the parameter space by performing direct numerical simulations of the fluid flow and Lagrangian tracking of the particles, for various particle number density, particle inertia and fluid domain size. As the inertia of the particle is increased, we observe a transition to the clustering regime. Based on an analogy with a phase transition, we discuss the implications for the scaling of the relevant statistical quantities (e.g. variances and dissipation rates of velocity, temperature, and concentration) with the aforementioned parameters.

#### EVAPORATION IN SUPERSONIC CO<sub>2</sub> EJECTORS: ANALYSIS OF THEORETICAL AND NUMERICAL MODELS

Francesco Giacomelli, University of Florence, Italy; Federico Mazzelli, University of Florence, Italy; Adriano Milazzo, University of Florence, Italy

Supersonic ejectors can be used in supercritical CO<sub>2</sub> vapor compression chiller to recover the throttling losses. Many configurations are possible that allow great benefits in terms of efficiencies (COP increases up to 30%). Although the application and testing of ejectors is not particularly difficult in these systems, the analysis of the internal dynamics is an extremely complex task, both by theoretical or numerical means.

In particular, all phase-change phenomena inside the ejector occur at high levels of speed and compressibility. This implies, for instance, that non-equilibrium transformations occur inside the primary nozzle. Further complex phenomena are the formation and atomization of liquid ligaments, the modulation of turbulence intensity by liquid droplets, the interaction between shocks and droplets, the release and absorption of heat by the mixture, etc... Unfortunately, all these aspects are very difficult to predict and no definitive theory has yet been formulated.

This work presents an analysis of various approaches that can be exploited to analyze the high-speed, non-equilibrium flashing of CO<sub>2</sub> inside a supersonic ejector. The limits and advantages of each approach are highlighted and some numerical results, obtained with commercial software, are discussed.

## A PHYSICALLY CONSISTENT SUBGRID SCALE BOILING APPROACH FOR MULTIPHASE CFD

Emilio Baglietto, *Massachusetts Institute of Technology, USA*

Traditional boiling closures for Multiphase Computational Fluid Dynamics (M-CFD) are based on the physical representation of nucleate boiling heat transfer, and are strongly biased towards simpler pool boiling conditions. All attempts at extending its applicability to complex flow boiling applications have illustrated a high sensitivity of critical closure parameters on local conditions, which are not replicated in the experiments. A novel subgrid scale approach is proposed, which aims at leveraging the advancements of the experimental techniques in the multiphase flow community. The sub-grid representation works to capture all physical phenomena affecting heat transfer at the heated surface. The model explicitly tracks: (1) the size of the bubbles on the surface, (2) the amount of microlayer and dry area under each bubble, (3) the amount of surface area influenced by sliding bubbles, (4) the quenching of the boiling surface following a bubble departure, and (5) the statistical bubble interaction on the surface. This approach provides the foundation for extension to high heat fluxes and boiling crisis, by explicitly tracking the local surface dry-spot conditions. Development validation, challenges and future advancements of the approach are critically reviewed in this work.

## TOWARDS A MORE GENERAL CFD MODELING FOR ALL FLOW REGIMES

Stephane Mimouni, *EDF R&D, France*; Solene Fleau, *EDF R&D, France*; Stephane Vincent, *Université Paris-Est, France*

Boiling crisis and flows occurring in a steam generator or during the dewating step of the cold leg of the reactor pressure vessel remain a major limiting phenomenon for the analysis of operation and safety of both nuclear reactors and conventional thermal power systems. Firstly, the choice is made to investigate a hybrid modeling of the flow, considering the gas phase as two separated fields, each one being modeled with different closure laws. In so doing, the small and spherical bubbles are modeled through a dispersed approach within the two-fluid model, and the distorted or large bubbles are simulated with an interface locating method.

This kind of approach is requiring the set of mass transfer terms between the continuous and the dispersed fields of the same physicochemical phase

The main outcome is the simulation of three field cases with a complete set of coupling terms between the two gas fields. The aim of the paper is to simulate the METERO experiments with this multifield approach. METERO is an experimental rig dedicated to the study of turbulent mixing of air and water in horizontal flows. The different regimes encountered in the METERO experiments, i.e. stratified flows, slug flows, plug flows, stratified dispersed bubbly flows and dispersed bubbly flows are simulated and presented in the paper.

## FIBERS SETTLING IN CELLULAR FLOW FIELD

Diego Lopez, *LMFA UMR 5509, CNRS – INSA de Lyon, France*; Laurence Bergougnoux, *Aix-Marseille University, CNRS, France*; Gilles Bouchet, *Aix-Marseille University, CNRS, France*; Elisabeth Guazzelli, *Aix-Marseille University, CNRS, France*

The transport of particles in a turbulent environment is relevant to many industrial and natural processes. In particular, most particles involved in such processes are not spherical, and particle anisotropy can yield complex behaviors.

In this work we present an experimental and theoretical study of fibers settling through a cellular flow field. Using a two-dimensional model experiment consisting in a simplified vortical flow, we investigate the influence of the particle aspect ratio and inertia on the settling rate and trajectories.

The experimental results are then compared to theoretical models, testing the accuracy of slender body theory and the influence of the different forces acting on the particle.

## CFD STUDY OF WATER DROPLETS IN A TOP ROTATING WALL DOUGHNUT CELL

Patrick Verdin, *Cranfield University, UK*; Lloyd Brown, *Science Deployed LLC, USA*

A transient three-dimensional CFD model was developed to study the flow inside a circular channel where the top wall was rotating. Details of the fluid flow within the channel under different thermal conditions, the bottom wall shear stress distribution, and its dependency on top wall rotational speed are provided. Water droplets injected in the domain are assumed to bounce on the upper, lower, inner diameter and outer diameter walls of the domain. The influence of the top wall rotation speed, secondary flow effects and temperature on the droplet concentration inside the doughnut cell and at the walls is also discussed. Results show that for small droplets, the highest concentration is located in the top/outer region of the doughnut cell while the concentration appears more uniform for large droplets.

## ENERGY EFFICIENT HIGH SOLIDS PROCESSING OF FIBER SUSPENSIONS USING MRI

Maria Cardona, *University of California, USA*; Emilio Tozzi, *Proctor & Gambel, USA*; Nardrapee Karuna, *University of California, USA*; Tina Jeoh, *University of California, USA*; Robert Powell, *University of California, USA*; Michael McCarthy, *University of California, USA*

Magnetic resonance imaging (MRI) velocimetry is used to characterize the real-time evolution of yield stress of cellulose suspensions undergoing enzymatic hydrolysis in a recycle-flow reactor. Enzymatic hydrolysis of cellulosic biomass is a key step of biological routes for production of fuels and chemicals. Economic considerations for large-scale implementation of the process require processes that operate in the high solids regime, concentrations of 15% by weight or higher. Such slurries are non-Newtonian and typically described as having a yield stress and a shear thinning viscosity. Such properties introduce processing challenges especially in the initial stages of hydrolysis (liquefaction). As implemented, MRI, a non-invasive measurement technique, is a key part of a rheometer that provides in-line real time measurements of complex fluid properties. Previous studies of the liquefaction of biomass have shown that in the initial stages rheological properties change quickly. Hydrolysis was performed in fed-batch mode, with fibers being added at various times. Each addition of fibers caused a rapid increase in yield stress followed by a decay that results from enzymatic action. The decrease in the yield stress was more rapid for the initial additions of fibers, and became slower as the hydrolysis progressed. Different strategies for adding enzymes to the reactor were found to impact overall cellulose hydrolysis and process efficiency.

## EXPERIMENTAL STUDY ON DISPERSION OF SPHERICAL PARTICLES AND FIBRES IN A JET PENETRATING A CROSSLFLOW

Lars Pasternak, *Martin-Luther-University Halle-Wittenberg, Germany*; Martin Sommerfeld, *Martin-Luther-University Halle-Wittenberg, Germany*

In the present work, the dispersion of spherical particles or fibres in a small jet issuing into a crossflow is investigated in dependence of the local Stokes-number. The experimental configuration is a turbulent round jet (inner diameter  $d_i = 11$  mm) injected perpendicularly into a fully developed turbulent crossflow through a square channel (100 mm x 100 mm). The measurements are conducted by using a Particle Image Velocimetry (PIV)-system in connection with a high speed camera and a LED flashlight for shadow imaging.

The jet is seeded with different types of fibres and spherical particles at low concentration and in addition with 40  $\mu$ m PMMA tracer for measuring the jet fluid velocity. The arithmetic fibre aspect ratio has values between 6 and 10 for the different fibre types. The spherical particles are chosen according to the equivalent fibres Stokes-numbers. The local Stokes-number is obtained by means of the equivalent circular diameter of the fibres  $d_{10} = 90$   $\mu$ m and  $d_{10} = 226$   $\mu$ m (material density of 1.19 g/cm<sup>3</sup>) and the characteristic time scale of the jet vortices. Besides the velocity of spheres and fibres, also the fibre orientation and angular velocity is determined. Several image filters are used to distinguish tracer and fibres/solids and determine the orientation of the fibres. In addition the dispersed phase concentration distribution and correlations between particle properties and fluid velocities are evaluated.

## MODELLING OF GAS-SOLID FLOWS WITH NON-SPHERICAL PARTICLES

Berend Van Wachem, Imperial College London, UK; Thomas Curran, Imperial College London, UK; Fabien Evrard, Imperial College London, UK; Robert Wilson, Imperial College London, UK

In this contribution, a novel methodology is introduced to predict the behaviour of turbulent flows laden with non-spherical particles. The computational model comprises different elements. Firstly, a novel collision model to deal with the collisions between non-spherical particles and these particles and the wall, including wall roughness, is constructed based upon a Quaternion approach. Secondly, the drag, torque and lift relations of various particle shapes is determined by means of true DNS, where the particle is represented by the immersed boundary method. A large number of simulations are performed for each particle, to gather data on the drag, torque, and lift on the particle under various Re numbers and angles of attack. All these simulations are coarse-grained into drag, lift and torque correlations. Finally, the resulting relations and collision model are used in a fully four-way coupled point-particle approach to study: (i) horizontal turbulent channel flow, and (ii) dense fluidized beds. Four types of particle shapes will be considered: spheres, ellipsoids, disk-shape and fiber-shape.

The results show that the particle shape has a large effect on both the behavior of the particles as well as the behavior of the flow. Moreover, it will be shown that wall roughness has a lesser effect on non-spherical particles than on spherical particles. Finally, the preferential orientation of non-spherical particles in fluidized beds and channels will be shown.

## NUMERICAL SIMULATION OF COLLISION DYNAMICS FOR FINITE SIZE PARTICLES IN TURBULENT SUSPENSION

Stephane Vincent, Université Paris-Est, France; Jorge Cesar Brandle De Motta, CORIA, France; Eric Climent, Université de Toulouse, IMFT, CNRS, France; Jean-Luc Estivalezes, ONERA, France

We present in this paper particle resolved direct numerical simulation suspended in homogeneous sustained turbulence. Three particle-fluid density ratio (1,2,4) has been simulated for a turbulent Reynolds number of 73 with 512 spherical particles and solid fraction of 3%. A soft sphere and a sub-grid lubrication models are used for particle collisions. Pdf are analyzed for particle acceleration, inter-collision time as well as velocity angle between particles at contact. The analysis has been investigated first without lubrication model. Concerning acceleration pdf, an exponential distribution is found with a clear influence of collision for the largest accelerations. This effect is enhanced as density ratio rises. The inter-collision time has been compared to the kinetic theory of granular media. The same orders of magnitude are found. In order to describe more precisely the collision regime, velocity angle pdf at collision are studied. Whatever is the density ratio, the pdf are shifted towards low angle compared to the theory, indicating that particles come into contact with nearly parallel velocities. Taking into account lubrication model shows a reduction of the inter-collision time which is more significant for low density ratio. This can be explained by the occurrence of many body solid-solid collisions during the activation of the lubrication model. This is clearly the signature of consecutive collisions as lubrication slows down the particles during separation after collision. For highest density ratio simulated, this effect is less pronounced.

## MODELING HEAT TRANSFER IN REACTING GAS-SOLID FLOW USING PARTICLE-RESOLVED DIRECT NUMERICAL SIMULATION

Shankar Subramaniam, Iowa State University, USA; Bo Sun, Iowa State University, USA; Sudheer Tenneti, CD-Adapco Inc., USA

The purpose of this work is to develop gas-solid heat transfer models using Particle-resolved Direct Numerical Simulations (PR-DNS). Fluid heating is important in gas-solid heat transfer, especially in dense low-speed flows. Gas-solid heat transfer in a steady thermally fully-developed flow through a homogeneous fixed assembly of monodisperse spherical particles is simulated using the Particle-resolved Uncontaminated-fluid Reconcilable Immersed Boundary Method (PUREIBM). The average gas-solid heat transfer that appears in the average fluid temperature evolution equation is quantified and modeled using PR-DNS results. Fluctuations in velocity and temperature are also important in gas-solid flow, and the transport of temperature-velocity covariance that appears in the average fluid temperature evolution equation is computed using PR-DNS data and modeled by a gradient-diffusion model. It is found that the transport of temperature-velocity covariance is significant when compared to the average gas-solid heat transfer. A budget analysis of the mean fluid temperature equation provides insight into the variation of the relative magnitude of the various terms over the parameter space. The temperature variance and Probability Density Function (PDF) of temperature are important in reacting gas-solid flow and are also discussed. These models can be used to improve two-fluid simulations of heat transfer in fixed beds.

\* Supports from grant DE-AC02-07CH11358, CBET 1034307 and CBET 1336941 are gratefully acknowledged.

## MODELLING THE MOTION OF FINITE-SIZE PARTICLES NEAR A MOVING WALL BY A TWO-WAY COUPLING APPROACH

Francesco Romano, TU Wien, Austria; Hendrik Kuhlmann, TU Wien, Austria

Steady incompressible two-dimensional particle-laden flow in a lid-driven cavity is investigated. Owing to the driving mechanism the streamlines gather closely near the constantly moving lid. Due to its finite size each particle will experience a strong force during its passage along the moving wall. Such a particle-boundary interaction manifests itself by affecting the particles' trajectories globally and can give rise to particle accumulation structures. To study such dissipative particulate structures induced by a moving wall, the transport of a finite-size particle is investigated by a numerical two-way coupling approach aimed at resolving all relevant scales of the flow, including the lubrication gap. The latter turns out to crucially determine the particle's trajectory if the distance between its centroid and the moving lid is of the order of the particle radius. This effect leads to a sensible change of the particle path from the one that can be calculated via a one-way coupling approach for a point particle initially placed in the same position. Considering small Stokes numbers, the two-way coupled simulations are compared with trajectories obtained using the Maxey-Riley equation with and without a particle-wall interaction which is modelled as an inelastic collision.

## TRANSLATIONAL AND ROTATIONAL DYNAMICS OF FINITE-SIZED LIGHT SPHERES IN TURBULENCE

Varghese Mathai, University of Twente, The Netherlands; Chao Sun, University of Twente, The Netherlands; Detlef Lohse, University of Twente, The Netherlands

We report experimental results on the translational and rotational dynamics of finite-sized light spheres in turbulence. We employ an orthogonal camera setup to simultaneously track the particle's position and absolute orientation using imaging techniques. The detection method employs a minimization algorithm to obtain the orientation from a 2D projection of a specific pattern drawn onto the surface of the sphere. We find that even a marginal reduction in the particle's density from that of the fluid can result in strong modification of its dynamics, which we trace back to the particle's own wake-induced forces. At sufficiently low Ga ( $\leq 30$ ) the particles respond to the carrier turbulent flow, while at higher Ga ( $\geq 225$ ) their motions are dominated by wakes, with only a weak influence from the surrounding turbulent flow. We present results on the statistics of rotation and the coupling between rotation and translational dynamics.

## SETTLING OF FINITE-SIZE PARTICLES IN TURBULENT ENVIRONMENTS AT DIFFERENT GALILEO NUMBERS

Walter Fornari, KTH, Royal Institute of Technology, Sweden; Francesco Picano, University of Padova, Italy; Luca Brandt, KTH Royal Institute of Technology, Sweden

Particle sedimentation is widely encountered in environmental and engineering applications such as dust storms, fluidized beds and soot particle dispersion. However, little is known about the settling of finite-size particles in homogeneous isotropic turbulence. The aim of the present study is to investigate the sedimentation of large, non-colloidal spheres at different Galileo numbers (Ga), in both quiescent and turbulent environments. The Galileo number is the ratio among buoyancy and viscous forces and different Ga correspond to different solid to fluid density ratios. To this aim, Direct Numerical Simulations are performed using an Immersed Boundary Method to account for the dispersed phase. The solid volume fraction considered is 0.5%, while the Galileo number is in the range 19 to 200. The particle diameter is chosen to be approximately 12 Kolmogorov lengthscales in nominal conditions. The results show that the mean settling velocity is lower in an already turbulent flow than in a quiescent fluid. The reduction in mean settling velocity (with respect to the quiescent cases) increases from about 10% up to 60% as the Galileo number is reduced. In the final contribution, we will discuss the importance of nonlinear and unsteady contributions to the total drag for each case.

## SCALE INDEPENDENCE OF COLLECTIVE PARTICLE SETTLING IN FRACTAL CHANNELS

Yudai Kurose, Hokkaido University, Japan; Kanami Ishizawa, Hokkaido University, Japan; Shusaku Harada, Hokkaido University, Japan

Collective settling of particles in a liquid-filled fractal-shaped channel by gravity is investigated experimentally. When the concentration gradient exists opposite to the gravity, the particulate suspension consisting of fine particles settles collectively and it forms a finger-like blob due to Rayleigh-Taylor instability. Besides, the suspended region behaves immiscible to surrounding fluid. We recorded such a collective settling behavior in a fractal-shaped channel by video cameras and quantified the occupancy ratio of suspended region by image analysis. The occupancy volume and the settling velocity of suspended particles in channels are greatly dependent on the geometric conditions of the channel, particularly the aspect ratio of the channel cross-section. The invasion behavior of particles into each branch channel has been modeled by sigmoid function. As is well-known, the geometric quantities of fractal channel such as the length or the volume of each branch channel are directly related to the fractal characteristics. Consequently, the developed model consists of only fractal characteristics of the channel such as the homothetic ratio or the number of branches. The model results agree quantitatively with the experimental results. These facts indicate that it is possible to predict the gravitational dispersion behavior of particles in an arbitrary fractal channel regardless of its lengthscale.

\* This study was supported by JSPS KAKENHI Grant No. 26420095.

## VALIDATION OF THE TWOPHASEEULERFOAM SOLVER FOR JET INLET FLUIDIZED BEDS

Giovanni Tretola, Consiglio Nazionale delle Ricerche, Italy; Francesco Saverio Marra, Consiglio Nazionale delle Ricerche, Italy

Gas-solid fluidized beds operating in the bubbling regime are widely used in chemical engineering and in the power industry. The huge number of particles involved in these systems makes the adoption of Lagrangian models not suitable for the simulation of devices of practical interest. As a consequence, the Eulerian-Eulerian two-phase approach is preferred. An implementation of this approach is offered in the application twoPhaseEulerFoam, presents in the OpenFOAM library. This implementation represents a base for several extensions proposed in the literature, like the BIOTC code or the biomassGasificationFoam code. However any extensive validation of the solver has been conducted up to now. Aim of this work is to verify the ability of the solver to correctly reproduce the experimental set-up proposed in Kuipers et al. (Powder Tech. 1991). Firstly, numerical parameters are investigated, to determine the grid spacing and time step requirements. Then the effect of the available models and closure correlations on the correct reproduction of the transient phenomenon will be illustrated. Finally several numerical caveats, as a proper selection of the linear solvers for the different equations and the scalability for parallel execution will be highlighted, giving a guidance for a successful integration.

## GRANULAR TEMPERATURE AND PARTICLE VELOCITY-CONCENTRATION CROSS-CORRELATION IN A LIQUID FLUIDIZED BED

Maike Baltussen, University of Twente, The Netherlands; Martin Van Der Hoef, University of Twente, The Netherlands; Micheline Abbas, LGC-CNRS/University of Toulouse, France; Olivier Masbernat, Tlernoine, France

A statistical model of the granular temperature in LFB is proposed. It relies on production-dissipation equilibrium of particle kinetic energy as derived from the Kinetic Theory of Granular Flows. Based on a first order assumption, it can be shown that the production term is scaled by the cross-correlation of particle velocity and concentration fluctuations. In the present work, we want to test this assumption on the basis of fully-resolved simulations, which can provide detailed information that is not readily accessible through experiments. For simulating the flow between the spheres, a lattice Boltzmann model is used, where the no-slip condition at the surface of the particles is controlled by Ladd's bounce back rules. Particle-particle interactions are modelled as hard-sphere, with lubrication forces added. The simulations provide us direct insight into the nature of fluctuations in LFB. Particle velocity components PDF, variance and anisotropy factor as a function of particle solid fraction are compared with experimental data on the fluctuating motion of 6.3mm PMMA particles in LFB. Cross-correlations of the concentration fluctuation with the particle velocity and pressure gradient fluctuation are calculated and their implication for the statistical models is discussed.

## PARTICLE FLUXES AND IRREVERSIBILITY DUE TO SHEAR FLOW IN A BIDISPERSE SUSPENSION

Amanda Howard, Brown University, USA; Martin Maxey, Brown University, USA; Kyongmin Yeo, IBM Thomas J. Watson Research Center, USA

We consider a primary viscous suspension of neutrally buoyant particles in a shear flow and introduce particles of the same size but subject to a body force, as may arise in sedimentation under gravity. The goal is to study how the two phases interact in terms of irreversibility of the particle motion and the net flux of particles. Numerical simulations are used to investigate this model system, which while related to the classic problem of viscous resuspension sheds light on the details of stress balance models and the role of contact forces due to particle roughness. Standard models rely on empirical correlations for hindered settling and a flux generated by particle stresses. The full simulations allow for a detailed evaluation of particle stresses and instantaneous microstructures such as force chains that may form. For neutrally buoyant particles, contact forces have been shown to be the primary factor in the irreversible motion of the particles, while sedimenting particles may be chaotic simply through longer range hydrodynamic interactions.

## A DISCRETE PARTICLE MODEL FOR MEMBRANE ASSISTED FLUIDIZED BED REACTORS

Lianghui Tan, Eindhoven University of Technology, The Netherlands; Milan Mihajlovic, Eindhoven University of Technology, The Netherlands; Ivo Roghair, Eindhoven University of Technology, The Netherlands; Martin Van Sint Annaland, Eindhoven University of Technology, The Netherlands

Gas-solid fluidized bed membrane reactors are relatively new, integrated reactor concepts that allow both the reaction and separation in a single apparatus, with the benefits of increased conversion, purified product streams and reduced costs of operation. Previous work, employing permeable walls (non-selective membranes), shows that the hydrodynamic behaviour of these reactors may be strongly influenced (e.g. formation of densified zones near the membranes), possibly increasing mass transfer resistance. In this work, we present the development and use of a Discrete Particle Model (DPM, an Euler-Lagrange method) extended with the gas phase species balances and selective wall-mounted membranes. The newly developed technique allows us to quantify the actual, selective product extraction from the reactor, investigate the effect of the selective membranes on the degree of concentration polarization and mass transfer limitations as a whole and to come up with design and operation specifications for this type of reactors.

\*This project is financially supported by the Netherlands Organization for Scientific Research (NWO), STW VIDI Grant No. 10244.

## INVESTIGATION OF GAS-SOLIDS FLOW CHARACTERISTICS IN A CIRCULATING FLUIDIZED BED WITH MULTI-CYCLONE SEPARATORS BY CPFD SIMULATION AND ECT MEASUREMENT

Haigang Wang, Chinese Academy of Sciences, China; Guizhi Qiu, Institute of Engineering Thermophysics, China; Jiamin Ye, Institute of Engineering Thermophysics, China; Zhipeng Wu, School of Electrical and Electric Engineering, UK; Wuqiang Yang, School of Electrical and Electric Engineering, UK

Super-critical circulating fluidized bed (CFB) is a clean coal combustion technology that is becoming customary in coal-fired power plants. However, it is necessary to investigate the complex gas-solids flow characteristics in a circulating fluidized bed with multi-cyclone separators, especial for a large scale CFB system. In this research, an Eulerian-Lagrangian model called CPFD was applied to investigate the whole-loop of the gas-solids flows in a CFB with four cyclone separators. Electrical Capacitance Tomography (ECT) and Microwave Doppler technologies are used to measure the solids concentration as well as solids velocity with the same CFB systems. Experimental results are given and compared with CPFD simulation results. Key process parameters, including solids phase concentration, solids mass flux among the different loop seal and pressure profiles along the CFB riser are given and analyzed. In the CPFD simulation, the effect of drag model on the prediction errors is given. The research presented a new strategy combined CPFD simulation and ECT plus Microwave Doppler measurement to investigate the complex gas-solid two phase flow hydrodynamics characteristics in a CFB system. The results provide valuable information for the optimum design of a circulating fluidized bed.

\* Support from grant NSFC No.61320106004 and No.XDA07030100 is gratefully acknowledged.

#### **AEROSOL SAMPLING OF SUB-MICRON PARTICLES USING A PORTABLE ELECTROSTATIC IONIZER: EFFECT OF BULK FLOW VELOCITY ON COLLECTION EFFICIENCY**

*Imani Jajarmi Ramin, KTH University, Sweden; Robert Etienne, Polytechnique de Montreal, Canada; Ladhani Laila, KTH University, Sweden; Wouter Van Der Wijngaart, KTH University, Sweden*

The collection efficiency of a portable electrostatic ionizer used to sample biological aerosols is assessed in a wind tunnel. The objective is to gather information on the effect of the flow field surrounding the sampler on its collection efficiency, especially as a function of particle size. The ionizer is designed to collect aerosols in indoor spaces, where air currents are expected from ventilation systems. In these settings, the low collection efficiency associated with electrostatic precipitators for sub-micron particles is compounded by the presence of a bulk flow. To investigate the effect of the electrical properties of the particles on the collection efficiency both conductive and non-conductive aerosol were generated. Downstream of the sampler, an aerosol spectrometer was used to measure the aerosol concentration and size distribution. The velocities and particle sizes range investigated are 0-2m/s and 250nm-5µm. The results show as expected lower collection efficiency as the particle size decreases, with a sharp decrease when the bulk flow is increased. For aerosols of oil droplets particle sizes smaller than 500nm were repelled instead of being captured by the ionizer's collector, which is most likely the result of the formation of a thin charged oil film on the collection plate.

\* The research presented here is supported by the KTH Linné Flow Center and the European Union's Seventh Framework Programme (FP7) under Grant Agreement No. 604244 (Norosensor).

#### **CONTINUOUS PRODUCTION OF ORGANIC POLYMERS IN MICROSTRUCTURE REACTORS: FEASIBILITY AND SCALE-UP**

*Juergen J. Brandner, Karlsruhe Institute of Technology, Germany; Christof Kuesters, Cargill, Belgium; Berta Spasova, Karlsruhe Institute of Technology KIT, Germany; Bruno Stengel, Cargill, Belgium*

Polymerization processes are normally not performed in micro structured equipment due to increased pressure losses and possible fouling and blocking of the miniaturized equipment. In this work, the results of a feasibility study for the synthesis of an organic polymer using micro structured reactors are presented. The product is an important raw material for the food industry. The conventional production process takes place in a batch reactor. The aim of this work is the transformation of the batch processes into continuous two-phase processes with micro structured reactors. Using micro structured equipment, the process parameters can be adapted to the true process needs and are no longer limited to the device properties. Thus, due to the advantages of microstructure reactors concerning excellent heat transfer, improved mass transfer and short mixing times, the reactor volume and reaction time have been significantly decreased compared to the conventional processes. In a multi-stage microstructure-based process, the reaction temperature has been raised by more than 30% compared to the standard, while the residence time was reduced from several hours to some ten seconds. This results in an increase in space-time yield by a factor of about 100.

#### **HYDRODYNAMIC NON-NEWTONIAN LIQUID-SOLID FLOW IN COMPOUND TAYLOR-COUPETTE FLOW**

*Anatoly Aleksandrov, Bauman Moscow State Technical University, Russia; Vladimir Devisilov, Bauman Moscow State Technical University, Russia; Elena Sharay, Bauman Moscow State Technical University, Russia*

Currently there is a need for filtration of two-phase flows with high viscosity of non-Newtonian dispersion medium in many industries. The feature of high viscosity non-Newtonian liquid-solid flow separation is small resource of plant equipment and large power consumption. The authors suggest using a combination of filtration with force action on the non-Newtonian liquid-solid flow to create additional strain rates to solve the above problems. This organization of the flow reduces effective viscosity of a pseudoplastic fluid complying with rheological power law de Waele-Ostwald. The aim of this work is to study the behavior of solid particles in power-law non-Newtonian liquid in complex force action on the flow. The authors developed the mathematical model and performed the calculation of two-phase swirling flow moving in narrow annular channel with rotating permeable inner wall, which is a special case of the Taylor-Couette flow. It was determined the influence of the geometrical dimension of the channel and operating conditions on the flow stability and on separation efficiency of solid particles smaller than 80 µm. The positive effect has been shown in combination of filtering and complex force action for the non-Newtonian liquid-solid flow.

\* This work was performed under the state order of the Ministry of Education and Science of the Russian Federation No. 2014/104 (project No. 1469).

#### **SIMULATION OF TWO-PHASE FLOWS IN MICROSCOPIC NETWORKS OF INTERCONNECTED PORES TO IMPROVE MANUFACTURING PROCESS OF CERAMIC MATRIX COMPOSITES (CMCS).**

*Audrey Pons, SAFRAN Herakles / I2M, France; Eric Arquis, CNRS, France; Marc Valat, CNRS, France; Cedric Le Bot, CNRS, France; Cedric Descamps, SAFRAN Herakles, France; Elliott Bache, France*

The development of ceramic matrix composites (CMCs) to replace certain metal components in internal hot parts of aircraft engines is an active research field for the aeronautical industry. These advanced components may be manufactured with a fluid processing called Reactive Melt Infiltration (RMI). The densification step is the capillary rise of a molten metal such as silicon within a woven preform. The molten metal can react with the previously introduced ceramic powder and lead to a refractory matrix. The competition between capillary rise and the reaction between molten silicon and introduced carbon powder has to be managed. The ambition is to prevent choking off effects and closed pores created by preferential paths. Simulations of two-phase flows at microscopic scale were undertaken with CFD code Thétis (developed at I2M, Bordeaux) based on Volume Of Fluid approach. An Eulerian model has been previously added to take into account capillary effects on solid surfaces. As a first step, infiltration distance, rise velocity and pressure jump during capillary rise in microscopic simple geometries were examined and compared with analytical predictions given by Bosanquet's and Washburn's laws and Dullien's approach. The next step is the inclusion of a microscopic reactive model to handle modifications of pore space, solid surface composition and thermic field.

\* This work is financially supported by SAFRAN Herakles and the ANRT.

#### **Combined effect of Tip wall distance and Heterogeneity of wall potential on electroosmotic flow within a charged Micro and nano-channel**

*Ameeya Kumar Nayak, IIT Roorkee, India*

There have been numerous experimental and numerical studies of electroosmotic flows (EOF) in micro and nano channels. Surface roughness has been considered as a passive means of enhancing species mixing in EOF through micro fluidic systems. It is highly desirable to understand the synergetic effects of roughness and surface heterogeneity on the EOF through microchannels. One way to enhance biochemical reactions on sensing surfaces is to increase the reaction (sensing) surface area. In a microchannel, creating many roughness elements on the microchannel walls can increase the surface area. These roughness elements can be produced by photolithography-based microfabrication techniques. However, these roughness elements will inevitably influence the liquid flow and sample transfer in the microchannel. If the surface of these roughness elements is chemically modified, the electrokinetic properties of these roughness elements will be different from those of the substrate surface. This will bring more complication to the electrokinetic transport processes. The simulation results can be used as guidelines for the optimization and design of micro devices in terms of mixing and species transport. In this paper a model has been developed for the EOF based on binary diffusion process. The governing equations for the Nernst Plank model are derived from Fick's law of diffusion. Navier-Stokes equations for the flow along with electric field terms are derived from Maxwell's equation along with the species terms

#### **DYNAMICS OF A RISING BUBBLE IN A VISCOELASTIC LIQUID**

*Matthias Niethammer, Technische Universität Darmstadt, Germany; Holger Marschall, Technische Universität Darmstadt, Germany; Christian Kunkelmann, BASF SE, Germany; Dieter Bothe, Technische Universität Darmstadt, Germany*

Direct numerical simulations (DNS) are performed to study the dynamics of a rising gas bubble in a viscoelastic liquid. The polymeric solution is modeled by the Oldroyd-B constitutive equations. The constitutive equations are reformulated to obtain numerically more robust representations for high Weissenberg number (HWN) flows. The numerical stabilization is realized using a generic framework that includes several published specific stabilization methods, such as the logarithm conformation representation (LCR) and the root conformation representation (RCR). The mathematical formulation of the two-phase flow problem is derived by conditionally averaging the local instantaneous transport equations and applying a volume of fluid (VoF) model to obtain a closed-form one-field representation. The framework is implemented into a finite volume method (FVM) on general unstructured computational grids. We investigate characteristic flow phenomena that occur at certain critical bubble volumes, i.e. the cusp-shaped trailing edge, the negative wake, and the jump discontinuity in the steady-state rise velocity.

\* The authors would like to acknowledge BASF SE for the financial support of this work, as well as Erik Wassner and Sebastian Weisse for their cooperation, assistance, and enlightening discussions.

## THE DYNAMIC BEHAVIOR OF A BUBBLE RISING IN A HYDROPHOBICALLY MODIFIED ALKALI-SOLUBLE EMULSION POLYMER (HASE) SOLUTION; DEPENDENCE ON THE HASE TYPE

Mitsuhiro Ohta, Tokushima University, Japan; Mao Ogawa, Tokushima University, Japan; Shuichi Iwata, Nagoya Institute of Technology, Japan

It has been reported by Soto et al. (2006) and Ohta et al. (2015) that the motion of a bubble rising freely in a Hydrophobically modified alkali-soluble emulsion polymer (HASE) solution has a very unique shape distinct from the shape of a bubble rising in a water-soluble associative polymer solution. In their studies, PrimalTM TT-935 (Dow Chemical Com.) was used as the specific HASE solution. In this study, PrimalTM TT-615 (Dow Chemical Com.) is used as the HASE solution. The bubble dynamics, such as deforming bubble shape and rise velocity are compared between a bubble rising in a TT-615 solution versus previous experiments of a bubble rising in a TT-935 solution. First, it is found that the rheological properties for PrimalTM TT-935 and PrimalTM TT-615 are quite different. Second, and surprisingly, it is observed that the unique bubble dynamics/bubble-shapes that were observed for a single bubble rising in a HASE PrimalTM TT-935 solution do not appear in the experiments using the HASE PrimalTM TT-615 solution. In other words, it is found that the very peculiar bubble-shapes that were observed for a bubble rising in a HASE PrimalTM TT-935 solution are not caused by the characteristic features specific to HASE materials as much as they are caused by the differences between types of HASE materials. Based on the experimental results observed using the PrimalTM TT-615 HASE solution, the specific mechanisms underlying the unique bubble shapes formed in a PrimalTM TT-935 solution is analyzed.

## TWO-FLUID MODEL PREDICTION OF GAS-LIQUID FLOW IN A VERTICAL PIPE USING AN ALGEBRAIC SECOND MOMENT CLOSURE

N.A. Adoo, University of Saskatchewan, Canada; A. S. M. Atiqul Islam, University of Saskatchewan, Canada; Donald J. Bergstrom, University of Saskatchewan, Canada

The present study considers a numerical prediction for fully-developed gas-liquid flow in a vertical pipe using the Eulerian two-fluid model. A radial force balance for the gas phase is used to determine the gas volume fraction profile; the predicted profiles agree with the experimental data for a range of bubble sizes. Although an eddy viscosity model together with a two-equation closure is able to predict the turbulent transport in a simple shear flows, it is fundamentally incapable of predicting more complex flows such as flow through a pipe bend, where the effects of curvature can profoundly modify the flow structure. In anticipation of such applications, the present study documents the implementation of a second moment closure, in this case an explicit algebraic Reynolds stress model (EARSM). Of special interest is the approach for incorporating the effect of the bubble phase on the Reynolds stress components in the liquid phase, as well as the treatment of the bubble induced turbulence in the context of an EARSM closure. The study also examines the performance of wall functions used to avoid fully resolving the near-wall region. Overall, the paper documents a turbulence model capable of predicting gas-liquid flow in complex duct geometries.

## EXPERIMENTAL STUDY ON STRUCTURE OF A DILUTE GAS-PARTICLE AXISYMMETRIC OPPOSED JETS FLOW

Jing Li, Huazhong University of Science and Technology, China; Hanfeng Wang, Central South University, China; Zhaohui Liu, Xi'an Jiaotong University, China; Chuguang Zheng, Huazhong University of Science and Technology, China

Turbulent structures in a gas-particle axisymmetric opposed jets are experimentally investigated by means of a simultaneous two-phase PIV method. The measurements are conducted at  $Re = 14500$ . Glass beads with an averaged diameter of  $100 \mu m$  ( $St = 205$ ) are used as dispersed phase. Three low mass loading ratios (0.01, 0.02 and 0.04) and three nozzle separations (5, 12 and 20 times nozzle diameter) are tested. 2D gas-phase velocity fields were analyzed with proper orthogonal decomposition (POD) and finite time Lyapunov exponents (FTLE). Meanwhile, particle-phase was examined by particle tracing velocimetry (PTV). It is found that, in the present dilute gas-particle opposed jets flow, inter-particle collision can be neglected, and particles can penetrate into the opposite stream until to the vicinity of the opposite nozzle. The presence of particles significantly modifies the axial jet structures, representing the coherent vortex elongating in streamwise direction and shortening in transverse direction, meanwhile noticeably suppresses the turbulence kinetic energy transport from the axial jet to the radial jet. Furthermore, the presence of particles also can alter the Lagrangian coherent structures (LCS), which represent characteristics of impingement plane.

\* Support from grant Nos. 51306065, 51390494, 51276076 and FSKLCCB1409 are gratefully acknowledged.

## INVESTIGATION OF DIRECT NUMERICAL SIMULATIONS OF QUASI-STEADY PRIMARY BREAK-UP OF SHEAR THINNING LIQUIDS

Moritz Ertl, University of Stuttgart, Germany; Grzegorz Karch, University of Stuttgart, Germany; Filip Sadlo, Heidelberg University, Germany; Thomas Ertl, University of Stuttgart, Germany; Bernhard Weigand, Universitaet Stuttgart, Germany

Sprays from Non-Newtonian fluids are used in many technical applications, such as spray drying to create well defined particles. The resulting droplet size distribution has a large influence on the properties of the particles, but also on the efficiency of the process. To improve our understanding of spray drying, we investigate direct numerical simulations (DNS) of the primary break-up of Non-Newtonian shear-thinning fluids. We use our in-house code Free Surface 3D (FS3D), which is based on the volume of fluid (VOF) method to track the interface, and we use piecewise linear interface calculation (PLIC) to obtain a sharp liquid interface. For the simulation, we inject aqueous solutions of different concentrations of the polyacrylamide Praestol into a computational domain with over 700 million cells. We simulate the injection velocity profiles of different nozzles at a Reynolds number of about 20,000. Simulations are run until a quasi-steady state is reached, in which we quantitatively investigate the surface deformations of the liquid core and if break-up has started, we analyze the sizes of the created droplets. With visualization techniques tailored for two-phase flows, we also analyze the origin of the droplets within the liquid core as well as their spatio-temporal development. With this investigation, we hope to better understand the destabilizing parameters, that influence the droplet size.

\*Support for the German Research Foundation (DFG) under the Priority Programme SPP 1423 is gratefully acknowledged.

## PREDICTION OF GAS VOLUME FRACTION FOR POLY-DISPERSE GAS-LIQUID TURBULENT FLOW IN A VERTICAL PIPE

A. S. M. Atiqul Islam, University of Saskatchewan, Canada; N.A. Adoo, University of Saskatchewan, Canada; Donald J. Bergstrom, University of Saskatchewan, Canada

This paper presents a numerical model for predicting fully developed poly-disperse gas-liquid upward flow in a vertical pipe. The Reynolds Averaged Navier-Stokes equations, using an Eulerian two-fluid model formulation, are solved using the finite volume method; a low Reynolds number eddy viscosity model is used to predict the turbulence field for the continuous phase. The gas volume fraction profile is obtained from a radial force balance on the bubble phase. The results obtained for the mono-disperse case are shown to be in good agreement with the experimental data, and indicate that the effect of the bubbles on the liquid phase turbulence is strongly depended on the bubble size. A major objective of the present study is to investigate the effect of multiple bubble size groups on both the liquid phase turbulence and bubble induced turbulence. The study will also incorporate models for bubble coalescence and break-up, so that it can predict a pipe flow with a realistic range of bubble sizes. The authors acknowledge the financial support provided by the Canadian Nuclear Laboratories for the initial stage of this ongoing research project.

## TURBULENT STRUCTURES IN LEAD-BISMUTH FLOWS MEASURED BY USING INTRUSIVE PROBE METHODS

Gen Ariyoshi, Graduate School of Energy Science, Kyoto University, Japan; Daisuke Ito, Kyoto University Research Reactor Institute, Japan; Yasushi Saito, Kyoto University Research Reactor Institute, Japan; Kaichiro Mishima, Institute of Nuclear Safety System, Incorporated, Japan

To make clear flow structure in a liquid-metal two-phase flow, Lead-Bismuth Eutectic (LBE) two-phase and single phase flows were measured by using intrusive probe methods. Miniature electro-magnetic probe and electrical conductivity probe were used to measure local liquid velocity fluctuation and local void fraction in the LBE two-phase flow at low void fraction conditions, respectively. An LBE test loop was used which consists of a test section, a gas-liquid separator, a gas injection nozzles and an electro-magnetic pump. The test section was a vertical round pipe having 50 mm of the diameter and 2 m of the height. The working fluids were nitrogen gas and LBE. Nitrogen gas was injected from the bottom of the test section. Therefore, vertical upward flow could be formed in the test section. From the experimental results, it was found that liquid velocity profiles showed wall-peak at low void fraction conditions. And similar tendency could be found also in the void fraction profile. We consider that such experimental results might be attributed to the poor wettability of the test section wall. To verify this hypothesis, the liquid velocity and void fraction were measured by using a test section with good wettability condition. In addition, the liquid velocity and void fraction were compared with existing experimental results to discuss the details of the turbulent flow structure of the LBE two-phase flow.

## CFD MODELLING OF POLYDISPERSE BUBBLY FLOWS

Alexander Vichansky, CD-adapco, UK

Accurate CFD modelling of gas/liquid, liquid/liquid polydisperse flows requires two key models: (i) turbulent dispersion of bubbles (droplets) in order to produce correct spatial distribution of the dispersed phase and (ii) population balance in order to account for the polydisperse nature of the flow. In the present work we use the A-MuSiG method implemented in STAR-CCM+ CFD code of CD-adapco. The dispersed phase is split into M size groups having their own velocities. Size of a group is not prescribed a priori, but calculated from an additional scalar equation. The instantaneous two-phase Navier-Stokes equations balance gravity inertia, pressure, drag and virtual mass forces. The latter four are non-linear and being Reynolds averaged produce unclosed terms, which are modelled using a gradient closure. That is, the apparent turbulent dispersion is a sum of several physical mechanisms. Different breakup and coalescence models are implemented in the code and we discuss their performance. We model bubbly flows in horizontal and vertical pipes and compare void and size distribution to the experimental data. Different bubble size have different velocities what has a significant implications for the mixing in the transversal direction.

## CFD MODELLING OF PHASE SEPARATION AND COALESCENCE IN A MICELLAR THREE-PHASE SYSTEM

Lisa-Maria Wagner, Otto-von-Guericke Universität Magdeburg, Germany; Lena Hohl, Technische Universität Berlin, Germany; Luis Medeiros De Souza, University of Magdeburg "Otto von Guericke", Germany; Dominique Thevenin, University of Magdeburg "Otto von Guericke", Germany; Matthias Kraume, TU Berlin, Germany

A practically relevant micellar multiphase system used to carry out a long-chain hydroformylation reaction is considered in this project. It contains three phases (water, 1-dodecene and a surfactant), is mixed for reaction and should be separated afterwards to reuse the extremely expensive catalyst, which is dissolved in the aqueous phase within the reactor. Extensive experiments have been carried out to determine the drop size distribution (DSD) of the dispersed phase using an endoscopic measurement technique. In this manner the time-dependent evolution of the separation process can be measured quantitatively in a double-wall glass tank as a function of the process conditions. Based on these experimental data, a laminar Euler-Euler simulation based on Computational Fluid Dynamics (CFD) has been set-up. Currently, these simulations are carried using ANSYS-Fluent. Coupled with a population balance model (PBM), such a simulation allows a detailed analysis of the separation process. This can be then directly compared with the experimental data. These comparisons have demonstrated that the choice of the coalescence model is of central importance and strongly impacts the separation process and time. This issue will be investigated in more detail in further CFD simulations.

This work is part of the Collaborative Research Centre "Integrated Chemical Processes in Liquid Multiphase Systems" (TRR 63). The financial support by the German Research Foundation (DFG) is gratefully acknowledged.

## FULLY RESOLVED SIMULATION OF FLUID FLOWS WITH SUSPENDED PARTICLES

Andrea Prosperetti, Johns Hopkins University, USA; A. Sierakowski, Johns Hopkins University, USA; D. Willen, Johns Hopkins University, USA; Y. Wang, Johns Hopkins University, USA

Progress in computational capabilities - and specifically in the realm of massively parallel architectures - render possible the simulation of fully resolved fluid-particle systems. This development holds the promise of major improvements in the physical understanding and modelling of these systems when the particle size is not small and their concentration appreciable. This paper will start with a brief overview of the Physalis method for the solution of the incompressible Navier-Stokes equations in the presence of finite-sized spheres, and it will then describe there examples of fully resolved simulations. The first one is the fluidization of particles for several different values of the density ratio. Among other results, a statistical analysis of the time evolution of tetrads with vertices at the particle centers is carried out. Other aspects include methods to detect concentration waves and clustering. The second situation investigated is a particle the center of which is kept fixed in a turbulent stream, but which is free to rotate. Statistics of its angular velocity and acceleration will be described. Finally, results for the slumping of a particle pile in a fluid will be presented.

\*Work supported by NSF grant CBET 1335965

## HIGH-ORDER MOMENT-CONSERVING METHOD OF CLASSES IN CFD CODE.

Mohamed Ali Jama, Aalto University, Finland; Antonio Buffo, Aalto University, Finland; Alopaeus Ville, Aalto University, Finland

The widely used liquid-liquid extraction (LLE) apparatus of Rotating disc contactor (RDC) is subject to this study. Polydispersity is the case in the technological LLE processes, therefore it is important to model the droplet size distribution that affects the transport phenomena of the system. The population balance equation (PBE) is used to calculate the change in droplet number density. The high-order moment-conserving method of classes (HMMC) developed by Alopaeus et al., 2006 is chosen as the numerical scheme for solving PBE. The HMMC is implemented in the commercial computational fluid dynamics code of FLUENT 16.0 by using UDF codes. Scalar transport equations equal to the number of droplet classes (20) are formed. The source terms cover the complex phenomena of droplet breakage and agglomeration. HMMC, with a low number of classes, have been used efficiently in stand-alone software. HMMC coupling with CFD code is practically unknown. The first results of this study show challenges when solving HMMC with higher order discretization schemes. The purpose of the research is to implement and test HMMC with FLUENT software.

Ville Alopaeus, Marko Laakkonen, Juhani Aittamaa, 2006, Solution of population balances with breakage and agglomeration by high-order moment-conserving method of classes, Chemical Engineering Science, 61, 6732-6752.

## CFD STUDY ON AGGLOMERATION IN A COUNTER-CURRENT SPRAY DRYING TOWER DUE TO PARTICLE-WALL INTERACTIONS

Muzammil Ali, Institute of Particle Science and Engineering, UK; Tariq Mahmud, Institute of Particle Science and Engineering, UK; Peter Heggs, Institute of Particle Science and Engineering, UK; Mojtaba Ghadiri, Institute of Particle Science and Engineering, UK; Andrew Bayly, Institute of Particle Science and Engineering, UK; Luis Martindejuan, Procter and Gamble, UK

Spray drying is one of the most common methods of powder formation in the food, chemical and pharmaceutical industries and involves the drying of an atomised solution or slurry in a spray drying tower. The poly-dispersed droplets come into contact with the hot gas and exchange heat, mass and momentum, resulting in the formation of dried particles. The design and optimisation of spray drying process rely heavily on the past operating experience and the experimental data from laboratory and pilot-scale plants due to the complexity of the process. In detergent spray drying, the mean particle size of the product is significantly larger than the mean droplet size due to agglomeration. One of the main causes of agglomeration is the collision of wet particles with the walls; they get deposited and eventually get entrained as agglomerates. A multiphase CFD modelling study is carried out considering drying of detergent particles in a large counter-current tower utilising a semi-empirical droplet drying model for drying kinetics and agglomeration due to particle-wall interactions. The modelling investigation provides predictions that are in closer agreement with measurements compared to the results from simulations where agglomeration resulting from wall interactions is ignored.

## FULLY RESOLVED NUMERICAL SIMULATION OF INTER-PHASE HEAT TRANSFER IN GAS-SOLID TURBULENT FLOW

Junjie Xia, Institute for Thermal Power Engineering Zhejiang University, China; Kun Luo, Zhejiang University, China; Jianren Fan, Zhejiang University, China

Here begins the body of the abstract.  
 We employ a high-order Immersed Boundary Method (IBM) to study the thermal interaction between entrained solid spherical particles and a turbulent velocity and temperature carrying flow. The sphere diameter ( $d$ ) is about nine times the Kolmogorov scale ( $\eta$ ). The ambient turbulent field is isotropic, and the Taylor microscale Reynolds number is 50. The inflow turbulent intensity held a constant value of 0.1, while the temperature fluctuation varies from 0.1 to 0.5. The particle volume fractions are 0.01 and 0.02, and particle to fluid density ratios are 1.2, 10.0 and 100.0. It is shown that the introduction of particles damp out temperature fluctuation and alter turbulence length scales. It is also shown that the averaged particle mean Nusselt number can be predicted by the well-known Gunn's correlation, while both the mean and the instantaneous Nusselt numbers of each particle deviate significantly from those estimated by the correlations for a steady and uniform ambient. Local Nusselt number distribution reveals that this deviation is partly due to the ambient temperature fluctuation, but largely because of the interaction with thermal wake from upstream.

\* Support from the National Natural Science Foundation of China (Nos. 51136006, 51390491) is gratefully acknowledged.  
 Here ends the body of the abstract.



#### **DNS ANALYSIS OF WEAKLY-COMPRESSIBLE VAPORIZING TURBULENT TWO-PHASE FLOWS**

*Benjamin Duret, Université de Rouen, France; Thibaut Menard, Normandie Université, France; Julien Reveillon, Université de Rouen, France; Francois-Xavier Demoulin, Université de Rouen, France*

Recent improvements in numerical methods allow researchers to perform Direct Numerical Simulations (DNS) of two-phase flows with interface tracking or interface capturing method. Taking into account phase change and describing the interface accurately is still a challenge due to the presence of discontinuities (density, velocity, viscosity ...) across the interface. In this work, a two-phase flows DNS including phase change with (and without) interface regression is used to study the impact of vaporization on the interface, but also its impact on the mixture fraction and scalar dissipation rate statistics in the gas phase. These quantities are of great interest to evaluate the molecular mixing efficiency and, for example, have a direct impact on combustion ignition.

The numerical configuration used is a two-phase flow Homogenous Isotropic Turbulence (HIT), which has been validated in previous work. Since most numerical methods dedicated to phase change are designed with a constant gas density in mind, specific numerical methods have been developed to consider the density variation in the gas phase due to the combination of the vapor production induced by vaporization and the presence of periodic boundary conditions (confinement effect).

#### **SHOCK-INDUCED COLLAPSE OF A SINGLE CAVITATION BUBBLE ATTACHED TO AN ELASTIC-PLASTIC SOLID**

*Cary Turangan, Institute of High Performance Computing, Singapore*

Shock-induced collapse of cavitation bubbles in liquid is an extremely violent event. Its phenomena (high-speed jet impingement and blast wave emissions) are argued to be the main mechanisms of stone fragmentation in lithotripsy treatment and cavitation erosion in fluid machinery. Using a Lagrangian scheme, we simulate the interaction of a stable 40 $\mu$ m-radius air bubble in water with a 60MPa-lithotripter shock whereby the bubble is attached to an aluminium model (2785kg/m<sup>3</sup> density, 26.5GPa shear modulus and 0.3GPa yield strength). Our objective is to understand how jetting collapse cause solid erosion. The result shows that a high pressure region develops near the bubble after the interaction induces the impinging jet formation. The shock reflected by the aluminium further compresses the bubble leading to an elongated bubble shape and a narrow jet whose velocity exceeds 2km/s. Jet impact on the aluminium emits an intense blast wave with pressure > 3GPa. Both phenomena are responsible for a plastic wave seen propagating inside the aluminium, which precedes the growth of indentation crater on the aluminium surface. The collapse timescale is within 0.2 $\mu$ m. By directly coupling the fluid compressibility, multi-phases and material model, the present methodology allows plastic deformation associated with cavitation collapse to be quantified.

#### **A COMPUTATIONAL MODELLING APPROACH TO MINIMIZE PARAMETER SPACE AND GEOMETRY ADAPTABILITY WHEN INVESTIGATING CLEANROOM PARTICULATE LEVELS**

*Nazar Baker, Cork Institute of technology, Ireland; Paul O Sullivan, Cork Institute of technology, Ireland; Ger Kelly, Cork Institute of technology, Ireland*

Recent scientific studies have investigated different variables that influence particulate levels in indoor environments using CFD. These studies often adopt one or more variables and focus on the effect of these variables independently. This paper presents a generalised approach to the development of a CFD model that can be adopted to investigate the variables that influence cleanliness level in cuboid-type grade C cleanrooms. To this end the EU GGMP standard has been adopted to impose constraints on the CFD models variable parameter space. The method involves definition of a reduced parameter state space using both dimensional analysis and Design of Experiment (DOE) and generation of a modular blocked-hexahedral (MBH) mesh to facilitate model adaptability to alternate dimensioning (i.e. varying air diffuser locations). The approach vastly reduces the number of simulations necessary to study the influence of fourteen standard variables on particulate levels. Using, dimensional analysis reduces the parameter space to 9 non-dimensional groups. The DOE matrix produces 16 different scenarios. Investigation of varying inlet/outlet diffuser configurations is greatly simplified using the MBH mesh approach with only 2 geometry re-meshings needed. The non-dimensional groups along with the MBH technique present an efficient, simplified approach to study particulate levels in cuboid-type cleanroom at reduced simulation time and model errors.

#### **DIRECT NUMERICAL SIMULATION WITH HEAT TRANSFER OF GAS-PARTICLE-MULTIPHASE FLOW CONTAINING SHOCK WAVES BY IMMERSED BOUNDARY METHOD**

*Yusuke Mizuno, Tokai University, Japan; Shun Takahashi, Tokai University, Japan; Taku Nonomura, Institute of Space and Astronautical Science, Japan; Takayuki Nagata, Tokai University, Japan; Kota Fukuda, Tokai University, Japan*

The aim of this study is to validate a phenomenon of the flow containing multiple particles and shock waves by the direct numerical simulation. We develop a Cartesian flow solver based on three-dimensional compressible Navier-Stokes equations with an immersed boundary method to investigate the interactions between particles and shock waves. The heat exchange process between particles and flows which takes an important role to develop the characteristic flow structure can be evaluated by the immersed boundary method. In the present method, comprehensive value of the heat flux was obtained by a simple algorithm in which the surface polygons are not necessary but the staircase surface is utilized.

This paper is conducted to analyze the heat exchange on the moving objects. The Reynolds and the shock Mach number are set to be 150 to 300 and 1.5 to 2.0, respectively. We discuss a flow structure and the temperature change of particles. We will confirm that characteristic vortex structure is formed in the wake when multiple particles pass the shock wave.

#### **NUMERICAL INVESTIGATIONS OF SHOCK-INDUCED FLOW PATTERNS IN INHOMOGENEOUS MEDIA**

*Andrzej Nowakowski, University of Sheffield, UK; Franck Nicolleau, University of Sheffield, UK*

This computational study uses the formulation consisting of the pressure non-equilibrium two-phase flow model and the high-resolution Godunov type finite volume method introduced in [Phys.Rev. E 92, 023028 (2015)]. The numerical scheme considers both the non-conservative equations and non-conservative terms that exist in the model which facilitates capturing interfaces separating compressible fluids and the resolution of the baroclinic source of vorticity generation. This model treats each component of the flow with its own equation of state. A shock bubble interaction problem is investigated to evaluate the effect of the Atwood number and shock wave intensity (various Mach numbers) on the interface evolution and on the vorticity generation. When experimental data are available results are successfully validated for the initial development of the complex flows topology. The initial conditions promoting a shock-bubble interaction process are within moderate to high Mach number from 1.5 to 3 and include isolated bubbles of helium, argon, nitrogen, krypton and sulphur hexafluoride. The study is further extended to cover the cases of several isolated bubbles arranged in-line vertically or in a staggered manner. Apart from highlighting the cases characterised by the difference in acoustic impedance the study utilizes the unique features of the present numerical approach to account for the influence of each component heat capacity ratio on the interface deformation.

#### **EFFECTS OF ANISOKINETIC SAMPLING IN STEAM QUALITY MEASUREMENTS FOR STERILIZATION PROCESSES**

*Sonia Serra, Spirax Sarco Ltd, UK*

Dryness is an essential parameter in order to ensure adequate steam conditions in sterilization processes. An adequate cycle requires a dryness level in the range of 95 and 100%. Throttling calorimeters are the standard devices used for assessing dryness, and the conditions for the testing are regulated. However, this method involves a wide range of uncertainty from different sources. An automated and enhanced version of this method can significantly improve the repeatability and reduce some of the uncertainties, but the sampling conditions of steam remains as a fundamental error source.

This work presents experimental and computational results for the effect sampling at a different conditions, quantifying the error in dryness, for both the standard throttling calorimeter and for a new automated version. The deviation of the trajectory of droplets respect to steam that occurs at the probing point is calculated, and the effect of the velocity ratio between sampling and steam line are quantified, and the sensitivity to droplet size is addressed. A computational methodology is proposed and validated against experimental results. The conclusions of this results aim to improve the understanding of the current testing methodology for quality in sterilizers and proposes methods to improve accuracy and repeatability.

## MODULATION OF SPRAY DROPLET NUMBER DENSITY AND SIZE DISTRIBUTION BY AN ACOUSTIC FIELD

Javier Achury, Technische Universität München, Germany; Wolfgang Polifke, Technische Universität München, Germany

Multiple interactions may occur when a poly-disperse spray is immersed in an acoustic field. In the context of spray combustion instabilities, the formation of a number droplet density (NDD) wave and the modulation of the droplet size distribution (DSD) are important effects. An NDD wave, i.e. preferential concentration of droplets in space, may result from size-dependent, one-way momentum coupling between the acoustic field and the spray. The modulation of the DSD, which has been evidenced in the experimental work of Gurubaran and Sujith (AIAA 2008-1046), is then a consequence of the NDD wave formation. In the present work the mechanisms that produce these two effects are simulated and analyzed in depth by means of CFD. The acoustic field is modeled as an oscillating flow and the spray with both Lagrangian (particles mass-point approach) and Eulerian (continuous phase approach) descriptions. The particular Eulerian method used is a presumed density function Method of Moments, which allows to account for the effects of poly-dispersity. Both the Lagrangian and Eulerian models are validated against experimental data for spray dynamics and spray response to an acoustic field.

\*Support from COLCIENCIAS (Republic of Colombia) is gratefully acknowledged.

## MORPHODYNAMICS AROUND A SLENDER CYLINDER UNDER A WAVY BOUNDARY LAYER

Sara Corvaro, Università Politecnica delle Marche, Italy; Maurizio Brocchini, Università Politecnica delle Marche, Italy; Massimo Miozzi, CNR - INSEAN, Italy; Francisco Alves Pereira, CNR - INSEAN, Italy; Carlo Lorenzoni, Università Politecnica delle Marche, Italy; Alessandro Mancinelli, Università Politecnica delle Marche, Italy; Cristina Tondi, Università Politecnica delle Marche, Italy

We analyze the interaction of flow, sediment and a slender cylinder due to the boundary layer that develops under the action of propagating water waves. We focus on two highly correlated topics: the evolution of the bed surface due to the flow-induced sediment-particle dynamics and the development of coherent flow structures that are responsible for the sediment dynamics close to the cylinder's base. A couple of digital CCD cameras provides the stereoscopic view of the sediment evolution. The final bed morphology is measured by using a Laser Distancemeter. The in-phase analysis of the differential sediment removal and deposition around the cylinder identifies the position of the areas that are more subject to the action of the coherent structures, thus mapping the flow/sediment interaction magnitude as a function of the removal efficiency. The flow kinematics and the sediment motion is reconstructed by means of a PTV Technique. For a better characterization of the nearbed dynamics we also analyzed the vorticity field, with particular interest in the coherent structures generated in the bottom boundary layer. Statistics of flow structures and sediment particles path are extracted at different wave characteristics and results are compared for different wave forcing parameters.

## FLUID DYNAMICS OF SELF-PROPELLED DROPLET JUMPING ON SUPERHYDROPHOBIC SURFACES

Patric Figueiredo, Germany; Joram Wasserfall, RWTH Aachen University, Germany; Philipp Pischke, RWTH Aachen, Germany; Reinhold Kneer, RWTH Aachen University, Germany

The fluid dynamics of coalescing microdroplets on superhydrophobic surfaces is governed by a subtle balance of surface forces, kinetic energy and viscous dissipation. For vanishing viscous forces, coalescing droplets may jump off the superhydrophobic surface, promising various applications such as anti-icing coatings or non-wetting condensers. In this study, the fluid dynamics of droplet jumping is investigated with finite volume computations, presenting an OpenFOAM solver specifically designed towards the simulation of interfacial flows with high contact angles.

Based on a variation of the Ohnesorge numbers and the static contact angles, the present study elaborates on the different modes of microdroplet coalescence. Aside from droplet jumping and adherence regimes, results indicate the presence of a bubble-entrapment regime, where a bubble is trapped under the capillary bridge, a phenomenon known from droplet impact on hydrophobic surfaces. The study shows stability limits between droplet jumping and adherence, and provides a regime diagram for different modes of microdroplet coalescence.

Results are validated by comparisons to dynamic test cases, such as eigenfrequencies and decay rates of oscillating droplets and capillary waves, providing a dependable basis for further investigations.

## DIRECT NUMERICAL SIMULATION OF MULTI-COMPONENT DROPLET INTERACTIONS

Karin Schlottke, University of Stuttgart, Germany; Christian Meister, University of Stuttgart, Germany; Grzegorz Karch, University of Stuttgart, Germany; Filip Sadlo, Heidelberg University, Germany; Thomas Ertl, University of Stuttgart, Germany; Bernhard Weigand, Universität Stuttgart, Germany

Multi-component problems are part of many technical applications, such as combustion engines, where we observe mixing processes of fuel droplets and the oil wall-film. The presented investigation covers three-dimensional numerical simulations of multi-component droplet interactions using the direct numerical simulation code FS3D. This in-house code solves the incompressible Navier-Stokes equations and employs the Volume-of-Fluid (VOF) method in order to account for multiple phases. The convective transport is performed based on the piecewise linear interface calculation (PLIC) of the surface. In order to account for multiple species in the liquid phase, an additional VOF-variable is introduced for each species. It represents the volume fraction of the respective species within the liquid phase of each cell. Using this approach, it was possible to demonstrate the applicability of FS3D to the computation of the impact morphology and splashing/deposition limit for two-component droplet wall-film interactions. For the visualization of the obtained results, an approach based on the PLIC reconstruction was used that obtains the interface by isosurface extraction from the first-order Taylor approximation of the VOF-field. The benefits of this approach are also highlighted. With this investigation, we hope to better understand how the fuel interacts with the oil wall-film and hence to help reduce the contamination of the oil film.

\*Support from the Stuttgart Research Centre for Simulation Technology is gratefully acknowledged

## THREE-PHASE ROUGHNESS EFFECTS ON SUPERHYDROPHOBIC SURFACES FOR TURBULENT DRAG REDUCTION

Kevin Golovin, University of Michigan, USA; James W. Gose, University of Michigan, USA; Marc Perlin, University of Michigan, USA; Steven L. Ceccio, University of Michigan, USA; Anish Tuteja, University of Michigan, USA

Superhydrophobic surfaces have consistently shown meaningful drag reduction in laminar flows. Utilizing the same surfaces to produce turbulent drag reduction has produced mixed results. One critical parameter is the stability and deformation of the three-phase air-water-solid interface that gives superhydrophobic surfaces their amazing, non-wetting properties. In this work we examine the roughness effects of the three-phase region in fully developed turbulent boundary layer flow. By screening a large number of state-of-the-art superhydrophobic surfaces, produced using a range of chemical and physical techniques, we are able to predict the expected drag due to asperity roughness and thereby design the optimal surface for turbulent drag reduction. We develop a simple scaling using high pressure droplet studies in conjunction with profilometry that aids in the rational design of superhydrophobic surfaces with the application of high Reynolds number turbulent drag reduction. We also experimentally alter the solid-liquid-vapor interface through applied or reduced tunnel pressure, thereby shifting the local height of the meniscus. Overall, our studies confirm that turbulent drag reduction is possible using superhydrophobic surfaces, but also that the correct design of the local surface roughness is highly dependent on the location of the air-water interface.

## ON THE INTERACTION BETWEEN WAVES AND A MODULAR OSCILLATING WAVE SURGE CONVERTER

Thomas Abadie, Institut de Mécanique des Fluides de Toulouse (IMFT), France; Frederic Dias, University College Dublin, Ireland

Wave Energy Converters (WECs) are a new class of offshore structures, which aim at collecting energy from the waves in highly energetic locations for approximately 25 years. The design of these machines therefore requires improving the knowledge on wave-structure interaction, as well as the wave loads exerted on such devices under extreme sea states.

We focus on a modular Oscillating Wave Surge Converter (OWSC), which is an alternative to Oyster, the OWSC developed by Aquamarine Power Ltd. Oyster consists of a wide buoyant flap, located in 10 to 15 meters water depth, which pierces the free surface and oscillates back and forth under the action of incoming waves. The dynamics of Oyster and wave impacts on this device have been studied over the past few years and a new concept, where several thinner modules replace the unique wide flap, has been introduced recently.

3D numerical simulations based on a finite volume Navier-Stokes flow solver are performed to study the response of the modules depending on the wave excitation. The free surface is captured thanks to a Volume of Fluid method and the motion of the solid bodies is modeled through an Immersed Boundary Method. The amplitude of oscillation of the inner modules is larger than that of the side ones and the interactions between the different modules depending on wave characteristics, as well as geometrical parameters are studied.

## ENERGY-BASED MODEL FOR THE PREDICTION OF MINI-CHANNELS DRY-OUT CONDITION

Niccolo Giannetti, Waseda University, Japan; Kiyoshi Saito, Waseda University, Japan; Seiichi Yamaguchi, Waseda University, Japan

The complexity of heat transfer by means of mini-channels grows when two phases coexist. In particular, their performance is strongly affected by the local flow configuration and the accurate prediction of partial or complete dry-out of the exchange surface becomes of critical importance. This, in turn, depends on a variety of parameters representing the dominant effects at play. Once the fluid properties are defined, experiments highlight that when the diameter falls below Taylor instability length of the circulating fluid, specific hysteresis phenomena of the fluid wettability occur. An exhaustive model of these devices is required to include wettability effects for an optimized design and accurate control of the system. In order to capture this phenomenon inside mini-channels, this study develops a criterion for predicting the occurrence of the channel dry-out and a method to estimate the wetting behavior. Since, at this characteristic scale, surface tension and inertia are the dominant hydrodynamic effects, a stable condition can be identified by minimizing the free energy (surface tension and kinetics) of the flow system when annular and slug configurations are compared. Theoretical results are compared with experiments. This model results to be a useful predicting tool.

## THE ROLE OF VISCOSITY STRATIFICATION ON THE GLOBAL STABILITY OF TWO-PHASE JETS

Otti Tammisola, University of Nottingham, UK; Jean-Christophe Loiseau, Royal Institute of Technology (KTH), Sweden; Luca Brandt, KTH Royal Institute of Technology, Sweden

Two-phase flows are encountered in numerous industrial applications from atomization to microfluidics. Though global stability analyses have proven helpful in numerous single-phase flow situations to get a better understanding of the physics, such approach still is scarcely used in the two-phase flow community. Tammisola et al. (JFM 2012) used global stability to investigate non-parallel immiscible co-flowing jets and wakes with constant viscosity and density. It was found that surface tension has a counter-intuitive destabilizing effect on these flows.

The present work relaxes the uniform viscosity assumption and aims at shedding some light on the interplay between viscosity stratification and the surface tension-driven global instability for co-flowing jets. It is well known that one of the most common mechanism for interface instability is the Yih mechanism resulting from viscosity stratification which leads to net work being done by the perturbation velocity and stress at the interface separating the two phases. Our results indicate that the global stability of the flow is governed by a viscosity-modified effective Reynolds number. These stability analyses are performed using a new algorithm based on a time-stepper formulation of the linear problem. Such approach can be readily implemented within any existing code with relatively little effort thus allowing the investigation of numerous complex flow situations.

## MODELLING OF GAS-LIQUID, TWO-PHASE FLOW IN POROUS MEDIA AND CHANNELS OF A PEM WATER ELECTROLYSIS CELL USING THE EULER-EULER FRAMEWORK OF OPENFOAM

Anders Christian Olesen, Aalborg University, Denmark; Soren Knudsen Kaer, Aalborg University, Denmark

In PEM water electrolysis, a gaseous oxygen phase inherently develops and prevails the liquid water phase from uniformly distributing throughout the catalytic porous layer of the anode. An even distribution of liquid water not only ensures good mass transport characteristics of the reactant specie, it also secures an even heat removal of the cell.

In a previous research effort, the gas-liquid, two-phase flow was investigated in the commercial CFD framework of ANSYS CFX using the Euler-Euler model. Particularly, the treatment of gas-liquid flow in a porous medium domain subject to capillary pressure was shown to push the limits of the codes capabilities. In order to improve simulation stability and time, a new model is developed in the open source CFD software OpenFOAM. The customizability of this code not only allows for specific relaxation strategies, it also permits the implementation of various boundary conditions at porous-porous or porous-fluid interfaces found in the membrane electrode assembly. Particularly the treatment of a discontinuous phase volume fraction at the porous-porous interface, and the merging of two pressures fields into a single dominating at the porous-fluid interface influence the convergence stability of the solver.

\* Support from the Innovation Foundation Denmark under the project e-STORE is gratefully acknowledged.

## TOWARDS A 3D GLOBAL LINEAR STABILITY ANALYSIS FOR BUOYANT FLUID PARTICLES

Johannes Kromer, TU Darmstadt, Germany; Dieter Bothe, Technische Universität Darmstadt, Germany

We introduce an algorithm for the stability analysis of fluid particles, rising freely in an ambient liquid. Albert et al. (doi:10.1017/jfm.2014.57) introduced the basic concept, resorting to Chiba's method (1998, J. Jpn. Soc. Comput. Fluid Dyn. vol. 7), for the case of 2D falling films. The dynamical system is governed by the isothermal incompressible Navier-Stokes equations with constant material properties. The associated state vector contains the velocity field and interface position. The evolution equation is linearized around a steady state, which is perturbed by a set of orthonormal deviations. The time evolutions are computed numerically using the VOF-code FS3D, superposition of the evolved states allows to assess linear stability. Due to the moving interface, linear algebra cannot be performed on the state space. Instead, the velocity fields are transformed appropriately. The mathematical concept as well as the developed algorithms will be discussed. Furthermore, the computations of volume fractions from spherical harmonics will be addressed, being also of independent interest. The work of J. Kromer is supported by the Excellence Initiative of the German Federal and State Governments and the Graduate School of Computational Engineering at Technische Universität Darmstadt.

## NUMERICAL STUDY ON EFFECTS OF ORIENTATION AND GRAVITY ON TWO-PHASE FLOW OF MINICHANNEL OF PEM FUEL CELLS

Ashrafi Moosa, K.N. Toosi University of Technology, Iran; Shams Mehrzad, Khaje Nasir Toosi university of technology, Iran; Ali Bozorgnezhad, K.N. Toosi University of Technology, Iran; Goodarz Ahmadi, Clarkson University, USA

In this study, effects of gravity and orientation of PEM fuel cell on two-phase flow of minichannel of cathode side were investigated. A three-dimensional numerical model, based on the volume of fluid method (VOF), was employed to simulate gas-liquid flow in the single-serpentine minichannel of a PEM fuel cell with a 1.0mm×1.5mm rectangular cross section and a 51.5 mm length (21 passes). Water enters the minichannel from the microstructure of porous gas diffusion layer (GDL). The hydrophilic walls and hydrophobic GDL were considered. Surface tension and wall adhesion forces were applied. The orientations of channel were horizontal and vertical. The position of gas inlet and outlet varied from down to up. The effects of gravity and orientation of flow-field were studied using water coverage ratio in the minichannels. The results showed that the vertical orientation had more water coverage ratio than horizontal one. In both horizontal and vertical positions, the water coverage ratio of channels was increased by changing the inlet position from up to down. Unsteady flow patterns of gas-liquid two-phase flow in the minichannels were investigated.

## DYNAMICS OF GAS-LIQUID SEGMENTED FLOW IN POROUS MEDIA: MONOLITHIC OPEN CELL SOLID FOAMS VERSUS DENSE PACKED BED

Marion Serres, Laboratoire de Physique/CNRS-ENS de Lyon-Université de Lyon, France; Marie-Line Zanotta, LGPC/CNRS-CPE Lyon-Université de Lyon, France; Frederic Borette, LGPC/CNRS-CPE Lyon-Université de Lyon, France; Regis Philippe, LGPC/CNRS-CPE Lyon-Université de Lyon, France; Valerie Vidal, Laboratoire de Physique/CNRS-ENS de Lyon-Université de Lyon, France

Multiphase flow through porous media is commonly encountered in natural processes like enhanced oil recovery, but also in industrial chemical applications like catalytic reactions. This multidisciplinary topic is characterized by a large variety of porous media. In this study, we focus on the hydrodynamics of a gas-liquid flow crossing two different porous media: (1) a monolithic open cell solid foam, highly porous and permeable, recently highlighted for catalytic reactors optimization and (2) a classical dense packed bed of spherical glass particles, a reference porous medium in chemical and in reservoir engineering. Air-ethanol segmented flow is investigated in a millimetric channel filled either with the monolithic foam or with the dense packed bed and operated at different velocities, gas / liquid flow-rate ratios and different fluid-solid interactions (porosity, permeability and wettability). The global hydrodynamics is first studied. We model and discuss the retention time distribution and the dispersive behavior of the liquid phase. Then, the local hydrodynamics is investigated through direct visualization and image analysis. We develop a spatio-temporal frequency analysis of the segmented flow along the cell. A dimensionless number is proposed to parameterize the transitions among the observed regimes. Overall, this study contributes to the elucidation of a possible confinement impact on two-phase contacting efficiency with a non-conventional porous medium.

## LES OF A REACTIVE, MULTI-PHASE FLOW IN THE INJECTOR NEAR-FIELD OF AN ENTRAINED-FLOW GASIFIER

Georg Eckel, German Aerospace Center (DLR), Germany; Alexander Saenger, Karlsruhe Institute of Technology (KIT), Germany; Christian Hotz, Karlsruhe Institute of Technology (KIT), Germany; Trupti Kathrotia, German Aerospace Center (DLR), Germany; Michael Rachner, German Aerospace Center (DLR), Germany; Patrick Le Clercq, German Aerospace Center (DLR), Germany; Manfred Aigner, German Aerospace Center (DLR), Germany

The rising share of renewable energy systems to the global energy supply increases the contribution of highly fluctuating energy sources. Load-flexible energy storage and conversion systems can compensate for these fluctuations. In addition, the finiteness of high-grade fossil fuel resources leads to a growing demand in fuel-flexible and efficient energy conversion systems coping with low-grade fuels like waste- or biomass-based fuels. Entrained flow gasification is a promising process for converting low-grade feedstock, e.g. highly viscous slurries and suspensions with a significant content of solid particles, to high quality fuels. A major scientific challenge is the prediction of the physical and chemical phenomena occurring in such high-temperature, high-pressure multi-phase systems. To reduce complexity, this study focuses on a two-phase (gas-liquid) flow system with a model fuel (ethylene glycol) under atmospheric conditions. The unsteady flow and the chemical conversion in the gasifier are investigated using Large Eddy Simulations with a detailed chemistry solver including 44 individual species and 329 chemical reactions. The dispersed phase is solved by Lagrangian Tracking. The predictive capabilities of the numerical tool are evaluated by comparison to measurements performed on a research entrained flow gasifier. The gasifier is also analyzed in terms of flow, species and temperature field as well as droplet dispersion to gain insight into rich-burn and gasification.

## NUMERICAL STUDY ON DROPLET EVAPORATION AND AUTO-IGNITION OF HYPERGOLIC PROPELLANTS

Hiroumi Tani, Japan Aerospace Exploration Agency, Japan; Yutaka Umemura, Japan Aerospace Exploration Agency, Japan; Hiroshi Terashima, University of Tokyo, Japan; Nozomu Kanno, Meijo University, Japan; Mitsuo Koshi, Yokohama National University, Japan

Liquid fuel and oxidizer of space propulsion engines often have highly reactive characteristics which mean fuel and oxidizer spontaneously auto-ignite when they come into contact with each other in combustors. To control the timing of the ignition and consumption rate of such reactive liquids, the phase change and chemical reactions near the liquid-liquid and liquid-gas interfaces should be understood. Lagrangian droplet-tracking method, which is often employed for spray combustion of industrial fuels, cannot accurately predict the vaporization and auto-ignition of reactive droplets. Thus, the present study developed a CFD method, by coupling an interface tracking method with a phase change model and chemical reaction model, to explore the reactive flows near the liquid-gas interface of reactive droplets. The auto-ignition processes and the interaction between chemical reactions and evaporation of reactive droplets will be discussed. Furthermore, the effects of the droplet size and ambient pressure upon the ignition delay time and burning rate will be presented to develop or modify the droplet evaporation models of lagrangian droplet-tracking methods.

## TURBULENT CHANNEL FLOW WITH SUSPENDED BI-DISPERSE FINITE SIZE NEUTRALLY BUOYANT SPHERES

Francesco Picano, University of Padova, Italy; Iman Lashgari, KTH Mechanics, Sweden; Pedro Costa, Delft University of Technology, The Netherlands; Wim-Paul Breugem, Delft University of Technology, The Netherlands; Luca Brandt, KTH Royal Institute of Technology, Sweden

Particle suspensions are usually found in geophysical and industrial flows such as pyroclastic and planetary flows, fluidized bed and hopper dredger. In those applications particles are often large with respect to the smallest scale in the flow and have a wide range of sizes. In the present work we study a turbulent channel with a suspension of finite size rigid spheres of two different sizes,  $2h/d=20$  and  $30$  where "h" is half channel height and "d" is the particle diameter. We perform DNS at bulk Reynolds number  $Re_b = 5600$  where the box size and number of grid points in wallnormal, spanwise and streamwise directions are  $2h^*3h^*6h$  and  $480^*720^*1440$ . An Immersed Boundary Method is employed to mimic the presence of finite size particles. The interactions below the mesh size are treated by lubrication correction and soft sphere collision model. Five different simulations have been performed fixing the total particle volume fraction at  $\Phi=0.2$  and partitioning the ratio between small and large particles, 0%-100%, 25%-75%, 50%-50%, 75%-25% and 100%-0%. The preliminary results regarding the dispersed phase show that the small and large particles tend to differently distribute across the channel and especially in the near wall region. The relative particle velocity and the collision kernel is strongly influenced by the different percentage of particle sizes. A wider and deeper analysis will be presented in the final paper.

## INTERACTION OF FLUID DYNAMICS AND CHEMICAL KINETICS IN A THREE-PHASE DEHYDROGENATION REACTION

Norbert Heublein, Technische Universität München, Germany; Thomas Sattelmayer, Technische Universität München, Germany

We present our experimental work towards the phenomenological understanding and the quantitative modeling of a three-phase system exhibiting complex interactions between fluid dynamics and chemical kinetics.

The storage of hydrogen in so called "Liquid Organic Hydrogen Carriers" (LOHC) is a promising approach to enable a future energy supply based on renewable energy sources. The LOHC-technology allows the safe and durable storage of about 6 wt% of hydrogen. Because of the favorable properties of the liquid carrier substance, hydrogen stored in LOHC can be distributed using the existing infrastructure of hydrocarbon fuels including pipelines, ships and trucks. At the location of energy demand, the hydrogen can be released from the liquid carrier substance in a catalytic dehydrogenation reaction and be employed as fuel for internal combustion engines or fuel cells. Although in theory, the LOHC-technology offers numerous advantages over the established approaches of gaseous or cryogenic hydrogen storage, a broad implementation is still hindered by severe problems in practice. Most of these problems are caused by the process of dehydrogenation, where large amounts of gaseous hydrogen are released from the liquid carrier contacting the surface of a solid catalyst. In this work, we present our experimental investigation of the interaction between the fluid dynamics in a reacting three-phase system and the observed kinetics of the catalytic hydrogen release.

## FLOW TOPOLOGY AND ATTRACTORS FOR THE MOTION OF FINITE-SIZE PARTICLES IN A THREE-DIMENSIONAL STEADY CAVITY FLOW

Hendrik Kuhlmann, TU Wien, Austria; Francesco Romano, TU Wien, Austria; Stefan Albensoeder, Carl von Ossietzky Universität Oldenburg, Germany

The structure of spanwise periodic steady three-dimensional cavity flow driven by the steady motion in opposite directions of two facing walls is investigated numerically in terms of the coexistence of regular and chaotic streamlines. Based on this flow template the one-way coupled motion of spherical particles is calculated. Particular attention is paid to modeling the particle motion near the boundaries where the finite size of the particles cannot be ignored. In order to model the particle interaction with the moving walls an inelastic collision model is employed. The model is aimed at providing a first approximation to the lubrication effects between particle and wall when the distance between the particle's centroid and the moving wall is of the order of the particle radius. The importance of the moving-wall effect on the particle motion and the creation of particle attractors is demonstrated and discussed.

## DNS OF TURBULENT BOUNDARY LAYER WITH FULLY RESOLVED PARTICLES AT LOW VOLUME FRACTION

Fan Wu, Zhejiang University, China; Kun Luo, Zhejiang University, China; Jianren Fan, Zhejiang University, China

In the present work, we have conducted a direct numerical simulation of dilute particulate flow in a boundary layer in the turbulent regime, considering thousands of finite-size rigid particles with multi-direct forcing immersed boundary method to resolve the interfaces. The particles have a diameter of approximately 11.3 wall units, a density of 3.3 times the fluid density and a solid volume fraction of 10<sup>-3</sup>. The inlet stream-wise Reynolds number according to the momentum thickness of the boundary layer is 800. To the authors' knowledge, this is the first fully resolved DNS study of the particles in a turbulent boundary layer. The results regarding mean and fluctuating statistics of the boundary layer containing the particle phase are compared with that of the single phase. Due to much smaller scale than other works with point-particle assumption describing the solid phase, no evidence of significant formation of particle clusters is found. In the latter sections, the particle distribution and movement characteristics are discussed.

Acknowledgement: Financial supports from the Key Program of the National Natural Science Foundation of China (Grant No. 51136006)

## FULLY RESOLVED SIMULATIONS OF SUSPENSION OF SPHERICAL PARTICLES IN SHEAR FLOWS

Mona Rahmani, Stanford University, USA; Anthony Wachs, University of British Columbia, Canada

In this study, a Distributed Lagrangian Multiplier/ Fictitious Domain (DLM/FD) method is used to study the non-Brownian suspension of spherical particles in a plane Couette shear flow in a Newtonian fluid. The range of particle Reynolds numbers studied is between 0.002 and 20, and the range of volume fractions is between 0.05 and 0.4. The total normal and shear stresses of the suspension are the sum of the contributions of the fluid and solid phases. The particle stresses, on the microscopic scale, are the result of hydrodynamic and particle contact stresses. Here, we examine the stresses of each phase at macro- and micro-scale levels. We compare the results of our simulations to the suspension balance models (SBM), e.g. that of Nott & Brady (1994) and empirical models, such as the Krieger-Dougherty model. We extend the results to the inertial range of particle motions, where the Stokesian dynamics do not hold. By adding the inertial term to the SBM a satisfactory comparison is obtained between our numerical results and this model. We also study the shear-induced diffusion of the particles, which reveals an anisotropy in the direction the suspension is sheared compared to the transverse directions.

## NUMERICAL STUDY OF MASS TRANSFER OF A RISING BUBBLE WITH BOUNDARY LAYER APPROACH

Shafiu Islam, Eindhoven University of Technology, The Netherlands; J. A. M (hans) Kuipers, Eindhoven University of Technology, The Netherlands; Niels G. Deen, Eindhoven University of Technology, The Netherlands

In this work we study the mass transfer from bubbles rising in a liquid. In the past decades, many researchers have predicted gas-liquid mass transfer using fixed grid approaches. As the mass boundary layer is very thin compared to the momentum boundary layer, fixed grid approaches require very fine grids, which makes the computation very expensive. To overcome this issue, a boundary layer approach was embedded in our front tracking model. We show simulation results of well-defined limiting cases to demonstrate the merits of this method.

Keywords: Gas-liquid mass transfer, Boundary layer, Front tracking, Fixed grid approach.

## EFFECT OF GAS PROPERTIES ON THE CHARACTERISTICS OF A BUBBLE COLUMN EQUIPPED WITH FINE POROUS SPARGER.

Ariadni Chatzidafni, Aristotle University of Thessaloniki, Greece; Agathoklis Passos, Aristotle University of Thessaloniki, Greece; Spiros Paras, Aristotle University of Thessaloniki, Greece; Aikaterini Mouza, Aristotle University of Thessaloniki, Greece

The effect of various parameters (i.e., liquid physical properties, sparger diameter and pore size, column diameter) on the characteristics of a bubble column equipped with a porous sparger has been systematically studied in the last decade. However, there is still lack of experimental studies concerning the effect of gas properties on the bubble column characteristics. The aim of this project is to experimentally study the effect of gas density on the initial bubble size distribution, the transition point and gas hold-up of a bubble column equipped with a fine pore sparger. Various gases (e.g. CO<sub>2</sub>, helium) covering a range of physical property values are employed, while the liquid phase is water and aqueous glycerin solutions.

A fast video recording technique is used both for the visual observations of the phenomena occurring onto the sparger and for the bubble size measurements. The validity of previously proposed correlations concerning the prediction of the initial mean Sauter diameter of bubbles, the transition point and gas hold-up are tested. The experiments, which are still in progress, show that the mechanisms of bubble formation as well as the initial bubble size distribution depend on the gas density and flow rate.

## EFFECT OF A CONCENTRATION GAS BUBBLES IN THE NEAR-WALL LIQUID FLOW ON THE DRAG REDUCTION

Aleksei Evseev, Kutateladze's Institute of Thermophysics, Russian Federation; Leonid Maltsev, Kutateladze's Institute of Thermophysics, Russian Federation

Currently microbubble gas saturation of a near-wall liquid flow is recognized as one of the most promising and cost-effective ways to reducing the turbulent friction of ships, having minimal impact on the environment. Our experiments have confirmed the earlier results that the drag reduction may reach 80% (or even more) while the concentration of microbubbles in the near-wall region (which is a very important parameter) may be more than 70%.

The report presents new data on the microbubble method of friction reduction. The paper presents the results on bubbles concentration measurement employing fiber-optical sensor, as well as measurements of a local friction using "floating wall" type sensor. The distribution of the bubble concentration was measured at different flow velocities and gas flow rates on the top and bottom walls of the horizontal channel at various distances downstream of the gas generator. The gas was injected into the boundary layer through the porous plate with a pore size of 20 microns. The paper shows a correlation between a local turbulent drag reduction and the concentration of the gas phase in the near-wall flow region. The measuring results are compared with the known data of physical and mathematical modeling of the phenomenon, and the main processes of the mechanism of friction reduction using gas-saturation of the turbulent boundary layer are discussed.

## MANIPULATING OF MICRO-BUBBLES USING ULTRASONIC WAVE PAIRS

Fubing Bao, China Jiliang University, China; Sujuan Xiang, China Jiliang University, China; Yuebing Wang, China Jiliang University, China

Here begins the body of the abstract.

Micro-bubbles have widespread applications in medicine, environmental, mining, chemical, nuclear energy and other fields. Recently, the interaction of acoustic fields with micro-bubbles in liquids is of interest to a wide range of people. Micro-bubbles with diameters from 50(m to 150(m are generated in the present study using a microfluidic flow-focusing device, by changing the device geometry and control parameters. Ultrasonic generator pairs are then applied in a tank to emit ultrasonic wave which preform to control the movement of a single micro-bubble in the water. In this study, we focus on discovering the kinetic characteristic of the micro-bubble in the action of the ultrasonic wave. The motion trajectory of a single bubble could be precisely controlled by adjusting the frequency and phase position of the ultrasonic wave pair. This technology may provide new potential opportunities to targeted therapeutics.

\* Support from the National Natural Science Foundation of China (grant number 11372298) is gratefully acknowledged.

Here ends the body of the abstract.

## MEASUREMENT OF BUBBLE VELOCITY, SHAPE, AND SIZE DISTRIBUTION IN AN UNDIVIDED ELECTROCHLORINATION CELL BY TELECENTRIC DIRECT IMAGE METHOD

Nikolaj Andersen, Aalborg University, Denmark; Lau Hedensted, Aalborg University, Denmark; Rodica-Elisabeta Stroe, Aalborg University, Denmark

The determination of velocity, shape, and size distribution of gas bubbles in a single undivided electrochlorination cell is the subject of this study. The production of hypochlorite by electrochlorination has gained more attention in recent time due to potential hazards of handling of large volumes of liquid chlorine for water disinfection. Still, the combination of electrochemistry, mass transfer, and two-phase flow are not fully understood. Hence, accurate measurements of such electrochlorination processes are desired so these can be modelled accurately in order to obtain more insight into the combined phenomena influencing the performance.

The electrochlorination cell is equipped with two titanium electrodes and a dilute sodium chloride electrolyte solution. The cell is operated at current densities of 25 mA/cm<sup>2</sup> to 100 mA/cm<sup>2</sup>, corresponding to voltages of approximately 1 V to 3.5 V. Measurements are conducted using a Telecentric Direct Image Method (TDIM); where a digital CCD camera, equipped with a telecentric lens and LED background lighting, is used to acquire images of the gas bubbles. Subsequently, particle tracking and edge detection algorithms, integrated in NI Vision and MATLAB, are applied to the images to determine the velocity, shape, and size of the bubbles.

## THE FORMATION OF BUBBLES FROM THE CAPILLARY IN A LIQUID FLOW

Pavel Lobanov, Kutateladze institute of thermophysics, Russian Federation;  
Oleg Kashinsky, Kutateladze institute of thermophysics, Russian Federation;  
Aleksandr Chinak, Kutateladze institute of thermophysics, Russian Federation;  
Maxim Vorobyev, Kutateladze institute of thermophysics, Russian Federation

Bubbly gas-liquid flows are widely used in various engineering applications. It is possible to control heat and mass transfer characteristics of the apparatus by changing the bubble size because this changes the main parameters of two-phase flows such as interfacial area, relative slip velocity, gas phase distribution over cross section. Therefore the study of bubble formation process is important to predict operating properties of the equipment.

The purpose of this paper is to study the process of bubble detachment from a single capillary in the liquid flow. The dependencies of detachment bubble diameter on liquid velocity and gas flow rate through a capillary placed in the liquid flow in a channel were obtained. An estimate of bubble detachment frequency was made. The effect of bubble detachment regime on the distribution of bubble size in the flow is discussed.

## NUMERICAL INVESTIGATION ON MICROBUBBLE DYNAMIC EVOLUTION AND TURBULENCE MODULATION IN A HORIZONTAL CHANNEL BUBBLY FLOW

Mingjun Pang, Changzhou University, China; Jinjia Wei, Xi'an Jiaotong University, China

Bubbly flows exist extensively in natural and industrial fields, and thus it is very important for understanding some physical phenomena to deeply investigate hydrodynamic characteristics of them. Here, the developed Euler-Lagrange two-way method was used to study microbubble dynamic evolution and influences of microbubbles on the spanwise vortices and the turbulence field of liquid in a horizontal channel. The liquid turbulence field was solved by direct numerical simulations and the microbubble motion was tracked by Newtonian equations. For the present investigation, the buoyant force of microbubbles, on both sides of the channel, points at the corresponding wall so as to understand how microbubbles influence the evolution of the spanwise vortices when they move towards the corresponding wall. The present studies shows that most microbubbles accumulate in the region very close to the walls, and a minority of microbubbles hesitate in the region near the walls; the spanwise vortices of the liquid phase slightly increase in the region away from the walls, leading to the increase in the velocity fluctuation intensity and Reynolds shear stress and the decrease in the streamwise mean velocity of liquid. The influence of microbubbles on the spanwise vortices is an individual behavior of the single microbubbles and is not a collective one of a microbubble layer.

\*Support from grant NSFC Fund No. 51376026 is gratefully acknowledged.

## BENCHMARKING CFD AND NEW MECHANISTIC APPROACH FOR ENABLING SCALE-UP OF MULTIPHASE REACTORS VIA SOPHISTICATED MEASUREMENT AND COMPUTING TECHNIQUES

Muthanna Al-Dahhan, Missouri Univ of Science & Technology, USA

In our laboratory advanced measurement techniques, facilities and scale up methodologies have been developed, verified and implemented on various complex multiphase reactors and flow systems with internals and without internals. These have been used to benchmark CFD simulations for evaluation and validation and for enabling our new mechanistic scale up methodologies we recently developed. In this presentation we will give an overview about our recent findings and about some of these measurement techniques which are: I) noninvasive based on radiation: radioactive particle tracking (RPT), dual source gamma ray tomography (DSCT), gamma ray densitometry (GRD) for 3D flow field, velocity and turbulent parameters, phases distribution and flow pattern identification measurements, and II) invasive/noninvasive not based on radiation: 4-point optical probe for bubble dynamics, heat transfer probe, combination of bubble dynamics and heat transfer probe, optical probes for solids dynamics that measure simultaneously solids velocity and holdups and their fluctuations, integration of hot wire anemometry and heat transfer probe, gas tracer dynamics, optical probe for local mass transfer, gas tracer technique for global mass transfer, optical probe for liquid velocity distribution in packed beds, pressure transducers, and others. These techniques are augmented with sophisticated mathematical algorithms and programs that have been developed in our laboratory for data gathering, processing and image reconstruction.

## EXPERIMENTAL INVESTIGATION OF PRESSURE DROP OF COUNTER CURRENT AIR-WATER FLOW IN A VERTICAL COLUMN

Maryam Mashayekhpour, University of Kashan, Iran; Ebrahim Nemati Lay, University of Kashan, Iran

In this present work, the experimental pressure drop in a counter current air-water flow in a vertical column has been measured. For this reason, an experimental work is done in a vertical bubble column with 0.1 m diameter and 3 m height. The experimental study is carried out in upward flow of air and a downward flow of water at ambient temperature. The air and water superficial velocities are in ranges of 0.4 to 2.12 cm/s and 0.64 cm/s to 2.72 cm/s, respectively. The experimental data of pressure drop is obtained and discussed. This study is clarified the effect of different flow regimes due to variation of gas and liquid superficial velocity on the pressure drop in bubble column.

## EFFECTS OF PHYSICAL & GEOMETRICAL PARAMETERS ON COALESCENCE OF TWO BUBBLES

Sumedh Yadav, Indian Institute of Technology Kharagpur, India; Aniruddhe Pradhan, Indian Institute of Technology Kharagpur, India; Prasanta Kumar Das, Indian Institute of Technology Kharagpur, India

Bubble interactions have significant impact on the shape and motion of bubbles, and therefore the dynamics of bubbles in a swarm may be considerably different from that of an isolated bubble. This research presents a numerical study of two bubble interactions using a novel lattice Boltzmann method (LBM). A coupled Navier-Stokes/Cahn-Hilliard model is used for the simulations. The numerical approach is briefly illustrated, and the code is validated for bubble shapes for different  $Mo$  and  $Eo$  numbers using shape regimes literature. Then the method is applied to simulate the interaction between two bubbles during their buoyant rise. A pair of bubbles with spherical shapes is simulated under different configurations, wall conditions and non-dimensional parameters. Both attractive and repulsive interactions are observed in the simulations depending on the relative position, wall condition etc. Finally effects of these parameters on bubble coalescence is discussed.

## MEASUREMENTS OF LOCAL HEAT TRANSFER IN SUBCOOLED POOL BOILING USING INFRARED THERMOMETRY

Jaehoon Jung, Republic of Korea Atomic Energy Research Institute, Republic of Korea; Sung Jin Kim, Republic of Korea Advanced Institute of Science and Technology, Republic of Korea; Jungho Kim, University of Maryland, USA; Hyeseung Lee, Republic of Korea Advanced Institute of Science and Technology, Republic of Korea

Experimental work was undertaken to measure the local wall heat transfer as a surface transition through CHF using a mid-range IR camera ( $3.5 \sim 5.0 \mu\text{m}$ ) in subcooled pool boiling of FC-72 at atmospheric pressure. Variations of the heat transfer parameters (temperature, heat flux, contact line length density (CLD), wetted area fraction (WF)) were obtained as function of heat flux and subcooling ( $\Delta T_{\text{sub}}$ ). The results show that the WF monotonically decreases with increasing heat flux and increasing subcooling. Through the quantitative analysis of these data, the characteristics of contact line speed, frequency between dryout events and the dry patch size distribution on the surface were obtained. CHF mechanisms proposed in the literature were evaluated against the observations.

\* This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP) (No. 2012M2A8A4025885).

\* This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Ministry of Education, Science and Technology (MEST) (No. 2012R1A3A2026427).

#### MEASUREMENTS OF 3-PENTANONE LASER INDUCED FLUORESCENCE FOR IMAGING CONCENTRATION OF AN EVAPORATE N-HEPTANE LIQUID FROM A POROUS MEDIA

Quentin Mouret, CETHIL-INSa de LYON-UCBL-CNRS, France; Manuel Kuhn, CETHIL-INSa de LYON-UCBL-CNRS, France; Patrizio Vena, CETHIL-INSa de LYON-UCBL-CNRS, France; Cedric Galizzi, CETHIL-INSa de LYON-UCBL-CNRS, France; Dany Escudie, CETHIL-INSa de LYON-UCBL-CNRS, France

One of the most important challenges in development of engine is the reduction of consumption and pollutant emissions. One way to achieve this goal is using the Gasoline Direct Injection concept, but with this technology, during cold starting, up to 90% of unburnt hydrocarbon emissions occur. Slow vaporization from fuel films in the intake port leads regions of rich mixture and thus incomplete combustion. In order to master the unburnt emissions, understanding the evaporation phenomena during the engine cycle is fundamental. In this work, a simplified configuration was developed. The evaporation is considered on a porous media which is saturated with n-heptane. The concentration of vapor is measured with a laser induced fluorescence technique using the 3-pentanone as tracer. Before applying the concentration diagnostic directly on fuel film evaporation, the dependence of the fluorescence signal of 3-pentanone/n-heptane gas mixture with parameters such as liquid temperature, concentration must be characterized. This parametric study investigated in a calibrated vessel at ambient pressure and with a range of temperature between ambient temperature and 373 K allows calibration of the fluorescence signal under wall film-related concentration and temperature conditions. Upon characterization of the relevant parameters, the LIF technique is applied to obtain quantitative measurements of fuel concentration near the saturated porous media. Results are finally compared to Micro Gas Chromatography measurement.

#### MICROBUBBLE GENERATION FROM ULTRAFINE BUBBLE WATER BY ULTRASONIC STIMULATION

Koichi Terasaka, Keio University, Japan; Nozomi Harasawa, Keio University, Japan; Isako Yamazaki, Graduate School of Keio University, Japan; Satoko Fujioka, Keio University, Japan

Ultrafine bubbles (UFB) whose diameters are smaller than 1  $\mu\text{m}$  hardly escape from water because of negligible buoyancy. Therefore, water suspending ultrafine bubbles is called "ultrafine bubble water". UFB water has been used in cleaning, agriculture and aquaculture in Japan. When the UFB water is used to wash a wall, it is collided to the wall surface using a water injector. By the stimulation at the collision, a lot of microbubbles are born from the UFB water. Rapid expansion from UFB to microbubble causes to peel off the dirt on the wall. Therefore, the number density and the size of microbubbles generated in UFB water influenced on the performance of cleaning wall. The stimulation which is given at the microbubbles' birth point is very important. To understand the microbubble generation mechanism from UFB water, it is necessary not only to observe the microbubble dispersion but also to elucidate the birth and growth of microbubbles.

In this study, a novel batch examination system which contained an ultrasonic irradiator was developed to investigate the mechanism of microbubble generation from UFB water. Microbubbles were instantaneously generated from UFBs as nucleus in water by a pulsed irradiation of ultrasound as a stimulation. The number density of microbubble depended on the number density of UFB.

\*This work was supported by JSPS KAKENHI Grant Number 25280032.

#### NUMERICAL SIMULATION OF IMPINGING JETS ATOMIZATION

Peiyu Zhang, Tsinghua University, China; Bing Wang, Tsinghua University, China

Numerical simulations based on the volume-of-fluid method are performed to study the impinging jets atomization considering the effects of jet inflow velocity profiles and artificial turbulence on the break-down of impinged liquid sheets. Both the simulated flow patterns and the statistical atomization feature of droplet size distribution agree well with the experimental data from the literatures. It is shown that the disintegration of impinged sheet can result from the unstable aerodynamic or impact waves. Although the contribution of the two types of waves is not fairly well quantified, the simulation indicates that the impact waves dominate the breakup of the liquid sheet over a wide range of ambient pressures. Effects of the jet inflow conditions including mean velocity profile and fluctuations on the atomization process are investigated by comparing the temporal variations of velocity and turbulent kinetic energy, as well as the wave frequency. The inflow velocity profile determines the wave frequency and the distribution of impact waves characterized by different amplitudes in the sheet, but the inflow velocity fluctuations, via augmenting or reducing the artificial disturbance in the jets, only dominates the amplitude of impact waves.

\* Support from supported by Tsinghua University Initiative Scientific Research Program grant 20141081217 is gratefully acknowledged.

#### A NEW TWO-PHASE FRICTIONAL PRESSURE DROP FOR ADIABATIC AND CONDENSING MINI/MICRO-CHANNEL FLOWS

Maryam Mashayekhpour, University of Kashan, Iran; Ebrahim Nemati Lay, University of Kashan, Iran

In this research work, a new developing two-phase frictional pressure drop model based on adiabatic and condensation experimental data has been presented. In recent years, several two-phase friction pressure drop correlations in mini/micro channels were presented, but still the lack of accurate models for designing such systems are exist. The frictional pressure drop of 3500 experimental data points of mini/micro channels, having hydraulic diameters ranges from 0.0695 to 6.22 mm, mass flux rating from 4.0 to 8528  $\text{kg/m}^2 \text{s}$ , with different working fluids is collected from reliable experimental data sources. Recent two-phase frictional pressure drop correlations reviewed and compared with the experimental data sets. A new Lockhart-Martinelli type correlation, with some modifications, is proposed to evaluate the frictional pressure drop experimental data. The results showed that the new model can predict the two-phase frictional pressure drop for adiabatic and condensing mini/micro-channel flows appropriately in comparison of other models used in this work.

#### APPLICATION OF A COMPRESSIBLE TWO-PHASE MIXTURE FORMULATION TO TURBULENT CAVITATING FLOWS

Sung-Eun Kim, Naval Surface Warfare Center Carderock Division, USA

A cloud of collapsing cavitation bubbles generates noise and can cause erosion on solid surface. Compressibility of fluids plays an important role in those processes. Shock waves generated during bubble collapse propagate and interact with bubbles themselves and nearby surfaces such as a solid wall or a free-surface. Dynamics of cavitating bubbles is significantly affected by turbulence in the carrier fluid. The paper is concerned with validation and application of a compressible two-phase Navier-Stokes solver that has been developed to efficiently and accurately resolve the salient physics of turbulent cavitating flows carrying a broad range of length- and time-scale. The "use cases" come from fluid machinery applications such as hydraulic pumps and marine propellers. The solution algorithm employs a finite-volume discretization and an implicit projection method adapted to compressible flows. Being implicit, the algorithm removes the time-step size restriction which otherwise, as in explicit schemes, can seriously hamper the solver efficiency. A homogeneous mixture model based on volume-fraction is adopted to describe two-phase flows. A mass-transfer model drawing upon bubble dynamics is used to model phase-change. Turbulence is directly resolved with subgrid-scale turbulence modeled by solving a subgrid turbulent kinetic energy equation. The solver is validated for a number of canonical and practical cases including a one-dimensional shock-tube, a bubble collapsing near a solid wall and a turbulent cavitating flow around a hydrofoil.

#### NUMERICAL STUDY ON HEAD-ON COLLISION OF WATER DROPLETS AT HIGH WEBER NUMBERS

Muyuan Liu, Technische Universitaet Darmstadt, Germany; Dieter Bothe, Technische Universitaet Darmstadt, Germany

Binary droplet collisions at high Weber numbers result in secondary droplets which are splattered out from the rim of the droplet complex. We study head-on collision of water droplets by means of Direct Numerical Simulations, focusing on the understanding of the rim instability mechanism that leads to the droplet detachment. To avoid artificial rupture of the thin lamella arising at high energy collisions, we stabilize the lamella by correcting the surface tension computation and the surface reconstruction in the lamella region. A domain adjustment technique, which adjusts the computational domain to the deformation of the collision complex, is employed for saving computational resources. Exerting initial white noise disturbance properly, we obtained collision complex developments that are in very good agreement with experimental observations. We measure the perturbation growth on the rim by conducting several simulations with the same disturbance strength and averaging the spectrum diagram of rim perturbation. We show that the rim perturbation development can be predicted by Plateau-Rayleigh theory in a long time span. In addition, we identify a period in which the rim is accelerating, which has not been mentioned in previous studies. Our results show that the rim deceleration is not big enough to dominate the rim instability.

\* Support from the German Research Foundation (DFG) within the scope of SFB-TRR 75 is gratefully acknowledged.

## INFLUENCE OF PARTICLE-WALL COLLISIONS AND PARTICLE AGGLOMERATION ON CYCLONE PERFORMANCE

Oscar Sgrott Junior, Martin-Luther-University Halle-Wittenberg, Germany; Guilherme A. Novelletto Ricardo, Martin-Luther-University Halle-Wittenberg, Germany; Martin Sommerfeld, Martin-Luther-University Halle-Wittenberg, Germany

Cyclones are stationary mechanical devices that use centrifugal force to separate solid particles from a carrier gas. They are widely used in the industry due to its large range of operational conditions and simplicity of construction, leading to low investment and maintenance costs. The design of a cyclone involves a compromise between the main performance parameters: collection efficiency and pressure drop (energy). The collection efficiency has a great sensibility with regard to the particle size, thereby; the larger particles are collected more easily than the smaller ones, because they are more inertial. Agglomeration phenomenon lead to larger particles (agglomerates), which would be easier collected, improving the collection efficiency for smaller particles. This effect is captured in the frame of the stochastic inter-particle collision model. In that respect, it is also important to consider the effect of wall roughness on particle-wall collisions and dispersion, which eventually will also modify inter-particle collisions and agglomeration. In order to evaluate the influence of these phenomena on the cyclone's performance, a CFD model based on Eulerian/Lagrangian approach is implemented in OpenFOAM®. Several simulations with different operational conditions are performed and compared to experimental data.

\*Support from CNPq (National Council for Scientific and Technological Development – Brazil) and DAAD (German Academic Exchange Service) is gratefully acknowledged.

## OPTIMAL FREQUENCY OF ELECTRICAL-IMPEDANCE MEASUREMENT IN THREE-PHASE MIXTURES

Marlon Hernandez Cely, Engineering School of Sao Carlos, University of Sao Paulo, Brazil; Sergio Gomez Bernal, Engineering School of Sao Carlos, University of Sao Paulo, Brazil; Hugo Fernando Velasco Pena, University of São Paulo, Brazil; Oscar Mauricio Hernandez Rodriguez, University of São Paulo, Brazil

Many studies on the characterization of electrical properties of multiphase fluid are found in the literature. One of the main motivations of these efforts has been the development of instrumentation for the measurement of volumetric fraction using electrical sensors. An experimental and theoretical study is presented, where the best frequency to measure the volumetric fraction in two- and three-phase mixtures is selected. Several permittivity models are applied to measure the volumetric fraction. The fluids used in the experiments were tap water, deionized water, mineral oil, isopropyl alcohol and hexane. Known volumes of fluids were mixed until obtaining a homogeneous mixture. The data were taken by a 1×4 wire-mesh sensor (WMS) immersed in the mixture. The WMS had a gap between planes of 1.4 mm, the wires were 3 mm apart from each other and the diameter of the wires was of 0.2 mm. The experimental system consisted of a generator, an oscilloscope and conditioning circuits (formed by operational amplifiers). A frequency scan was performed between 7000 Hz and 20 MHz for each mixture. A total of 60 logarithmically spaced frequencies were applied. Errors were calculated and used to select the best measurement frequencies for each tested mixture.

\*The authors would like to thank STATOIL and ANP for the financial support and CNPq for the research grant.

## FINITE VOLUME / FINITE AREA INTERFACE-TRACKING METHOD FOR TWO-PHASE FLOWS WITH SOLUBLE SURFACTANTS AND MASS TRANSFER

Chiara Pesci, Technische Universität Darmstadt, Germany; Paul Stephan Weber, TU Darmstadt - Center of Smart Interfaces, Germany; Manuel Falcone, TU Darmstadt - Center of Smart Interfaces, Germany; Holger Marschall, Technische Universität Darmstadt, Germany; Dieter Bothe, Technische Universität Darmstadt, Germany

This contribution is concerned with continuum physical and numerical modeling of two-phase flows under the influence of surfactants with and without interfacial mass transfer. Extending the Arbitrary Lagrangian-Eulerian Interface-Tracking method by Tuković and Jasak (doi:10.1016/j.compfluid.2011.11.003), we have developed a general framework capable to simulate transport of soluble and insoluble surfactants both in the case of diffusion-controlled or kinetically-controlled adsorption. For this purpose, a different numerical treatment is employed, reflecting the differences in the physics of the adsorption problem. Within the same numerical framework, interfacial mass transfer is accounted for. Then, a thermodynamically consistent model (arXiv:1501.05610) allows to investigate the effects of surfactant adsorption on interfacial mass transfer combining the methods for the two separate problems. Simulation results will be presented for a well defined application case, that is Taylor bubbles rising in square or cylindrical channels, giving valuable insights regarding the pure hydrodynamics, surfactant effects on the hydrodynamics and on mass transfer. To validate our computations, the results for the contaminated case are compared to experimental data with the restriction to setups where the volume effects due to mass transfer are negligible.

\* Support from grant DFG SPP1506 and SPP1740 is gratefully acknowledged.

## INFLUENCE OF MODEL ENERGY SPECTRUM AND STRUCTURE FUNCTION ON THE FLUID PARTICLE BREAKUP MODELS

Jannike Solsvik, NTNU, Norway; Hugo A. Jakobsen, NTNU, Norway

The conventional model framework for fluid particle breakup in turbulent flows is based on the Kolmogorov energy spectrum and the second-order longitudinal structure function for the inertial subrange of turbulence. The chemical engineering flows in chemical reactors and separators are generally characterized by finite Reynolds numbers of turbulence. For such flow characteristics the width of the inertial subrange of turbulence becomes narrow or might even disappear. For this reason, the conventional model framework should be extended from considering only the inertial subrange of turbulence to the wide spectrum of turbulence, i.e. dissipation, inertial and energy-containing subranges of turbulence.

Based on the model energy spectrum by Pope (2000), a new algebraic expression of the second-order longitudinal structure function valid for the entire spectrum of turbulence has been developed by the present authors. This new relation has been employed to extend fluid particle breakup models from the inertial subrange only to the wide spectrum of turbulence. It is concluded that the choice of model energy spectrum and second-order longitudinal structure function is of severe importance as these functions influence the approximations of the breakup functions significantly.

## REMAINING SWIRLING MOTION ON THE GAS OUTLET DUCT OF CYCLONE SEPARATORS: ALTERNATIVES TO INCREASE EFFICIENCY

Ricardo Salvo, Federal Technologic University of Parana., Brazil; Francisco Jose De Souza, Federal University of Uberlândia, Brazil; Diego Martins, Federal University of Uberlândia., Brazil

Cyclone separators are widely used in many industrial branches where gas-solid separation is needed. In most published studies on numerical simulation of cyclone separators, the gas outlet duct is treated as a short straight duct, differing considerably from the experimental and/or industrial apparatus. This may affect the results, due, mostly, to the remaining swirl in the gas outlet duct. This same remaining swirl may also be used to increase the separation efficiency. This work deals with the simulation of different gas outlet duct geometries and alternatives to increase the performance of a small cyclone separator by the utilization of two approaches, namely: the post cyclone device (PoC) and an annular overflow gas outlet duct. The concomitant gas-particle flow is simulated by means of Large-Eddy Simulations (LES). Although computationally expensive, the use of LES is advantageous as it does not demand turbulence dispersion models for the particles.

\* Support from grant PETROBRAS is gratefully acknowledged.

## A CONSERVATIVE LEVEL SET METHOD FOR TWO-PHASE FLOWS

Amin Mahmoudi Moghaddam, Tarbiat Modares University, Iran; Mehdi Shafieefar, Tarbiat Modares University, Iran; Roozbeh Panahi, Tarbiat Modares University, Iran

In this study a Compact Conservative Level Set (CCLS) method has been developed to improve the accuracy and robustness of Level Set (LS) approach in simulation of moving interfaces. To resolve well-known mass conservation issue of LS approach, a novel method with an excellent mass conservation property; while keeping the simplicity of the original method; is introduced. Instead of using non-oscillatory advection schemes to transport LS function, application of compact schemes are investigated. Here, the hyperbolic tangent function; used as LS function; is conservatively transported through a High-Order Compact (HOC) finite difference scheme. In order to retain thickness of the coated interface while retrieving the hyperbolic tangent profile, a conservative re-initialization is adopted by introducing a novel Combined Compact Difference (CCD) method. Spurious oscillations produced in advection step are resolved in re-initialization step. The developed method is employed in a wide range of test cases and its noticeable mass conservation property is revealed even for complex cases, as well as its accuracy and stability in large Courant numbers and longtime simulation on a coarse mesh.



## AN EXPERIMENTAL STUDY OF THE FLOW CHARACTERISTIC OF AIR-WATER TWO PHASE FLOW IN A SINGLE SCREW CHANNEL

Bo Cai, *Beijing University of Technology, China*; Guodong Xia, *Beijing University of Technology, China*; Xianfei Liu, *China*; Guang Yang, *China*

An experimental study of air-water two phase flow behavior flowing through a single-screw channel is presented. The flow patterns are distinguished and recorded by high speed camera using the backlight imaging tomography. The single-screw channel made by organic glass is a helical pipe with variable cross-section. Its diameter is 155mm and the equivalent diameter in the helical angle of 90 degree is 25mm. Following previous works, the flow regimes considered are bubble, slug and annular. Then the flow regime map and the transitions of the single-screw channel are presented. The flow patterns evolution in the different helical angle (45°, 90°, 135°) of the single-screw channel is illustrated by the force analysis. However, flow characteristics consider a single flow regime in order to take into account the flow channel geometry. So the characteristic of slug flow and the coincidence of the slug are investigated. It is shown that flow is instability in the angle of 45 degree, but in angle of 90 and 135 degree the flow is very smooth and steady.

Acknowledgement: The work is financed by the National Basic Research Program of China (2011CB710704) and the National Nature Science Foundation of China (No. 51176002).

## ELECTRICAL AND ULTRASONIC DUAL-MODALITY MEASUREMENT FOR OIL-WATER FLOW BASED ON KALMAN FUSION

Pengfei Li, *Tianjin University, China*; Hao Wu, *Tianjin University, China*; Chao Tan, *Tianjin University, China*; Feng Dong, *Tianjin University, China*

Measuring the process parameters of two-phase flow is of great significance in industrial processes. Data fusion integrates information from different sensors to reduce the uncertainty and generate more reliable results. In this paper, an electrical and ultrasonic dual-modality measurement system is presented to measure phase fraction and flow velocity of oil-water two-phase flow based on data fusion. Ultrasonic sensor based on Doppler Effect can measure the velocity of dispersed particles. Electrical sensor detects the change of electrical parameters and determines the phase fraction based on Maxwell's equations. The proposed measurement method contains a two-layer fusion. In the anterior layer, centralized Kalman fusion is used to combine data from the conductance sensor and capacitance sensor to estimate water phase fraction  $\epsilon$ . In the posterior layer,  $\epsilon$  is regarded as a parameter. Cross-Correlation method is employed to analyze data generated by conductance ring to identify flow velocity  $u_c$ . The  $u_c$  and Doppler speed  $u_d$  are integrated to improve the accuracy of the estimation of flow velocity. The experimental results show that data fusion can improve the performance of measurement.

## GENERATOR FOR WIDE RANGE OF BUBBLE SIZE USING ACOUSTIC PRESSURE WAVE AND ELASTIC TUBE

Yuma Kasai, *Shizuoka university, Japan*; Toshiyuki Sanada, *Shizuoka University, Japan*

We demonstrate the control of bubble formation by combining a pulsed acoustic pressure wave in a gas phase and a slit elastic tube. We can control the bubble radius in the range from 150  $\mu\text{m}$  to 6 mm by using the same tube without liquid flow. In addition, we experimentally investigated the mechanism of this bubble generator. The results indicated that there was no significant difference in the slit opening time, even when the amplitude of the acoustic pressure wave was changed, and that the bubble radius was determined by the opening displacement of the slit, which was governed by the surface tension. In addition, the shape oscillation of a bubble due to surface tension was found to promote its detachment from an elastic tube with poor wettability. Furthermore, we investigated the propagation of the wave inside the tube. The time that the positive pressure is maintained on the waveform depends on the tube length and the boundary condition, and the generated bubble size changes depending on the time. We conclude that this phenomenon was caused by the change of the phase of the reflected wave at the tube end. We also simulated the acoustic pressure wave in the tube by using FDTD method and the result strongly supported our consideration.

## EFFECT OF DISPERSED MODALITY ON TURBULENCE CHARACTERISTICS OF LIQUID-LIQUID DISPERSION IN STIRRED TANK

Jiacheng Han, *China University of Petroleum, Beijing, China*; Nannan Liu, *China University of Petroleum, Beijing, China*; Fangyuan Liu, *China University of Petroleum, Beijing, China*; Meng Zhang, *China University of Petroleum, Beijing, China*; Wei Wang, *China University of Petroleum Beijing, China*; Jing Gong, *China University of Petroleum Beijing, China*

In this work, experimental investigation of effect of dispersed modality on turbulence characteristics of liquid-liquid dispersions in stirred tank was performed with Exxsol D130 oil and glycerol aqueous solution selected as test fluids while Span80 used as surfactant. 2D Angle-resolved particle image velocimetry combined with refractive index matching technique was utilized to record instantaneous velocities distributions of both phases. Various surfactant concentrations were added to control the extent of dispersion. Turbulence parameters including turbulent kinetic energy and energy dissipation rates were calculated based on velocities to analyze effect of the level of dispersion on turbulence flow field. Results showed the turbulent flow characteristics were drastically influenced by the drop size of dispersion under the constant dispersed volume fraction, including drop numbers and scales. Combining with the measurement of interfacial tension of various concentration system by Pendant Drop method and drop size distributions from emulsion systems, it was suggested that turbulence intensity was influenced by both momentum and kinetic energy transfer brought by droplets movements and augment or suppression effect determined by size of droplet itself.

\* Support from the Foundation for the Author of National Excellent Doctoral Dissertation of PR China(201254) is gratefully acknowledged.

## MULTIPHASE FLOW MEASUREMENT BY MULTIELECTRODE ELECTROMAGNETIC FLOW METER BASED ON REGION WEIGHT FUNCTION

Yuyang Zhao, *Hebei University of Science and Technology, China*; Zheng Li, *Hebei University of Science and Technology, China*; Wei Liang, *NORENDAR International LTD., China*

A multi-electrode electromagnetic flow meter (MEFM) based on region weight function, which is derived from Shercliff weight function theory, has been developed and presented in this paper. The novel flow meter uses the region weight function theory to reconstruct the local mean velocities of measurement cross-section and flow rate by acquiring the induced voltages at the boundary along flow pipe circumference and images the velocity profile in real time. This paper describes the design of MEFM. The experiments on soil-water flow has been undertaken and the results are presented in order to demonstrate that the novel multi-electrode electromagnetic flow meter is highly suited in measuring volumetric flow rate and mapping the local average velocity of non-axisymmetric flow.

## IbD – Intensified by Design

Milan Mihajlovic, *Eindhoven University of Technology, The Netherlands*; Ivo Roghair, *Eindhoven University of Technology, The Netherlands*; Martin Van Sint Annaland, *Eindhoven University of Technology, The Netherlands*

Intensified by Design (IbD®) is a European project aiming to create an holistic platform for processes in which solids are an intrinsic part. The platform will allow to facilitate process intensification design and optimization using currently existing approaches or through completely new approaches for solids processing. We support the project by a.o. developing phenomenological models for fluidized bed membrane reactors (FBMRs), based on insights obtained from more fundamental simulation techniques such as discrete particle models (DPM, an Euler-Lagrangian type model) and supported by experiments using advanced non-invasive optical techniques. Our DPM can simulate gas-solid flows with gas phase chemical species transfer as well as perm-selective membranes in the reactor walls. The model is used to better understand the prevailing phenomena such as concentration polarization and to derive closures that will further optimize the phenomenological models, and simultaneously improve our insight in FBMRs. The performance of gas-solid FBMRs is governed by the rate of gas-to-emulsion and emulsion-to-membrane mass transfer. Existing correlations for gas-to-emulsion phase mass transfer, and their assumptions, will be revisited using this new high-detail model. Additionally, we will develop closures for emulsion-to-membrane mass transfer (accounting for concentration polarization) that eventually allows phenomenological models to predict the performance of larger scale FBMRs with more accuracy.

#### CHARACTERISTICS OF TURBULENT BOUNDARY LAYER AFFECTED BY A SPHERE NEXT TO A WALL

Hui Zhao, Zhejiang University, China; Xiaofei Liu, Zhejiang University, China; Dong Li, Zhejiang University, China; Fan Wu, Zhejiang University, China; Kun Luo, Zhejiang University, China; Jianren Fan, Zhejiang University, China

The paper is concerned with some characteristics of the flat-plate boundary layer influenced by a stationary sphere in proximity to the wall. A combined multiple-direct forcing and immersed boundary method is used in the present direct numerical simulation work. The Reynolds number is 2500, and different gap ratios between the distance from the sphere bottom to the flat plate and the sphere diameter are considered. In order to study the characteristics of the velocity distributions and the vortex dynamics, the vortex structure and the significant variation of the turbulent boundary layer are investigated in detail. A jet flow forms between the sphere and the flat plate, which affects the vortex separation on the bottom of the sphere and the recirculation flow behind the sphere. The effects of the jet flow on the flow field are influenced by the gap ratios. In this paper, the distributions of the coherent structures and the effects of the sphere with different gap ratios to the flat plate are discussed. Besides the time-averaged velocity distribution and the vortex dynamics, distribution regularities of turbulent boundary layer affect by the sphere are also focal points in our investigation.

\* Support from The National Natural Science Foundation of China (No. 51136006) is gratefully acknowledged.

#### DENSITY WAVES IN THE GRAVITATIONAL FLOW OF GRANULAR MATERIAL IN NARROW PIPES

Carlos Alvarez, University of Campinas, Brazil; Erick De Moraes Franklin, UNICAMP - University of Campinas, Brazil

The dynamics of the density waves in gravity-driven flows of granular materials in narrow pipes is studied experimentally. The experimental device consisted of a vertical glass pipe through which different populations of glass spheres flowed. Scanning electron microscopy (SEM) was used in order to determinate the size distribution of glass spheres. The granular flow was filmed by a high-speed CCD camera. The length scales and celerities of density waves were calculated using the spatiotemporal diagrams obtained by digital image processing. Three wave flow regimes were observed, the first corresponding to density waves that propagate at a constant celerity, the second to low amplitude oscillating waves, and the third to large amplitude oscillations. The oscillating regimes appeared when the experiments were conducted using a batch of glass spheres with rough surface. A key observation concerns the direction of celerity: in the constant celerity wave regime, the density waves propagated upward, contrary to flow direction. In the oscillating wave regime, the density waves propagated downward, and analogies with sound propagation in two phase flows appear.

#### FOUR-WAY COUPLED SIMULATIONS OF THE GAS-SOLID FLOW IN A CYCLONE SEPARATOR

Diego Martins, Federal University of Uberlândia., Brazil; Francisco Jose De Souza, Federal University of Uberlândia, Brazil; Ricardo Salvo, Federal Technologic University of Paraná., Brazil

Cyclone separators are commonly used in industry to separate particles from gases. The collection efficiency in these devices can reach over 99.99%. Despite their high efficiency, the use of cyclones for separating particles may be impractical due to the high wear rates caused by the collision of particles with the walls. In this paper, the particle-gas flow in a cyclone separator will be simulated using the Euler/Lagrange approach. The exchange of momentum between particles and fluid, and the collision between particles are modelled in a four-way approach. The effect of the mass loading on the interparticle and particle/wall collisions is analyzed, and the resulting erosion is accounted by an empirical correlation. Some interesting effects, such as the reduction of the erosion rate with the mass loading, are reported and explained based on the interactions between phases. Such an effect has been reported experimentally as well. The inter-particle collisions are shown to be very important even at low mass loadings.

#### THE PROPAGATION OF SHOCK WAVES IN GAS-WATER-SATURATION BULK MEDIA

Andrei Zhurov, IMech USC RAS, Russian Federation; Alfir Akmetov, Institute of Mechanics of Ufa Branch, RAS, Russian Federation; Said Urmancheev, Mavlutov Institute of Mechanics, Russian Federation

Experimental study of the behavior of weak shock waves in media consisting of sand or glass beads, saturated with gas and water were carried out in a vertical shock tube equipped with a section of bulk media (SBM). Piezoelectric sensors with charge amplifier on the base Board L-card coupled with a computer were used for the registration of pressure. 3 sensors were used: first is in the chamber of low pressure at a distance of 72 cm from the aperture, the second is in the SNA at a depth of 5 cm from its surface and third is bottom. The experiments were conducted with water-saturated sand in the range of particle sizes from 50 to 700 microns and the beads from 250 to 500 microns with the water maintenance (the ratio of water volume to pore volume) 0 to 100%. The degree of absorption of the shock wave was from the ratio of amplitudes on the plots obtained from the readings of the first and bottom sensors. After vibratory compaction porosity for sand is 35%, and for glass beads – 32%. For the experiment in the dry bulk it is initially dried at a temperature of 1100 C for 6 hours. and the mixture was placed into a vacuum tank to get rid of gas bubbles in the medium at 100% water saturation.

It should be noted in common: the sand in the beads and the largest absorption is obtained at 100% water content. Also the dependence obtained for the velocity of the shock wave at low humidity, give a little blip. In the sand the amount of water increases from 0 to 90% leads to an improvement in the passage of a shock wave more than 3 times. The beads for this increase no.

#### UPWARD-VERTICAL TWO-PHASE FLOW IN A LARGE-DIAMETER ANNULAR DUCT

Alex Roger Almeida Colmanetti, University of São Paulo at São Carlos School of Engineering, Brazil; Marcelo Souza De Castro, State University of Campinas - UNICAMP, Brazil; Marcel Cavallini Barbosa, University of Sao Paulo, Brazil; Tor Kindsbekken Kjeldby, Statoil, Norway; Peter W. J. Derks, Statoil, Brazil; Oscar Mauricio Hernandez Rodriguez, University of São Paulo, Brazil

Gas-liquid flows in vertical pipes have been studied for a long time, leading to the development of models and correlations to predict void fraction and pressure gradient in pipes of diameters varying from microns to many inches. One of those is the drift-flux model, where the slip ratio between the phases is taken into account. On the other hand, studies of gas-liquid flows in annular ducts are scanty, even though it is present in many industrial applications, e.g., directional oil wells and wells equipped with ESPs. The few works devoted to this subject are based on small-duct data; the hydraulic-diameter concept is used and the drift-flux model is applied by simply adjusting some constants. The use of models based on the hydraulic-diameter concept presents significant errors. The goal of this work is to compare new experimental data of gas-liquid flow in an annular duct with models from the literature and then propose new closing relations for void fraction and pressure gradient. The experimental database was acquired in the Laboratory of Thermal and Fluids Engineering – LETEF of the University of São Paulo, Brazil. The development of new closure relations for the drift-flux model applicable to annular-duct flow is essential for the optimized design of oil wells.

\*The authors would like to thank STATOIL and ANP for the financial support and CNPq for the research grant.

#### ANALYSIS OF TWO-PHASE FLOW IN AN CENTRIFUGAL PUMP IMPELLER

Renzo Sabino, Federal University of Technology - Paraná, UTFPR, Brazil; Henrique Stel, Federal University of Technology - Paraná, UTFPR, Brazil; Dalton Bertoldi, Federal University of Technology - Paraná, UTFPR, Brazil; Rigoberto Morales, Federal University of Technology - UTFPR, Brazil

Some wells produce oil by artificial lift methods such as electrical submersible pumps. When gas is also present in the oil production, the pump efficiency tends to degrade. This degradation is moderate when dispersed gas bubbles flow along with the liquid. However, it can be severe when these bubbles coalesce and become trapped inside the impeller (surging). In order to comprehend these phenomena, it is important to understand how individual bubbles interact with the liquid phase as they flow through the impeller channels. In this sense, this study presents a theoretical and experimental analysis of the forces between a bubble and its surrounding liquid phase in a radial pump impeller. Transparent pump parts enable the visualization of the movement of the bubble by means of high-speed visualization. An algorithm computes the bubble velocity according to the bubble displacement through time. With the help of static pressure values and liquid velocity profiles extracted from a CFD model, the drag coefficient can be estimated by assuming a force balance and auxiliary expressions from literature. Results show that the drag coefficient increases significantly with the bubble diameter for low particle Reynolds numbers.

# A STUDY OF TWO-PHASE FLOW MALDISTRIBUTION BETWEEN CELLS OF A HIGH PRESSURE PEM WATER ELECTROLYSIS STACK VIA COMPUTATIONAL FLUID DYNAMICS

Xin Gao, Aalborg University, Denmark; Anders Christian Olesen, Aalborg University, Denmark; Carsten Romer, IRD A/S, Denmark; Soren Knudsen Kaer, Aalborg University, Denmark

In a high pressure (HP) proton exchange membrane (PEM) water electrolysis stack, liquid water serves both as reactant and coolant, its maldistribution hurts the performance and durability.

It has been proved that interdigitated flow field for the anode of a HP-PEM water electrolysis cell tends to cause the maldistribution and bubbles of product gas in flow field grows hotspots excessively. Under this circumstance, this study will investigate how these issues will be magnified with the stack size being increased. Analyses will be conducted on a three-dimensional multiphase stack model. The model will be born from a comprehensive cell model inside ANSYS CFX. The art of simplification will largely decide the effectiveness and feasibility of the stack model in this study. Design optimizations, e.g., redesign of manifolds, modifications on the head and the tail of the flow field, etc., to compensate the above issues will be tried out. Hopefully guidelines for improving the performance and manufacturing the electrolysis stack will be drawn out.

\* Support from the Innovation Foundation Denmark under the project e-STORE is gratefully acknowledged.

# LIQUID INFILTRATION CHARACTERISTICS INTO CLOSED END STRUCTURES UNDER THE EXTERNAL PRESSURE

Eri Yamaguchi, Shizuoka University, Japan; Hiroshi Nozaki, Shizuoka University, Japan; Toshiyuki Sanada, Shizuoka University, Japan

In the wet process of semiconductor device fabrication it is necessary to infiltrate liquid into patterns on wafers. There are many studies about open end structures such as capillary rise phenomena, however there are few studies about closed end ones. The purpose of this present study is to clarify the liquid infiltration characteristic under the external pressure. We observed the liquid infiltration into millimeter and micrometer scale tubes using a high speed video camera with a microscope. We applied pressure under two conditions, i.e. quasi-static and stepwise change. As results, in spite of applying pressure, both millimeter and micrometer scale tubes with high aspect ratio were not completely filled with ultrapure water. On the other hand, micrometer scale tubes with low aspect ratio were completely filled. These results indicate that to infiltrate liquid into closed end structures completely, it is necessary to dissolve all of the gas inside of the structures. We believe that the dissolution is limited by forming a concentration boundary layer. From present study, the gas-liquid interfacial area relative to the total liquid surface area play an important role for liquid infiltration into closed end structures.

# EXPERIMENT STUDY OF THE INSITU DROP SIZE DISTRIBUTIONS AT DIFFERENT SPATIAL REGIONS IN STIRRED TANK

Meng Zhang, China University of Petroleum, Beijing, China; Nannan Liu, China University of Petroleum, Beijing, China; Jiacheng Han, China University of Petroleum, Beijing, China; Wei Wang, China University of Petroleum Beijing, China; Jing Gong, China University of Petroleum Beijing, China

Experimental studies of drop size distributions in oil in water dispersions in stirred tank agitated by a four-bladed turbine are presented. Deionized water and D130 oil were selected as test phases. Agitation of liquid-liquid dispersions in stirred tank leads to continuous changes of drop size distribution due to breakage and coalescence processes, the turbulence intensity of different locations in the tank is different, the forces generated by turbulence will cause drop breakage, meanwhile, drop will coalescence which is connected with the shape-restoring interfacial tension. If forces generated by turbulence exceed the shape-restoring interfacial tension, drop breakage will occur, generating different drop size distributions within the tank. At the same time, the dispersed phase fraction will have an effect on the drop size distribution, for this purpose, drop size measurements are performed for different dispersed phase fractions at three locations in the tank, in the impeller zone and two locations in the bulk respectively. FBRM has been used as the drop size measuring technique. The Sauter mean diameter, drop size distributions and transient drop size distributions are obtained at different conditions. Upon comparison, the influence of dispersed phase fraction and turbulence intensity on the drop size distribution can be examined.

\* Support from the Henry Fok Foundation (142021) is gratefully acknowledged.

# LIQUID MALDISTRIBUTION IN STRUCTURED PACKING IN AN INCLINED COLUMN: CFD SIMULATION AND VALIDATION

Philippe Beard, IFP Energies nouvelles, France; Manel Fourati, IFP Energies nouvelles, France; Pascal Alix, IFP Energies nouvelles, France; Thomas Maubert, TOTAL, France; Vincent Carlier, PROSERNAT, France; Xavier Courtial, PROSERNAT, France; Clement Salais, PROSERNAT, France

As natural gas always contains contaminants such as acid gases, in particular CO<sub>2</sub> and H<sub>2</sub>S, acid gas removal is required to meet LNG (Liquefied Natural Gas) specifications. This process is performed in packed liquid-gas contacting columns operating under counter-current liquid-gas flow. In offshore conditions, the performance may be altered because of ship movement resulting in liquid maldistribution within packed beds.

In order to improve the design of such columns, this work investigates the effects of operating conditions on the liquid distribution in a 0.6 m diameter column filled with a structured packing. The influence of the column tilt angle and aspect ratio as well as the liquid mass flow rate and viscosity was studied in a test rig installed on a hexapod moving platform allowing ship motion simulation.

Unsteady 3D CFD (Computational Fluid Dynamics) simulations were carried out using a two-phase liquid-gas Eulerian approach. The structured packing was modeled as a porous medium wherein gas-liquid momentum exchanges and liquid dispersion were taken into account. Results of simulations are compared to experimental measurements to validate the liquid maldistribution predicted by the CFD model dedicated to the studied packing.

# ADSORPTION OF ALCOHOLIC ADDITIVES IN WATER/LIBR AQUEOUS SOLUTION AT LOW PRESSURE

Federico Lonardi, University of Kassel, Germany; Andrea Luke, University of Kassel, Germany

Optimization of absorption chillers is a fundamental task to pursue in the refrigeration industry, since these chillers are still characterized by large sizes and low efficiency. An efficient way to achieve this aim is by the mean of alcoholic surfactants. Indeed, when added to the working fluid pair (water/LiBr) in small quantities, they lower the surface tension, inducing a local Marangoni convection at the liquid/vapor interface of the absorber tube bundle. As a consequence, the heat and mass transfer in the absorber is increased. Nevertheless, values of surface tension measured in literature tend to scatter and the influence of many factors on the enhancement mechanism is not yet fully understood.

In this work, experimental measurements of surface tension of water and aqueous lithium bromide solution with additives are presented. The surface tension is measured according to the Pendant Drop Method. Several parameters are varied during the experiments, such as pressure, temperature and surrounding conditions. All the measurements are performed in a self-developed vacuum cell in order to have the same vacuum condition that occurs in the absorber and to produce reliable data.

\* Support from ITN Marie Curie Network "SHINE" is gratefully acknowledged.

# NUMERICAL STUDY ON MIXING CHARACTERISTICS IN SPHERICAL FLASK WITH MAGNETIC STIR BAR

Sang Hyuk Lee, Republic of Korea Institute of Machinery and Materials (KIMM), Republic of Korea; Wonsam Cho, Republic of Korea Institute of Machinery and Materials (KIMM), Republic of Korea; Dae-Hwan Kim, Republic of Korea Institute of Machinery and Materials (KIMM), Republic of Korea; Taehyun Lee, Republic of Korea Institute of Machinery and Materials (KIMM), Republic of Korea; Kyungha Ryu, Republic of Korea Institute of Machinery and Materials (KIMM), Republic of Korea

Recently, magnetic stirrer is commonly used as the mixing device in chemical and biological laboratories. In the magnetic stirrer, the liquid contained in a flask is generally mixed by spinning a magnetic stir bar immersed in the liquid with the rotating magnetic field. To know the mixing characteristics in a spherical flask with a magnetic stir bar, the present study numerically investigate the interfacial flow induced by the stirred magnetic bar using the VOF method for the interface tracking of two-phase flow simulation. To verify the numerical methodology, the numerical results on the shape of interface between liquid and gas for various rotating speeds were compared with the captured images obtained from the experiment with a CCD camera, the results were found to show good agreement with the experimental data. With the verified numerical methodology, the mixing characteristics were predicted for various geometric and operating conditions. From the results, it was found that the mixing characteristics are mainly determined with the size of magnetic stirrer and the rotating speed of magnetic stir bar.

\* Support from the R&D Program of Korea Institute of Machinery and Materials (KIMM) funded by the Ministry of Knowledge Economy (MKE) is gratefully acknowledged.

## INFLUENCE OF PARTICLE FORCES ON PARTICLES IN FLUID PHASE RESONANCE MIXERS

*Silvio Schmalfuss, Martin-Luther-University Halle-Wittenberg, Germany; Martin Sommerfeld, Martin-Luther-University Halle-Wittenberg, Germany*

In a Fluid Phase Resonance mixer, a pipe reaches into a liquid in a vessel and gas cushions are created above the liquid inside and outside the pipe. A drive attached to the gas cushion inside the pipe creates harmonically oscillating pressure. Thus, the liquid inside the pipe is forced to perform oscillations, which in turn induce a motion inside all the liquid in the vessel. This motion is used for mixing purposes.

For the prediction of mixing properties particle simulations employing an Euler-Lagrange approach are performed. In order to achieve a good balance between accuracy and computational efficiency, the influence of different forces acting on spherical point particles in the time and space dependent flow field is studied for different particle properties. Forces taken into account are added mass, buoyancy, drag, gravity, history force, pressure gradient, rotational lift, and shear lift. By analysing the contribution of the different forces under varying properties (Stokes number, Reynolds number, density ratio) in this industrial application, it shall be investigated, which forces are necessary for acceptable accuracy of simulations under specific flow properties.

\* Support by the German Research Foundation is gratefully acknowledged.

## SIMULATION OF TURBULENT LIQUID-LIQUID DISPERSIONS: VALIDATION OF DIFFERENT BREAKAGE AND COALESCENCE KERNELS AND COMPARISON BETWEEN DIFFERENT MODELLING APPROACHES

*Jeremias De Bona, Politecnico di Torino, Italy; Dongyue Li, Beijing University of Chemical technology, China; Antonio Buffo, Aalto University, Finland; Marco Vanni, Politecnico di Torino, Italy; Wioletta Podgorska, Warsaw University of Technology, Poland; Daniele Marchisio, Chemical Engineering, Politecnico di Torino, Italy*

In this work turbulent liquid-liquid dispersions in stirred tanks are simulated by using computational fluid dynamics (CFD), population balance models (PBM) and different simplified models. The different models can be divided in two types: zero-dimensional (0D) models, in which perfect mixing of the phases is assumed, and three-dimensional (3D) models, where the spatial inhomogeneities are considered. Models predictions are validated against experimental data for different test cases, consisting of different stirred tank geometries and different continuous and disperse phases. 3D simulations are performed with OpenFOAM-2.2.x, based on the use of the Eulerian-Eulerian approach, whereas the PBM is solved with the quadrature method of moments (QMOM). Different coalescence and breakage kernels are considered, based on the classical homogeneous and multifractal turbulence theories. The comparison of the different (0D and 3D) models highlights that the 0D models can be used under dilute conditions, whereas in the dense regime the 3D models should be used instead. The comparison allows also to derive a correct formulation for the 0D models, that are instead often wrongly formulated.

## A LATTICE BOLTZMANN MODEL TO SIMULATE LIQUID-LIQUID SLUG FLOW IN 90° BEND

*Shakendra Jain, Indian Institute of Technology Roorkee, India; Sumana Ghosh, Indian Institute of Technology Roorkee, India*

Understanding the interfacial dynamics of separated two phase flow is of importance to technological applications and it presents a relevant theoretical challenge too. The most common and simple form of separated two phase flow is slug flow. It consists of intermittent Taylor bubble of the secondary phase and the liquid slugs between two successive bubbles. Slug flow is commonly encountered in variety of industrial applications such as oil refineries, nuclear reactors etc. Due to its wide application, a number of studies both experimental and theoretical are reported. However, less information is available on the effect of different pipe fittings on this flow pattern. Hence an interest is felt to simulate liquid-liquid slug flow when it encounters a 90° bend in its flow path. Such type of fittings is very common in industry. A model based on lattice Boltzmann technique is used for this purpose. Drop dynamics over a wide range of drop volume and three different tube diameters is numerically simulated. Thermo physical properties of the drop liquid are varied to obtain the influence of such parameters on the drop shape and drop velocity which are found to be influenced by the presence of 90° bend in its flow-path.

## MODELING OF SCATTERING INTENSITY OF MINERAL AEROSOLS WITH A GAUSSIAN BEAM

*Hong Tang, China Jiliang University, China; Jianzhong Lin, China Jiliang University, China*

Based on the Generalized Lorenz Mie Theory (GLMT), the scattering intensity of mineral aerosols in the Gaussian beam is investigated, and an appropriate modeling of the scattering intensity for the real mineral aerosols including the feldspar, quartz, and red clay from the Amsterdam light scattering database is proposed. In this modeling, the spheroid is applied to represent the real non-spherical mineral aerosols, and these non-spherical particles are randomly distributed within the Gaussian beam region. Meanwhile, the Monte Carlo statistical estimate method is used to determine the distributed positions of these random non-spherical particles. Moreover, a statistical average method for the non-spherical particles is proposed to represent the scattered intensity of the real non-spherical mineral aerosols. On the other hand, the T matrix method is also used for calculate the scattering intensity of the spheroid particles in order to compare the scattering properties between the plane wave and the Gaussian wave. Simulation and experiment data results indicate that fairly reasonable results of the scattering intensity for the mineral aerosols can be obtained with this proposed method, and it can provide a reliable and efficient approach to reproduce the scattering intensity of the real randomly distributed mineral aerosols illuminated by the Gaussian beam.\* This work was supported by the National Natural Science Foundation of China (No.11132008 and No.11202202) and Zhejiang Province Natural Science Funds (LY15A020003).

## MODELING OF SCATTERING INTENSITY OF MINERAL AEROSOLS WITH A GAUSSIAN BEAM

*Hong Tang, China Jiliang University, China; Jianzhong Lin, China Jiliang University, China*

Based on the Generalized Lorenz Mie Theory (GLMT), the scattering intensity of mineral aerosols in the Gaussian beam is investigated, and an appropriate modeling of the scattering intensity for the real mineral aerosols including the feldspar, quartz, and red clay from the Amsterdam light scattering database is proposed. In this modeling, the spheroid is applied to represent the real non-spherical mineral aerosols, and these non-spherical particles are randomly distributed within the Gaussian beam region. Meanwhile, the Monte Carlo statistical estimate method is used to determine the distributed positions of these random non-spherical particles. Moreover, a statistical average method for the non-spherical particles is proposed to represent the scattered intensity of the real non-spherical mineral aerosols. On the other hand, the T matrix method is also used for calculate the scattering intensity of the spheroid particles in order to compare the scattering properties between the plane wave and the Gaussian wave. Simulation and experiment data results indicate that fairly reasonable results of the scattering intensity for the mineral aerosols can be obtained with this proposed method, and it can provide a reliable and efficient approach to reproduce the scattering intensity of the real randomly distributed mineral aerosols illuminated by the Gaussian beam.

\* This work was supported by the National Natural Science Foundation of China (No.11132008 and No.11202202) and Zhejiang Province Natural Science Funds (LY15A020003).

## DETERMINISTIC AND STATISTICAL MODELLING OF PARTICLE-PARTICLE COLLISIONS IN SUPERSONIC GAS-SOLID FLOWS PAST BODIES

*Dmitry Reviznikov, Moscow Aviation Institute (National Research University), Russian Federation; Andrey Sposobin, Moscow Aviation Institute (National Research University), Russian Federation; Aleksei Varaksin, Joint Institute for High Temperatures, Russian Federation*

The problem of interaction of gas-solid flows with bodies emerged as a result of studies of motion of various flying vehicles in a dust-laden atmosphere. The presence of solid particles in a flow may cause a significant (sometimes, many times over) increase in heat fluxes, as well as erosion wear of the surface subjected to flow. The intensity of the processes which accompany gas-solid flow past bodies depends on the inertia and concentration of particles. As to the concentration of particles, its value can many times exceed the initial value in an unperturbed flow due to abrupt deceleration of flow as approaching the body, particle-wall interaction and particle-particle collisions. The aim of this study is to compare two possible mathematical models taking into account the particle-particle collisions, namely: deterministic (direct numerical method) and statistical (Monte Carlo method). The authors have developed a full-scale discrete element method. All collisions in the framework of this method are simulated sequentially in chronological order by using of order of collisions. For comparison of above-mentioned methods the calculations have been made of supersonic gas-solid flow for a wide range of particle volume concentrations.

\*The work was supported by the Russian Foundation of Basic Research, Grant No. 15-08-06262.

#### ASSESSMENT OF THE EFFECT OF DYNAMIC PRESSURE CLOSURE ON THE SIMULATIONS OF VERTICAL ANNULAR GAS-LIQUID FLOW WITH THE 1D TWO FLUID MODEL

Eric Gonzalez, PUC-Rio, Brazil; Joao Carneiro, Instituto Sintef do Brasil, Brazil; Angela Nieckele, PUC-Rio, Brazil

Vertical annular flows are found in several industrial applications, in particular in the nuclear and oil industries. An analysis of vertical ascending annular flow with the one-dimensional two-fluid model is performed. Modeling of dynamic pressure is included, to render the system of equations well-posed. Non-uniformity of the axial velocity profiles in the cross section is also addressed, by the introduction of a profile shape factor. The finite volume method is applied to discretize the governing system of equations. A second order TVD spatial discretization is employed, as well as a second order time integration. A systematic grid convergence test is performed to demonstrate the stabilizing effect of the dynamic pressure and profile shape factor. Comparison with experimental data available in the literature is performed for air-water flows, regarding pressure drop, film thickness and wave characteristics. Good agreement was obtained, demonstrating the accuracy of the present model.

#### A KINETIC WAX DEPOSITION MODEL FOR W/O EMULSION WITH THE INFLUENCE OF CRYSTALLIZED WAX ADHERING TO OIL-WATER INTERFACE

Pan Song, China University of Petroleum, Beijing, China; Wei Wang, China University of Petroleum Beijing, China; Juheng Yang, China University of Petroleum, Beijing, China; Jing Gong, China University of Petroleum Beijing, China

In water-in-crude oil emulsion system, if the bulk phase temperature is lower than the wax appearance temperature (WAT), crystallized wax particles will precipitate from the oil phase, and has a tendency to adhered to the oil-water interfaces where surfactant molecule exist. In present work, the influence of adhered wax crystals on the oil-water interfacial properties is experimental analyzed, where an obvious trend of interface tension decrement is found. Meanwhile, the dispersed modality (drop size distribution) of a stirring oil-water system is discussed at various temperatures below WAT with a fixed cooling rate. Further, based on the mechanism of molecular diffusion and gelatinization, a kinetic model for wax deposition is established including the effect of dispersed modality on gelation and bulk phase properties.

\* Support from the National Natural Science Foundation of China(51422406) is gratefully acknowledged.

#### OPTIMAL FILTER WIDTH FOR CONTINUUM MODELING OF DISCRETE PHASES

Samuel Bateman, US Naval Research Laboratory, USA; Julian Simeonov, US Naval Research Laboratory, USA; Joseph Calantoni, US Naval Research Laboratory, USA

In geophysical modeling, it is typical to explicitly resolve only the large energy-containing scales and to parameterize the unresolved small scales. One approach to separate the scales is by means of spatial filters and here we discuss practical considerations regarding the choice of a volume averaging scale  $L$ . We use a macroscopically homogeneous scalar field and quantify the smoothness of the filtered field using a noise metric defined by the standard deviation of the filtered field normalized by the domain-averaged value of the scalar field. We find that  $\epsilon$  follows an inverse power law dependence on  $L$  with an exponent and coefficients proportional to the domain-averaged scalar field. The empirical relationship provides a systematic way to choose a volume averaging scale while simultaneously estimating the uncertainty of the volume-averaged continuum field. A qualitative explanation for the empirical power law is provided by assuming that the discrete phase can be described as a Gaussian random medium. Example noise laws are obtained for the concentration and stress fields of granular states produced in discrete element method (DEM) simulations and for salt fingers convection in Direct Numerical Simulations.

\*Support provided under base funding to the Naval Research Laboratory from the Office of Naval Research and in part by a grant of computer time from the DoD High Performance Computing Modernization Program at the ERDC MSRC.

#### NUMERICAL SIMULATION OF ANNULAR FLOW IN HORIZONTAL PIPES USING THE TWO FLUID MODEL

Izabel Souto, PUC-Rio, Brazil; Angela Nieckele, PUC-Rio, Brazil

Annular two-phase flow is characterized by the formation of a liquid layer spread around the pipe circumference with gas flowing in the core area of the pipe. The present work consists in the numerical simulation of an annular flow in horizontal pipe, with and without heat transfer through a one-dimensional code based on the Two Fluid Model. Two pairs of fluids are considered, being the first air-water, which is widely studied in the literature and a typical natural gas and oil fluid from production oil fields. Characteristics parameters of annular flow such as pressure drop, interface friction factor and liquid film height are obtained and compared with experimental and numerical data, showing in both cases good agreement. The natural gas is modeled as real gas, using the Peng-Robinson Equation of State, and compared with the ideal gas modeling. For the typical fluid selected, this effect is quite small on the hydrodynamics parameters such as phases' velocities and pressure drop and on the thermal parameters such as temperature, heat loss for the environment and heat exchange coefficient.

#### THE NUMERICAL MODELLING OF MULTIPHASE FLOW INSTABILITIES IN 3D CYLINDRICAL COORDINATES

Guido Oud, Delft University of Technology, The Netherlands

We have devised a numerical approach to the simulation of two-phase incompressible flow in 3D cylindrical geometries. The cylindrical coordinate system is chosen because fast and highly efficient solvers for multiphase flows have become available recently, which is essential with turbulent ambitions lying ahead. The cylindrical coordinate system comes with a number of challenges in the modelling of both the flow and the interface near the polar axis, and we show solutions to these problems. A combined Level Set and Volume of Fluid approach is used to represent the interface, together with an extension to use the fluid moments as well. The method is first tested on axisymmetric problems for validation, after which we move to the simulation of Kelvin-Helmholtz waves in full 3D.

#### MULTIPHASE SIMULATION FOR WATER COOLING TURBINE EXHAUST

Minyee Jiang, US Navy, USA; David Vinh, US Navy, USA

Handling high temperature turbine engine exhaust has always been a key issue when designing both commercial and military vessels. Hot and toxic exhaust gas is carried by ducts from the ship engines located in the bottom of the vessel, often through passenger areas, to the discharge point at the top and stern of the vessel. The presence of the engine exhaust duct within the vessel presents problems of heat transfer and safety. For military ships, the high temperature exhaust can damage topside antennas which are critical to ship combat operation. These concerns have led to studying ways to reduce the temperature of the exhaust gas as it exits the exhaust duct. One of these methods is the water injection method, which was investigated using multiphase simulation. A notional geometry of the water spray system has been modeled to simulate water mist injection. The detail of the transformation of water mist to vapor after the water mist is mixed with hot exhaust gas will be addressed. Commercial software STAR-CCM+, developed by CD-Adapco, was used for this simulation. STAR-CCM+ is capable of solving flow mixing problem with multiphase (liquid and vapor forms), and multi-species (engine exhaust gas and ambient air mixing) components. Visualization plots show the mixing of the exhaust gas and the cooling water mist. The phase change of water mist to vapor will be discussed.

## A LAGRANGIAN MULTISTAGE SPRAY DRYING MODEL FOR MULTIPHASE CFD APPLICATIONS

Matjaz Hribersek, University of Maribor, Slovenia; Gregor Sagadin, Silkem d.o.o., Slovenia; Matej Zadravec, University of Maribor, Slovenia; Jure Ravnik, University of Maribor, Slovenia

In spray drying, advanced computational models have to be able to account for conditions at the particle level in the fluid as well as for conditions inside the particle. In the framework of the Lagrange-Euler multiphase CFD models the spray drying models typically rely on the first drying stage only, which can lead to nonrealistic computational results. In the present work, the single stage spray drying model is improved by adding the two additional stages. The stage two covers modeling of heat and mass transfer in the porous interior of the particle, whereas the third stage accounts for the removal of adsorbed moisture in the solid phase. As the advanced spray drying model targets at drying material with significant adsorption characteristics, the drying kinetics model of third stage is introduced based on thermogravimetry analysis of the drying material. On the fluid side, the turbulent fluid flow inside a spray dryer is resolved by the SST-SAS model, enabling the drying conditions of the particle to vary along the particle trajectory. The contribution reports on the most important properties of the model as well as on computational results for the case of laboratory spray dryer of zeolite 4A-water slurry, which comparison with experimental results shows excellent agreement.

## EXPERIMENTAL STUDY AND MECHANISTIC MODELLING OF HEAT TRANSFER IN AIR-WATER SLUG FLOW UNDER COOLING PROCESS

Xin Wang, China University of Petroleum, China; Chuanshuai Dong, The Hong Kong Polytechnic University, China; Xiaoling Zhang, China University of Petroleum, China; Xiaoqing Yue, China University of Petroleum, China; Zhaoxing Wang, China University of Petroleum, China; Limin He, China University of Petroleum, China

The offshore petroleum industry in deep water is dependent on the flowlines and pipelines located on the sea bed to transport the natural gas and crude oil safely and economically. The fluids from subsea wells usually with high temperature will be cooled by the cold water and the gas hydrate or wax deposition blockages might occur leading to field shut down. Knowledge of heat transfer in cooling process of slug flow commonly encountered in the oil-gas pipeline is thus crucial for offshore oil and gas production. In this experimental study heat transfer of two-phase slug flow in cooling process in horizontal pipe is investigated. The coolant temperature at the cool side was equal to that in the deep water sea bed. The effect of liquid and gas superficial velocities, pressure drop, length and holdup of liquid slug and film, liquid slug frequency on the convective heat transfer coefficient of slug flow is discussed. In addition, the temperature difference between the top and the bottom of the pipe due to the intermittent flow is presented. A mechanistic procedure based on slug tracking model to predict the convective heat transfer coefficient is also presented. Model results are shown to be in good agreement with measurements.

\* Support from National Natural Science Foundation of China (51376197) and Shandong Province Natural Science Foundation (ZR2011EEM029) are gratefully acknowledged.

## A NOVEL VISCO-PLASTICALLY LUBRICATED METHOD FOR CORE-ANNULAR PIPELINE FLOW

Parisa Sarmadi, University of British Columbia, Canada; Ian Frigaard, University of British Columbia, Canada; Sarah Hormozi, Ohio University, USA

We present a novel methodology for efficient transport of heavy oil via a core-annular flow using a visco-plastic (yield stress) fluid. A triple layer structure is proposed in which the visco-plastic fluid acts as an intermediate skin layer. We borrow ideas from visco-plastic lubrication (VPL) flows [1,2] in which the yield stress effectively eliminates interfacial instability, but also adapt conventional core-annular methods in using a thin low viscosity outer layer. A shaped skin layer allows for a lubrication force to balance buoyancy of the core fluid, e.g [3-5]. We present preliminary results using analytical and numerical methods that establish the feasibility of this method and the parameter regimes where stable steady flows at reduced pressure drops can be achieved.

[1] M. Moyers-Gonzalez, I.A. Frigaard, C. Nouar, J. Fluid Mech., 506, pp.117-146, (2004).

[2] S. Hormozi, K. Wielage-Burchard, I.A. Frigaard, J. Fluid Mech., 166 (5-6), pp. 262-278 (2011).

[3] G. Ooms, M. Pourquie & J. Beerens Phys. Fluids, 25, 032102 (2013).

[4] Ooms & P. Poesio, J. Engng. Math. 85, pp 65-81, (2014).

[5] S. Hormozi, G. Dunbrack, I. Frigaard, Phys. Fluids, 26 pp. 093101 (2014).

## THERMOPHORETIC DEPOSITION OF NANOPARTICLES IN A 90° SQUARE BEND TUBE

Zhaoqin Yin, China Jiliang University, China; Zhongping Dai, China Jiliang University, China

Nanoparticles dispersion, distribution and deposition in gas bend flows play a critical role in many industrial, environmental and biomedical applications.

It has been known that thermophoresis and coherent structures are the important mechanisms leading particle deposition on the surface of a tube. The purpose of this paper is to study the nanoparticles dispersion and deposition on the walls of a 90° square bend tube with turbulent flow and different temperature concerning the effects of particle size and flow Dean number.

The results show that particle migration easily from hotter to colder parts in the bend. Particles deposition process depends strongly on the size and bend's curvature.

\* Support from grant National Natural Science Foundation (11402259) of China is gratefully acknowledged.

## RHEOLOGICAL EFFECTS ON FLUID INVASION INTO CEMENTED ANNULI

Marjan Zare, University of British Columbia, Canada; Matthew Ward, University of British Columbia, Canada; Lindfield Roberts Quentin, University of British Columbia, Canada; Alba Kamran, University of British Columbia, Canada; Ian Frigaard, University of British Columbia, Canada

Annular gas migration in primary cementing operations involves formation gas (or other reservoir fluids) invading into the annular space, formed between the steel casing and borehole, before the cement slurry can hydrate and harden. Pressure imbalance is one of the root causes of fluid invasion. With a purely viscous fluid in the annulus a pressure over-balance is sufficient to ensure that formation fluid will enter the annulus. However, the cement slurry is a yield stress suspension and further develops a gel strength during hydration. For controlling gas migration, setting a target time for the cement slurry to develop 500 lbf/100ft<sup>2</sup> value of gel strength and usage of various additives to influence gel strength development, suggests a common belief that fluid rheology (particularly a yield stress/gel strength) has an important role to play in gas migration.

We study experimentally the process of one fluid invading into a yield stress fluid column, through a single pore/hole, driven by a pressure imbalance. We capture the instant at which fluid invades, i.e. the critical invasion pressure, and understand how this is affected by the rheology of the invaded fluid. Post invasion we observe beautiful fingering patterns develop as the invading liquid propagates. We also present results of a simple modeling study.

## NUMERICAL SIMULATION OF INERTIAL MIGRATION OF ELASTIC PARTICLE IN POISEUILLE FLOWS

Xukun He, Xi'an Jiaotong University, China; Liang-Liang Fan, Xi'an Jiaotong University, China; Jiang Zhe, University of Akron, USA; Liang Zhao, Xi'an Jiaotong University, China

The inertial migration of an elastic particle in Poiseuille flow was numerically studied using front tracking method in this paper. The influence of the lateral migration velocity, slip velocity, and the deformation angle of the particle on inertial migration was investigated by varying the lateral position, Reynolds number (Re), particle-to-channel size ratio ( $\lambda$ ) and the particle deformability (shell stretching and bending coefficient). The results showed the deformation of the particle increases with the decrease of Re,  $\lambda$ . The relationship between Re, particle deformability and the equilibrium position was obtained as follows: the equilibrium position was determined by balance between the wall effect and the shear gradient effect. The equilibrium position shift closer to the wall with the increase of Re. Besides, the equilibrium position also depended on the particle deformability because the lift was related with particle deformability. These results will be useful for better understanding of dynamic behaviors of elastic particle such as cell and phytoplankton, and can be applied for the next generation of microfluidic cell separation chips.

\* Support from National Natural Science Foundation of China (No.: 51476127 and 51128601)

#### EXPERIMENTAL STUDY ON THE COALESCENCE BEHAVIORS UNDER VARIOUS SURFACTANT CONCENTRATIONS BETWEEN DROP-INTERFACE

Mengyu Ma, China University of Petroleum, Beijing, China; Kai Li, Institute of Mechanics, Chinese Academy of Sciences, China; Victor Voulgaropoulos, University College London, UK; Hang Jin, China University of Petroleum, Beijing, China; Wei Wang, China University of Petroleum Beijing, China; Jing Gong, China University of Petroleum Beijing, China

Droplet coalescence phenomenon has long been the topic related to dispersion dynamics and emulsion stability. This work is aimed at experimentally studying the behavior of droplet coalescence with interface in the presence of surfactants. By performing high-speed imaging and particle image shadowgraphy, the coalescence cascade of droplets are recorded and local flow field is obtained. Results of systems containing various concentrations of surfactant are compared and the role of surfactants on droplet coalescence is discussed. In the experiments, it was initially found that, as surfactant concentration increases, droplet coalescence patterns vary from partial coalescence to total coalescence. Further investigation of the behavior was carried out by the comparison of flow field which manifests that the coalescence process is controlled by the competition between the horizontal and vertical rates of collapse. The ratio of necking velocity to descent velocity of droplet under different surfactant concentrations is also calculated.

\* Support from the Henry Fok Foundation (142021) is gratefully acknowledged.

#### NUMERICAL INVESTIGATION OF HEAT TRANSFER AND FLUID FLOW BEHAVIOR AROUND AN OBLATE SPHEROID

Intissar Razgallah, Université de Lorraine, LEMTA, UMR 7563, France; Rafik Ouchene, Université de Lorraine - LEMTA, France; Mohammed Khaliq, Université de Lorraine - LEMTA, France; Boris Arcen, Université de Lorraine - LRGP, France; Anne Taniere, Université de Lorraine - LEMTA, France

Whereas the dynamical behavior of the non-spherical particles become an attractive subject for researchers, the thermal part does not receive the same success yet. This is the objective of the present work. The heat transfer of oblate spheroids, with aspect ratios of 0.2, 0.5 0.8 and 1, is numerically investigated in a steady incompressible flow. Different Prandtl numbers [0.7; 1; 7] are examined as well as particle Reynolds numbers ranging from 0.1 to 100. The temperature of the spheroid is considered as a constant. The uncoupled momentum and heat balance equations were solved numerically for particular corresponding to a laminar axisymmetric regime. The validation is carried out through a comparison of results of the drag coefficient, the local and the average Nusselt Numbers, with some literature results. Next, complementary simulations results are presented for different angles of attack and different axis ratio of oblate spheroid. They show the influence of these two parameters on the heat transfer rate

#### DRIFT AND MIXING BY AIR BUBBLES CROSSING AN INTERFACE OF A STRATIFIED MEDIUM AT MODERATE RE

Lamberto Diaz, Universidad Nacional Autonoma de Mexico, Mexico; Angel Ruiz Angulo, Universidad Nacional Autonoma de Mexico, Mexico; Roberto Zenit, Universidad Nacional Autonoma de Mexico, Mexico

The rising motion of a bubble across two different-density stagnant Newtonian miscible liquids has been experimentally investigated. When a bubble crosses interface, it drags some amount of the dense fluid to the lighter liquid. We distinguish two cases: (i) for small bubbles, rising in a straight trajectory, the drift volume returns to the bottom liquid after sometime. For this case, an heuristic model to predict the size and evolution of the drift volume was proposed which showed good agreement with experimental results, and (ii) for larger bubbles, as the trajectory transition to zig-zag or spiral, the drift volume becomes unstable and detaches from the bubble leaving a trail of heavier fluid blobs in the lighter phase. This unstable configuration could potentially overturn leading to more effective mixing. The Planar Laser-Induced Fluorescence (PLIF) technique was used to quantify this drift volume and the mixing induced in the flow.

#### NUMERICAL INVESTIGATION OF THE GROUPING BEHAVIOUR WHEN THE PARTICLES ARE COAXIALLY SETTLING

Deming Nie, China Jiliang University, China; Rongqian Chen, China Jiliang University, China

The well-known drafting-kissing-tumbling (DKT) motion could take place when multiple particles are sedimenting at finite Reynolds numbers, resulting from hydrodynamic interactions. However, if the particles are placed coaxially and released under only the influence of gravity at finite Reynolds numbers, it will take a very long time to observe the DKT motion. Our work shows that, before the onset of DKT motion, the settling particles may display different grouping behaviors. In other words, the settling particles may be separated into several groups. Each group is sedimenting as a whole with the same velocity. In this paper we studied the dependence of the grouping behavior on the number of particles,  $n$ , the initial inter-particle separation,  $h_0$ , and the Reynolds number,  $Re$ . In particular, we found that the mode of grouping is found to be independent of the number of particles when the Reynolds numbers is small. The two lowermost particles always come together first and form a vertical doublet and then the next two lowest particles form another doublet, and so on. Therefore, we can observe  $n/2$  doublets or  $(n-1)/2$  doublets when  $n$  is even or odd, respectively. The uppermost particle is always left behind when  $n$  is odd.

\* This work is supported by the National Natural Science Foundation of China (No. 11272302).

#### ABILITY OF ANSYS FLUENT® V 16.0 TO COMPUTE THE DRAG FORCE ACTING ON NON-SPHERICAL PARTICLES LIKE CYLINDRICAL DISKS

Jorge Sierra Del Rio, Instituto Tecnológico Metropolitano-ITM, Colombia; Anne Taniere, Université de Lorraine - LEMTA, France; Mohammed Khaliq, Université de Lorraine - LEMTA, France; Diego Hincapié, Instituto Tecnológico Metropolitano-ITM, Colombia; Sebastian Velez, Instituto Tecnológico Metropolitano-ITM, Colombia

This paper present a numerical simulation of the drag force for non-spherical particles using the commercial software ANSYS Fluent® V16.0. Drag force coefficient were obtained for non-spherical particle like a cylindrical disk with a sphericity factor of 0.23, particle orientation at 90° relative to the flow direction and particle Reynolds number between 1 and 300. In order to validate the numerical results, comparisons with the correlations presented by Haider & Levenspiel (1989) and Hölzer & Sommerfeld (2008) were performed. The maximum error of the Haider & Levenspiel's correlation formula can be up to 10%. Further works will be developed in order to obtain accurately correlation of hydrodynamics forces acting in cylindrical disks in function of the particle orientation and aspect ratio.

#### NUMERICAL ANALYSIS OF MICROBUBBLE DEFORMATION NEAR A HIGH-VISCOUS LIQUID IN ULTRASOUND FIELD

Kengo Mori, Keio University, Japan; Toshihiko Sugiura, Keio University, Japan

An ultrasound-mediated microbubble near a wall boundary can undergo large deformation, generating a microjet and a high-pressure field around the bubble. This phenomenon is expected to be used for sonoporation which is a promising technology for promoting transfer of drugs or genes into cells. This study introduces a model to analyze the dynamics of an ultrasound-mediated microbubble near an interface between a fluid surrounding the bubble and a higher-viscosity fluid. We numerically simulated bubble deformation by solving simultaneously both gas and liquid phases using the CIP method suited for analysis of multi-phase flow. This method may be useful for understanding the mechanism of sonoporation, because it is effective in diminishing the numerical instability caused by discontinuity of physical variables across the phase interface and also, it can handle topological changes of the bubble and the liquid automatically. Our numerical results show that there can appear a high-pressure region in the liquid behind the bubble moving toward the higher-viscosity fluid, causing a jet simultaneously. We further investigated effects of the parameters of ultrasound exposure, microbubble characteristics, and material properties of the higher-viscosity fluid on the above process.

## A PRELIMINARY STUDY ON THE STABILITY OF PARTICLE LADEN JETS THROUGH A FULLY COUPLED CFD-DEM SOLVER

Bernhard Peters, University of Luxembourg, Luxembourg; Gabriele Pozzetti, University of Luxembourg, Luxembourg

Jets are widely used in engineering applications. In material machinery, hydro-transportation systems as well as in chemical industry it is common to deal with a dispersed solid phase interacting with the jet, and therefore creating a so-called slurry-jet or particle-laden jet. The stability of a jet is a key issue for many of these processes, still the underlying physics of this turbulent multiphase flow is highly complicated. Conventional CFD approaches have been proven satisfying for the study of the stability of two-phase jets. When a solid dispersed phase is present in the system, the stability problem gets more complicated and dependent on the solid phase dynamic. A possible solution for the problem is to extend the CFD solver capability through a correct coupling with a DEM solver. In this work a preliminary investigation on the potentialities of this kind of approach is presented and compared with a pure CFD approach. In particular the effect of the presence of differently sized particles in the jet is outlined and the influence of particle properties and concentration is investigated.

Finally some considerations about the computational cost of different methods are proposed. The fluid phases are solved through an Eulerian finite volume (FV) multiphase solver based on the OpenFoam® libraries, and coupled with the XDEM code in order to treat the dispersed phase in a Lagrangian way.

## ON THE SEGREGATION OF PARTICLE SIZE CLASSES IN BIDISPERSE GAS-SOLID FLOW: DEPENDENCE ON GAS-SOLID DRAG AND INFLUENCE ON INTERPHASE ENERGY TRANSFER DYNAMICS

Mohammad Mehrabadi, Iowa State University, USA; Shankar Subramaniam, Iowa State University, USA

Gas-solid flows are encountered in many industrial processes. In most of these applications there is a distribution of solid particle size. This size distribution leads to a complex interplay of gas-particle and particle-particle interactions that in turn give rise to particle mass flux of one particle size class with respect to the others. This mass flux is the key signature of the segregation phenomenon observed in gas-solid suspensions. In this study, we use particle-resolved direct numerical simulation (PR-DNS) of bidisperse gas-solid flow to quantify the mean slip velocity between particle size classes, and then improve the prediction of this mean slip velocity by proposing a new bidisperse gas-particle drag model. We also show analytically how the rate of energy supplied by the mean pressure gradient to overcome the gas-particle drag on each particle size class is partitioned into sources of velocity fluctuations in the fluid phase and each of the particle size classes.

\* Supports from grants DE-FC26-07NT43098 and NSF CBET 1134500 are gratefully acknowledged.

## LARGE-EDDY SIMULATION TO STUDY PARTICLE DEPOSITION IN A CURVED PASSAGE

Nityanand Sinha, University of South Florida, USA; Chaitanya Ghodke, Oregon State University, USA; Ambarish Khot, Cummins Research Technology, India; Sourabh Apte, Oregon State University, USA

Studying particle-laden turbulent flow in a curved passage is of great practical importance in many engineering applications, such as internal cooling of gas turbine blades, turbochargers, aerosol transport and so on. Understanding flow physics behind inertial deposition of particles in turbulent flow environment and interactions with unsteady Dean vortices is critical and remains a challenge. To gain deeper understanding of these flows, wall-resolved Large Eddy Simulations (LES) of particle-laden turbulent flow in a U-bend passage at bulk Reynolds numbers 10000 and 20000, based on channel height are performed, where particulate phase is simulated using Lagrangian Discrete-Element approach. The influence of secondary flow in terms of oscillating Dean vortices and unstable shear layers in a curved passage on particle trajectories is investigated. Time-averaged distributions of velocity, Reynolds stresses, along with particle deposition patterns are reported. Effectiveness of Reynolds-Averaged Navier-Stokes (RANS) based models in predicting overall flow pattern is also studied.

## NANOPARTICLE COAGULATION AND DISPERSION IN A TURBULENT PLANAR JET WITH CONSTRAINTS

Chengxu Tu, China Jiliang University, China; Song Liu, Zhejiang University, China; Fubing Bao, China Jiliang University, China; Zhaoqin Yin, China Jiliang University, China

Numerical simulations of coagulating and dispersing nanoparticles in an incompressible turbulent planar jet with constraints are performed. The evolution of nanoparticle field is obtained by utilizing a moment method to approximate the particle's general dynamic equation. The spatio-temporal evolution of the first three moments along with the mean particle diameter and geometric standard deviation of particle diameter are discussed. The distribution for total particle number concentration is affected by particle coagulation and dispersion as well as the flow entrainment, and decreases along stream directions. The total particle mass decreases continuously along stream direction, and the area of the particles increases. The total particle mass is obviously affected by coherent vortices. The mean particle diameter increases along stream direction. The variation of mean particle diameter is dependent on particle coagulation, the largest particles are found within the center region of the jet.

\* This work was supported by the Doctoral Program of Higher Education in China (Grant No.20120101110121) and Special major science and technology of Zhejiang Province (Grant No. 2012C11015-3)

## ANALYSIS OF THE EFFECT OF PARTICLE-WALL AND INTER-PARTICLE COLLISION ON EROSION USING OPENFOAM®

Guilherme A. Novelletto Ricardo, Martin-Luther-University Halle-Wittenberg, Germany; Oscar Sgrott Junior, Martin-Luther-University Halle-Wittenberg, Germany; Santiago Lain, Universidad Autónoma de Occidente, Colombia; Martin Sommerfeld, Martin-Luther-University Halle-Wittenberg, Germany

Particle-laden flows are present in several industrial processes such as pneumatic conveying, cyclones, fluidized beds, and risers. Erosion is one of the main industrial issues related to particle-laden flows. The impact of solid particles on the wall leads to partial or complete deterioration, resulting in high costs due to maintenance or replacement of equipment. This work examines the influence of the wall roughness and inter-particle collisions on the erosion caused by the impact of spherical particles in a pneumatic conveying system using OpenFOAM®. For that purpose the gas-solid flow is calculated by the Euler/Lagrange approach with two-way coupling. Turbulence is modelled by means of the k-omega-SST model also accounting for two-way coupling. Transient numerical simulations are performed considering a 5 m horizontal pipe, a bend and a 5 m vertical pipe. The average conveying velocity is 27 m/s and the pipe diameter 150 mm. For particle tracking all relevant fluid forces are considered as well as dispersion of particles due to turbulence. Several erosion models available in the literature are tested and the results are compared with two-phase flow measurements (Huber & Sommerfeld, 1994, 1998) and published erosion measurements. Since erosion is accompanied by modifications of the surface roughness structure, special emphasis will be put on variations of the respective wall collision parameters. Comparison will be also made with results obtained by the in-house code FASTEST/Lag3D.

\*Support from CAPES and DAAD is gratefully acknowledged.

## SIMULATION OF SINGLE RISING BUBBLES IN A LINEAR SHEAR FLOW WITH THE COLOR-GRADIENT LATTICE BOLTZMANN METHOD

Phillip Malli, Martin-Luther-University Halle-Wittenberg, Germany; Martin Sommerfeld, Martin-Luther-University Halle-Wittenberg, Germany

Simulating two-fluid flows has always been a great challenge for computational fluid dynamics and since the Lattice Boltzmann method emerged, various extensions for multiphase flow applications have been proposed. One of these extensions is the Color-Gradient based approach. Because this approach lacked of a proper handling of high density ratios for a long time, most studies on liquid-gas flows were done using the Shan-Chen or the free surface approach. However, recent improvements on the Color-Gradient based method made it possible to handle high density ratios between two fluid phases.

This model for 3-dimensional flows was the basis for a new in-house code implemented in C++. Then simulations of the movement of single bubbles, initially placed at the bottom of a channel with an already developed shear flow, were executed for several variations of Eötvös and Morton number as well as different shear rates. The results were compared to available data and especially correlations in terms of bubble shapes, terminal rise velocities as well as for drag and lift coefficients were developed.

It can be shown that the Color-Gradient based approach is able to deal with single bubbles moving under the influence of a shear flow, leading to correct bubble shapes and drag coefficients for moderate Eötvös and Morton numbers and shear rates.



